Contents

Chapter 23 — Online Content

Articles

- 3D Printed Antennas by Paul Wade, W1GHZ
- 3D Printed Coax-to-Wire Connection Blocks by John Portune, W6NBC
- 3D Printed Fixture Simplifies Ground-Plane Antenna Construction by John Portune, W6NBC
- A No-Special-Tools SMD Desoldering Technique by Wayne Yoshida, KH6WZ
- Introduction to 3D Printing for Hams by John Portune, W6NBC
- Making Your Own Printed Circuit Boards
- Reflow Soldering for the Radio Amateur by Jim Koehler, VE5JP
- Reflow Soldering for the Radio Amateur — Revisited by Jim Koehler, VE5FP
- Repurposing Obsolete Instrument Enclosures by Scott Roleson, KC7CJ
- Soldering Surface-Mount Devices by Dino Papas, KLØS
- Surface Mount Technology — You Can Work With It (Parts 1 – 4) by Sam Ulbing, N4UAU

Chapter 23

Construction Techniques

Home construction of electronics projects and kits can be a fun part of amateur radio. A recent ARRL survey shows that more than half of active hams build electronic projects. There are many kit vendors with products ranging from simple introductory projects all the way to complete transceivers. Becoming familiar with building practices will help you repair and install commercial equipment as well.

Even experienced constructors will find valuable tips in this chapter. It discusses tools and their uses, electronic construction techniques, tells how to turn a schematic into a working circuit, and then summarizes common mechanical construction practices. This chapter's construction and fabrication sections are maintained by Joe Eisenberg, KØNEB, based on material originally written and compiled by Ed Hare, W1RFI, and Jim Duffey, KK6MC. Dale Grover, KD8KYZ, maintains the comprehensive introduction to the use of PCB design and layout software. Material on 3D printing was contributed by John Portune, W6NBC.

23.1 Electronic Shop Safety

Building, repairing, and modifying equipment in home workshops is a longstanding ham radio tradition. In fact, in the early days building your own equipment was the only option available. While times and interests change, home construction of radio equipment and related accessories remains popular and enjoyable. Building your own gear need not be hazardous if you become familiar with the hazards, learn how to perform the necessary functions, and follow some basic safe practices, including the ones listed below and in the section on soldering. Let's start with our own abilities:

Consider your state of mind. Working on projects or troubleshooting (especially where high voltage is present) requires concentration. Don't work when you're tired or distracted. Be realistic about your ability to focus on the job at hand. Put another way, if we aren't highly alert, we should postpone doing hazardous work until we can focus on the hazards.

Think! Pay attention to what you are doing. No list of safety rules can cover all possibilities. Safety is always your responsibility. You must think about what you are doing, how it relates to the tools, and the specific situation at hand. When working with tools, avoid creating situations in which you can be injured or the project damaged if things don't go "just right."

Take your time. If you hurry, not only will you make more mistakes and possibly spoil the appearance of your new equipment, you won't have time to think things through. Always plan ahead. Do not work with shop tools if you can't concentrate on what you are doing — it's not a race! Working when you are tired can also lead to problems, such as misidentification of parts or making poor or erroneous connections. Listening to the regular time notices on broadcast radio while you work will prompt you to take plenty of breaks so you do not work when you are too tired.

Protect yourself. Use of drills, saws, grinders, and other wood- or metal-working equipment can release small fragments that could cause serious eye damage. Always wear safety glasses or goggles when doing work that might present a flying object hazard and that includes soldering, where small bits of molten solder can be flung a surprising distance. If you use hammers, wire-cutters, chisels, and other hand tools, you will also need the protection that safety eyewear offers. Dress appropriately — loose clothing (or even hair) can be caught in exposed rotating equipment such as drill presses. Keep an eye-wash kit near any dangerous chemicals or power tools that can create chips or splinters. Even a small burn or scratch on your eye can develop into a serious problem if not treated promptly.

Don't work alone. Have someone nearby who can help if you get into trouble when working with dangerous equipment, chemicals, or voltages.

Know what to do in an emergency. Despite your best efforts to be careful, accidents may still occur from time to time. Ensure that everyone in your household knows basic first aid procedures and understands how to summon help in an emergency. They should also know where to find and how to safely shut down electrical power in your shack and shop. Get medical help when necessary. Every workshop should contain a good first-aid kit. If you become injured, apply first aid and then seek medical help if you are not sure that you are okay.

What about the equipment and tools involved in shop work? Here are some basic safety considerations that apply to them, as well:

Read instructions and manuals carefully…and follow them. The manufacturers of tools are the most knowledgeable about how to use their products safely. Tap their knowledge by carefully reading all operating instructions and warnings. Avoiding injuries with power tools requires safe tool design as well as proper operation by the user. Keep the instructions in a place where you can refer to them in the future.

Respect safety features of the equipment you work on and use. Never disable any safety feature of any tool. If you do, sooner or later you or someone else will make the mistake the safety feature was designed to prevent.

Keep your shop or work area neat and organized. A messy shop is a dangerous shop. A knife left laying in a drawer can cut someone looking for another tool; a hammer left on top of a shelf can fall down at the worst possible moment; a sharp tool left on a chair can be a dangerous surprise for the weary constructor who sits down.

Keep your tools in good condition. Always take care of your investment. Store tools in a way to prevent damage or use by untrained persons (young children, for example). Keep the cutting edges of saws, chisels, and drill bits sharp. Protect metal surfaces from corrosion. Frequently inspect the cords and plugs of electrical equipment and make any necessary repairs. If you find that your power cord is becoming frayed, repair it right away. One solution is to buy a replacement cord with a molded connector already attached.

Make sure your shop is well ventilated. Paint, solvents, cleaners, or other chemicals can create dangerous fumes. If you feel dizzy, get into fresh air immediately, and seek medical help if you do not recover quickly.

Respect power tools. Power tools are not forgiving. A drill can go through your hand a lot easier than metal. A power saw can remove a finger with ease. Keep away from the business end of power tools. Tuck in your shirt, roll up your sleeves, and remove your tie before using any power tool. If you have long hair, tie it back so it can't become entangled in power equipment.

23.1.1 Chemicals

Chemicals such as cleaners, adhesives, construction materials, and coolants are used every day by amateurs without ill effects. Take the opportunity to become knowledgeable of the hazards associated with these materials and treat them with respect. **Table 23.1** summarizes the uses and hazards of chemicals and other materials used in the ham shack. It includes preventive measures that can minimize risk. For advanced information, the Centers for Disease Prevention and Control maintains an extensive database at **www.cdc.gov**. Meridian Engineering maintains a collection of materials safety information at **www.meridianeng.com**. When in doubt, contact the manufacturer or distributor of the material for safety information, or use an internet search engine by entering the material name and "safety" into the search window.

Poison Control

If you think you have a chemical emergency, call your local poison control center immediately. Dial 1-800-222-1222, dial 911, or use the online tools at **www.poison.org**.

Here are a few key suggestions for safely storing, handling, and using chemicals:

Read the information that accompanies the chemical and follow the manufacturer's recommended safety practices. If you would like more information than is printed on the label, ask for a material safety data sheet (MSDS). Manufacturers of brand-name chemical products usually post an MSDS on their product websites.

Store chemicals properly, away from sunlight and sources of heat. Secure their containers to prevent spills, and so that children and untrained persons will not gain access. Always keep containers labeled so there is no confusion about the contents. It is best to use the container in which the chemical was purchased. If you transfer solvents to other containers such as wash bottles, label the new container according to its contents.

Handle chemicals carefully to avoid spills. Clean up any spills or leaks promptly but don't overexpose yourself in the process. Never dispose of chemicals in household sinks or drains. Instead, contact your local waste plant operator, transfer station, or fire department to determine the proper disposal procedures for your area. Many communities have household hazardous waste collection programs. Of course, the best solution is to only buy the amount of chemical that you will need, and use it all if possible. Always label any waste chemicals, especially if they are no longer in their original containers. Oil-filled capacitors and transformers were once commonly filled with oil containing PCBs. Never dispose of any such items that may contain PCBs in landfills. Contact your county or city recycling office or local electric utility for information on proper disposal.

Always use recommended personal protective equipment (such as gloves, face shield, splash goggles, and aprons). If corrosives (acids or caustics) are splashed on you *immediately* rinse with cold water for a minimum of 15 minutes to flush the skin thoroughly. If splashed in the eyes, direct a gentle stream of cold water into the eyes for at least 15 minutes. Gently lift the eyelids so trapped liquids can be flushed completely. Start flushing before removing contaminated clothing. Seek professional medical assistance. If using hazardous chemicals, it is unwise to work alone since people splashed with chemicals need the calm influence of another person.

Food and chemicals don't mix. Keep food, drinks, and cigarettes *away* from areas where chemicals are used, and don't bring chemicals to places where you eat.

23.2 AC and Power Connectors

The following tables show the most common connectors for ac power in the United States, along with wiring color codes and maximum current-carrying capacity. Pay particular attention to whether the diagram shows the plug or receptacle and the internal (wiring) side or external side of the connector. If you are unsure of how to connect the wiring, ask for professional help to avoid a shock or fire hazard.

Table 23.1A

Common US AC Power Receptacles

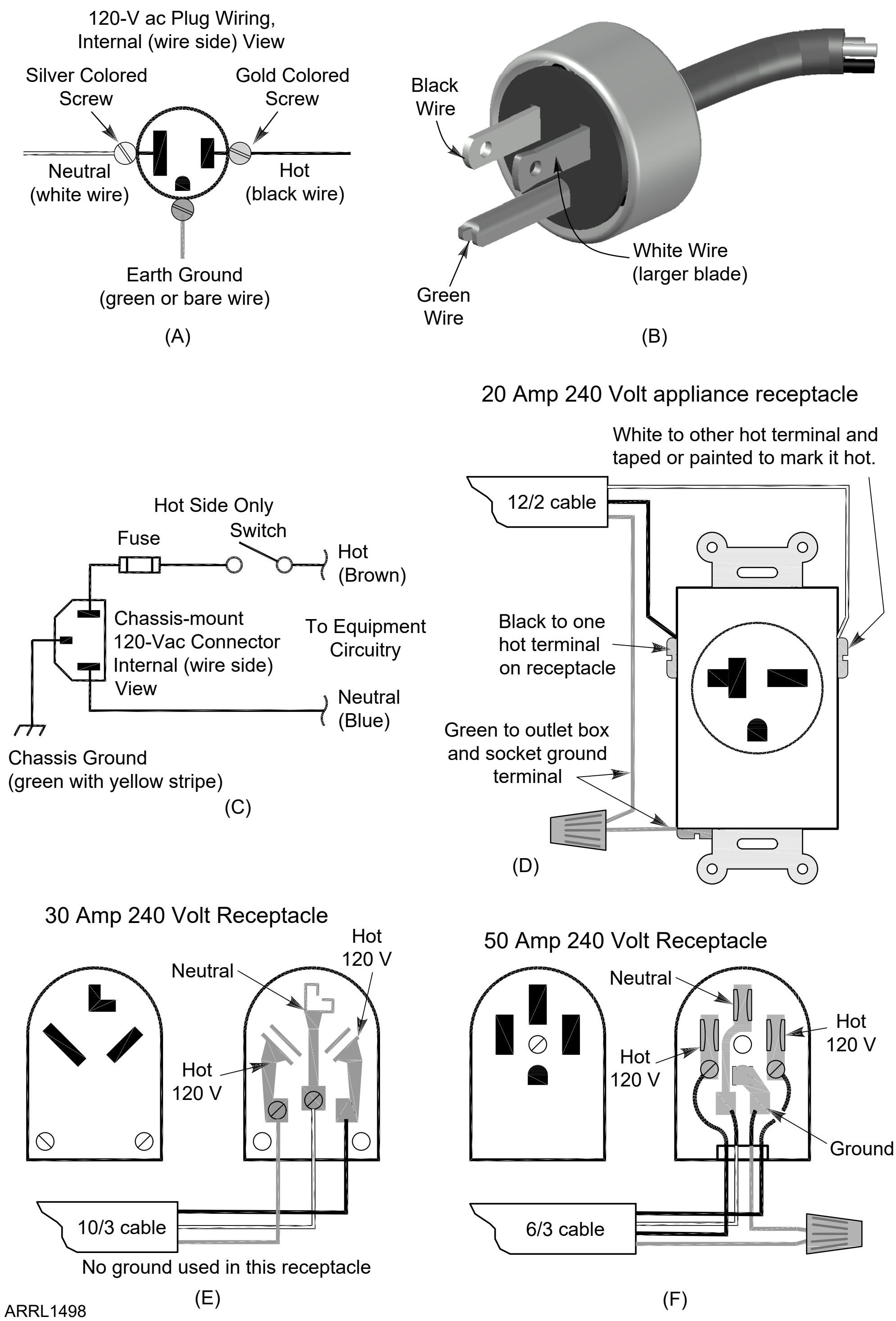


Table 23.1B

Circuit Wiring for 120 V and 240 V Connectors

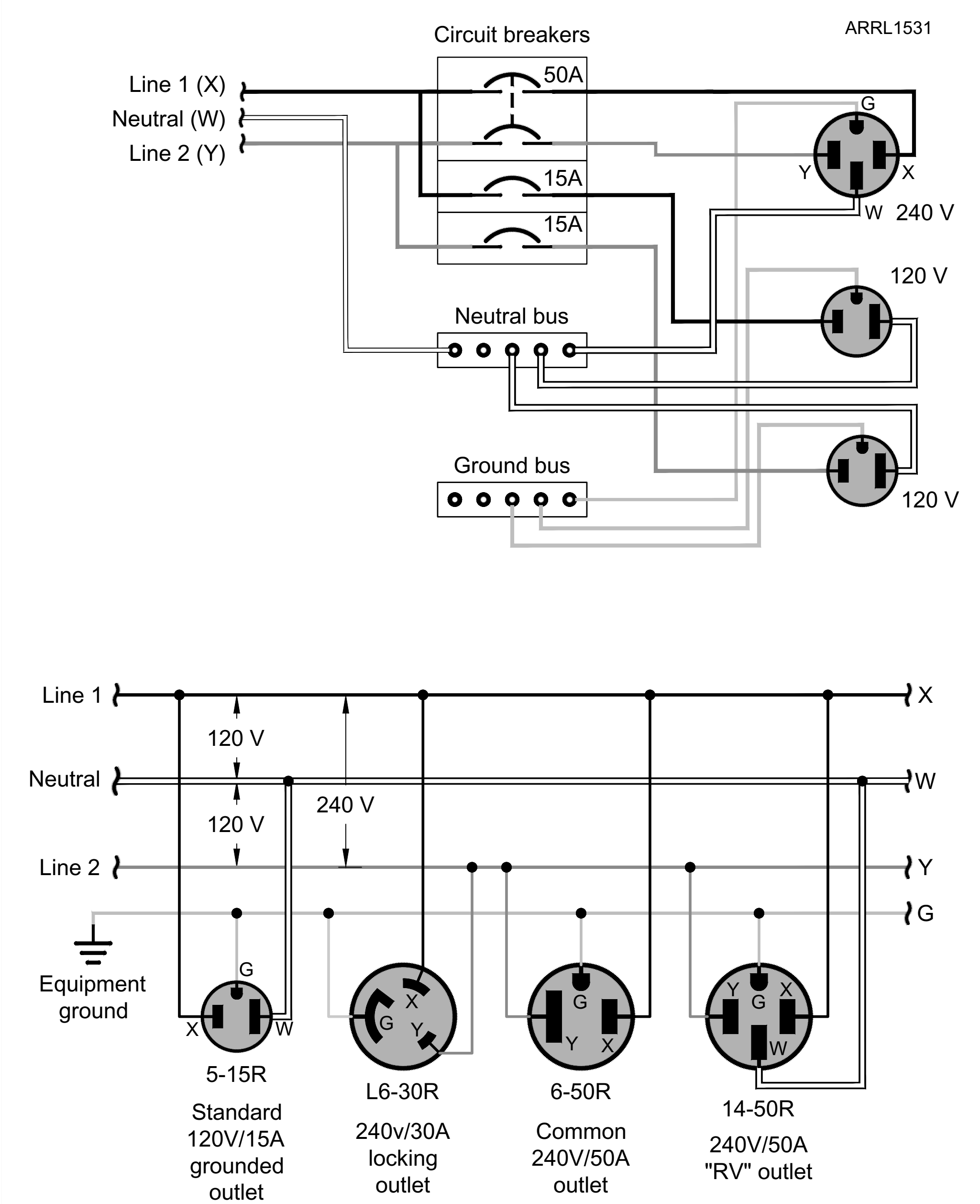
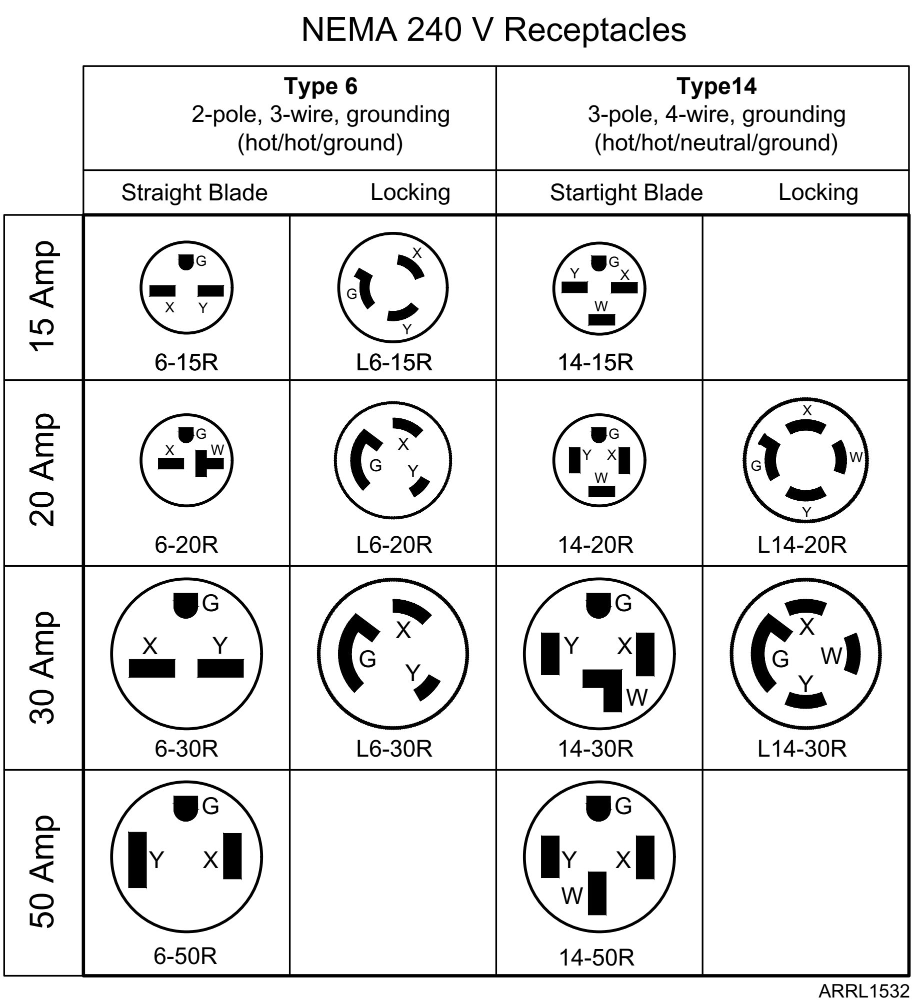


Table 23.1 C

NEMA 240 V Receptacles



23.3 Soldering Tools and Techniques

23.3.1 Soldering Tools

A soldering tool — we'll use the general term "iron" unless discussing a specific type of tool — must be hot enough to do the job and lightweight enough for agility and comfort. An assortment of soldering irons will help you do a wide variety of soldering tasks correctly. The powerful soldering tool you need for assembling wire antennas is too large for PC board work, for example, while a small fine-point tool works well for smaller jobs. The key is to be able to deliver the right amount of heat to do the job: Too much can damage the workpiece and too little can result in poor connections.

If you buy an iron for use on circuits that contain static-sensitive components, get one that has a grounded tip and use it on a static-dissipating workbench mat. Otherwise, you risk damage from static discharge. AC leakage current can also damage components. These irons are usually specified as having a grounded tip and will have a three-prong ac power plug. It is usually not necessary to have a grounded tip on a high-power iron or soldering gun, as those are not normally used on sensitive components.

For PCB soldering, the ideal temperature is around 600 °F when using either 60/40 or 63/37 leaded solder and about 700 °F for lead-free solder. For use with solder wick when desoldering, a higher temperature setting may be necessary, depending on the size of connection being desoldered. These temperatures enable rapid melting of solder and heating of the component leads as well as the PCB pads or terminals being soldered. Using excessive heat or applying heat for too long can damage PCB pads, terminals, or the components themselves. A temperature that is too low will result in cold solder joints that don't conduct properly or that fail mechanically. See the Solder and Soldering sections for more discussion.

SOLDERING IRONS

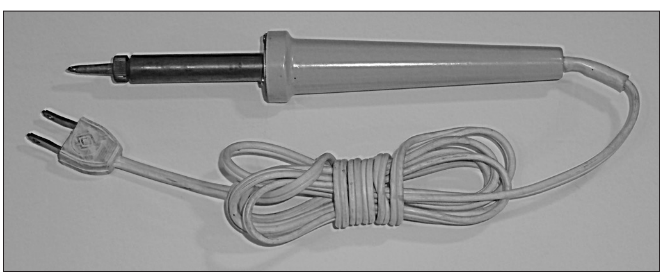
Soldering pencils and *soldering stations* (see **Figure 23.1A** and **B**) are by far the most common soldering tools for PCB and small component soldering. Pencil irons come in a variety of power ratings from 20 W to 40 W. The inexpensive pencil irons provided in soldering training kits will get you started but should be replaced with a higher quality iron for regular soldering of kits and projects.

Some pencil irons are available with a variable temperature control in the handle or on a cord-mounted module. Irons with accurate control of soldering temperature are a must for the best chances of successfully soldering a variety of components. Without accurate control of the temperature, the tips can overheat and corrode, wearing much faster, and they may overheat or underheat your connections.

There are also small battery-powered pencil irons that are useful away from the workbench. Battery-powered irons generally aren't powerful enough for larger connections and take time to recharge. Pencil irons powered by a 12 V dc cigarette lighter plug are useful for working on mobile stations and keeping in a kit for emergencies.

A soldering station is the most preferable soldering tool for most kits and projects. Soldering stations have a base that houses the controls, power supply, and a tip cleaner, and they include a holder to safely place the iron between uses. Some soldering stations even have a holder for a roll of solder. The temperature controls range from a simple knob to calibrated controls that can keep the iron's temperature set within 2 °C. Some soldering stations feature programmable temperature settings, allowing the user to specify different preset heat levels and idle cool-down timing. Features to look for include temperature memories, automatic idle cool-down, and automatic quick heating after idling when the handpiece is lifted for use.

The Metcal RF-powered soldering stations



(A)

(B)

(C)

Figure 23.1 — Types of soldering tools useful in amateur radio — (A) pencil iron; (B) soldering station; (C) soldering gun.

heat the tip with RF for rapid heating, very accurate temperature control, and ease of use. This type of soldering station is most commonly found in commercial manufacturing and product development; used or reconditioned stations are often available. If you do a lot of soldering, one of these might be worth the investment.

A *soldering gun* (**Figure 23.1C**) is used when a smaller soldering iron cannot supply the amount of heat needed for good solder flow. Typical power ratings are 100 to 250 W. Typical jobs for a gun include antenna wires, larger coax connectors, and heavy point-to-point wiring such as for vacuum-tube equipment. Soldering guns consist of an ac transformer with a low-voltage, high-current secondary formed by the copper wire tip. The high current heats the tip. The tips attach to terminals on the front of the soldering gun. Many soldering guns have a high and low power setting controlled by how far the trigger is pulled during operation. Soldering guns usually do not provide heat when not triggered.

Larger ac-powered soldering irons are also available with power ratings of 250 W or more. These irons are most often used for soldering very heavy connections or tubing and sheet metal. This type of iron is used where thorough heating is needed to allow solder to flow fully into the connection. If you plan on soldering a lot of PL-259 coax connectors, a large iron heats the connector body quickly to make a good connection without overheating the cable dielectric. Popular manufacturers include American Beauty, Weller, Hexacon, and others.

Tips for Soldering Irons

Pencil irons and soldering stations are used with a variety of tip shapes and sizes. Several common types are shown in **Figure 23.2**. A *conical* tip comes to a small round point; the *chisel* tip or *mini-chisel* tip is also useful. Low-wattage pencil irons generally have replaceable tips with several sizes and geometries available. For printed circuit board work, a good rule of thumb is to use a tip with a point of the same size as the component leads to be soldered. High-power irons of 100 W and larger usually have a non-interchangeable chisel tip.

Specialized tips are available for use with surface mount technology (SMT) components. These tips can take on the shape of the component being soldered, such as a square tip for working with square IC packages or a special tip for working with SMT resistors and capacitors. When working on SMT assemblies, a hot air gun or reflow oven is another common technique discussed in the section on Surface Mount Technology.

Flame-Powered Soldering

When soldering outdoors and working with larger connections, a more powerful iron is needed. In addition, running a long extension cord outdoors can be a problem. Making antennas from copper tubing requires a lot of heat to allow the solder to flow completely into and through the connection. Soldered connections that are exposed to the weather should use a silver-bearing, high-temperature solder. All of these require a large amount of heat that is best supplied by combustion instead of resistance heating.

Figure 23.2—Typical soldering iron tips (this group is from Weller, weller-tools.com). Conical tips are shown at the upper left and chisel tips at the upper right. Larger connections require the heavier tips.

Temperature and Heat Capacity

If two different tips are both at 600 °F, why is the larger one able to solder a larger connection better than the other? Temperature is a measurement of how fast the molecules in the tip are moving, and that can be the same regardless of tip size. But in the larger tip there are more moving molecules that can transfer their energy to the connection so it has more *heat capacity*. (*Thermal mass* is a similar concept.) Think of temperature as voltage and heat capacity as power. Tip temperature tells you the maximum temperature of the connection. Heat capacity tells you how fast the connection will heat and how big a connection can be heated to the maximum temperature.

In this case, a *butane iron* or *butane torch* is ideal for work outdoors, especially for antenna wire or coaxial connectors. Wind will cool a conventional gun or pencil, but a butane iron can become hotter if the moving air enhances combustion. Butane irons use the same fuel canisters as refillable cigarette lighters. Fueling is quick and can supply heat for a useful amount of time. Most butane irons use a flint-type striker to ignite the fuel once the valve is opened. Butane irons generally come with a variety of tips for soldering and cutting.

For really large jobs, a *propane torch* may be required. Propane torches can be found in most hardware stores and are most commonly used for household or commercial plumbing. A propane torch can deliver plenty of heat in a short amount of time, giving you enough time to flow the solder evenly with no gaps or irregularities in the connection. Be sure to follow the supplied directions carefully with a propane torch, including the correct amount of gas flow and the proper way to ignite the gas. A propane torch's open flame can overheat connectors, melt coaxial cable, and damage plastic components. (Some torches have a soldering style tip, but that limits the amount of heat they can deliver to the connection.) Use them with extreme caution and respect the amount of heat they can deliver.

The two types of gas most often used in larger torches include MAPP gas and propane. MAPP gas burns hotter than propane and is used with high-temperature solder. Propane or MAPP gas can be used for brazing or welding connections, depending on the type of metal involved. Information on brazing can be found at **www.thefabricator.com/thewelder/article/tubepipefabrication/6-steps-to-successful-brazing**.

Finally, for the very large connections used in ground systems and ground rods, *thermite welding* is used. Known as "one-shots" by electrical contractors, they use an exothermic chemical reaction to melt copper wires and ground rods into a solid, welded connection that can be buried. The most common types used by amateurs are made by ERICO ("Cadweld") and Harger ("Uni-Shot").

Maintaining Soldering Tips

Keep soldering iron and gun tips in good condition by keeping them well-tinned with solder. Tinning is performed by melting solder directly onto the tip and letting it form a coating. Do not keep the tips at full temperature for long periods when not in use. Automatic cool-down is a great feature that reduces the need to tin the tip.

After moderate usage, remove the tip and clean off any scale that may have accumulated. There are "tip tinner" products available that can also be used for this function. A tip tinner often comes in a small can and is made up of a mixture of solder, flux, and cleaner. Simply rolling the tip in the tinner makes it look and work like new again.

If a tip becomes oxidized during use, it can be restored to its shiny state by wiping the tip with a damp sponge or rag and then re-tinning. (Some tips are not supposed to be cleaned with water — check the manufacturer's recommendations.) A gentle scraping is also useful for stubborn cases. A copper or stainless-steel coil kitchen scrubber (do not use a scrubber that contains soap) works fine for this. Swipe the iron through the scrubber to clean it. There are coiled brass tip cleaners available that perform at least as well as using a soldering sponge. A brass scrubber or sponge can be placed in a clean tuna can next to the solder station.

If a copper tip becomes pitted after repeated use, file it smooth and bright and then tin it immediately with solder. The solder prevents further oxidation of the tip. Most modern soldering iron tips are nickel or iron clad and should not be filed, as the cladding protects the tip from pitting.

23.3.2 Solder

Solders have different melting points, depending on the ratio of tin to lead. Tin melts at 450 °F and lead at 621 °F. Solder made from 63% tin and 37% lead melts at 361 °F, the lowest melting point for a tin and lead mixture. Called 63-37 (or eutectic), this type of solder also provides the most rapid solid-to-liquid transition and the best stress resistance. 63-37 solder also stays in the plastic state for the briefest time, thus making it ideal for hand soldering because it is a lot less likely to cool unevenly.

Solders made with different lead/tin ratios have a plastic state at some temperatures. If the solder is deformed while it is in the plastic state, the deformation remains when the solder freezes into the solid state. Any stress or motion applied to "plastic solder" causes a poor solder joint.

60-40 solder has the best wetting qualities. Wetting is the ability to spread rapidly, coat the surfaces to be joined, and bond materials uniformly. 60-40 solder also has a low melting point. These factors make it the most commonly used solder in electronics. However, 63-37 solder is the best suited for manual construction techniques.

Some connections that carry high current can't be made with ordinary tin-lead solder because the heat generated by the current would melt the solder. Automotive starter brushes and transmitter tank circuits are two examples. Silver-bearing solders have higher melting points, and so prevent this problem. High-temperature silver alloys become liquid in the 1,100 °F to 1,200 °F range, and a silver-manganese (85-15) alloy requires almost 1,800 °F.

Because silver dissolves easily in tin, tin-bearing solders can leach silver plating from components. This problem can be greatly reduced by partially saturating the tin in the solder with silver or by eliminating the tin. Commercial solders are available which incorporate these features. Tin-silver or tin-lead-silver alloys become liquid at temperatures from 430 °F for 96.5-3.5 (tin-silver), to 588 °F for 1.0-97.5-1.5 (tin-lead-silver). A 15.0-80.0-5.0 alloy of lead-indium-silver melts at 314 °F.

Rosin-core wire-type solder is formed into a tube with a flux compound inside. The resin (usually called "rosin" in solder) in a solder is a *flux*. Flux melts at a lower temperature than solder, so it flows out onto the joint before the solder melts to coat the joint surfaces. The solder used for surface-mount soldering (discussed later) is a cream or paste, and flux, if used, must be added to the joint separately.

Flux removes oxide by suspending it in solution and floating it to the top. Flux is not a cleaning agent! Always clean the surfaces to be soldered before soldering. Rubbing alcohol on a cotton swab is a good cleaning aid. Cleaning an entire board before beginning to install components is easy and prepares the surface for the best soldering. Flux is not a part of a soldered connection — it merely aids the soldering process. Don't touch the board with your fingers after cleaning.

After soldering, remove any remaining flux. Rosin flux can be removed with isopropyl or denatured alcohol. A cotton swab is a good tool for applying the alcohol and scrubbing the excess flux away. Commercial flux-removal sprays are available at most electronic-part distributors. Water-soluble fluxes are also available. Solder is now available with "no-clean" flux, which minimizes the amount of flux left on the board after soldering.

Never use acid flux or acid-core solder for electrical work. They should be used only for plumbing or chassis work. If used on electronics, the flux will corrode and damage the equipment. For circuit construction, only use fluxes or solder-flux combinations that are labeled for electronic soldering.

A basic tutorial on "Soldering 101" including a video demonstration is available from Sparkfun Electronics at **learn.sparkfun.com/tutorials/how-to-solder-through-hole-soldering**.

LEAD-FREE SOLDER

In 2006, the European Union Restriction of Hazardous Substances Directive (RoHS) went into effect. This directive prohibits manufacture and import of consumer electronics that incorporate lead, including the common tin-lead solder used in electronic assembly. California recently enacted a similar RoHS law. As a result of these directives there has been a move to lead-free solders in commercial use. They can contain two or more elements that are not as hazardous as lead, including tin, copper, silver, bismuth, indium, zinc, antimony, and traces of other metals. Two lead-free solders commonly used for electronic work are SnAgCu alloy SAC305 and tin-copper alloy Sn100. SAC305 contains 96.5% tin, 3% silver, and 0.5% copper and melts at 217 °C. Sn100 contains 99.3% tin, and 0.6% copper, as well as traces of nickel and silver and melts at 228 °C. Both of these melting points are higher than the 176 °C melting point of 60-40 and 63-37 lead-bearing solder, but conventional soldering stations will be able to reach the melting points of the new solders easily. Tin-lead solders are still available, but the move away from them by commercial manufacturers will probably lead to the day when they will be unavailable to hams who build their own gear. Be prepared.

The new RoHS solders can be used in much the same manner as conventional solders. The resulting solder joint appears somewhat duller than a conventional solder joint, and the lead-free solders tend to wick higher than the lead-tin solders. Due to the higher heat, it is important that the soldering iron tip be clean, shiny, and freshly tinned so that heat is transferred to the joint as quickly as possible to avoid excess heating of the parts being soldered. The soldering iron should be set to between 700 °F and 800 °F.

Solder and soldering equipment vendors provide numerous guides to hand soldering with lead-free solders. Weller's "Weller University" online presentation **www.elexp.com/media/wysiwyg/cms-pdfs/Weller_Coping_with_Lead_Free.pdf** provides a great deal of detail about how soldering iron tips work with lead-free solder. More information is available from Kester (**www.kester.com**) in the Knowledge Base under the Hand Soldering link.

Most, if not all, RoHS-compliant components can be soldered with lead-tin solder. If the RoHS part has leads that are tinned or coated with an alloy to make soldering easier, it is necessary to use a hotter iron than would normally be required in lead-tin soldering. A soldering iron tip temperature of 315 °C (600 °F) or greater will be adequate for soldering RoHS parts with lead-tin solder. In contrast, a working tip temperature of 275 °C is generally adequate for working with conventional non-RoHS parts.

23.3.3 Soldering

The two key factors in quality soldering are time and temperature. Rapid heating is desired so that all parts of the joint are made hot enough for the solder to remain molten as it flows over the joint surfaces. Most unsuccessful solder jobs fail because insufficient heat has been applied. To achieve rapid heating, the soldering iron tip should be hotter than the melting point of solder and large enough that transferring heat to the cooler joint materials occurs quickly. A tip temperature about 100 °F (60 °C) above the solder melting point is right for mounting components on PC boards.

Use solder that is sized appropriately for the job. As the cross section of the solder decreases, so does the amount of heat required to melt it. Diameters from 0.025 to 0.040 inch are good for nearly all circuit wiring. The most common sizes suitable for amateur projects are 0.031 or 0.025 inch. Sensitive and smaller components can be damaged or surfaces re-oxidized if heat is applied for too long a period. Solder that is too thick can cause shorts between adjacent connections. (A heavier solder is useful for working on antennas, connectors, and light metal work such as shields.)

If you are a beginner, you may want to start with one of the numerous "Learn to Solder" kits available from many electronics parts and kit vendors. The kits come with a printed-circuit board, a basic soldering iron, solder, and the components to complete a simple electronics project. Or, seek out a group kit building session, such as those put on by the QRP ARCI at the annual Dayton Hamvention or by a local or regional group at a hamfest, or specialty convention. Robotics and maker groups often have learn-to-solder sessions too.

You can also download the online article "The Art of Soldering," which provides an overview of guidelines on soldering (**www.arrl.org/files/file/Technology/tis/info/pdf/0102072.pdf**).

Here's how to make a good solder joint. This description assumes that solder with a flux core is used to solder a typical PC board connection such as an IC pin.

• Prepare the joint. Clean all conductors thoroughly with fine steel wool or a plastic scrubbing pad. Clean the circuit board at the beginning of assembly and individual parts such as resistors and capacitors immediately before soldering. Some parts (such as ICs and surface-mount components) cannot be cleaned easily; don't worry unless they're exceptionally dirty.

• Prepare the tool. It should be hot enough to melt solder applied to its tip quickly (half a second when dry, instantly when wet with solder). Apply a little solder directly to the tip so that the surface is shiny. This process is called "tinning" the tool. The solder coating helps conduct heat from the tip to the joint and prevents the tip from oxidizing.

• Place the tip in contact with one side of the joint. If you can place the tip on the underside of the joint, do so. With the tool below the joint, convection helps transfer heat to the joint.

• Place the solder against the joint directly opposite the soldering tool. It should melt within a second for normal PC connections, within two seconds for most other connections. If it takes longer to melt, there is not enough heat for the job at hand. If you have a variable heat soldering iron, adjust it so that the solder flows quickly for the size of wire and joints you are soldering. Much more heat can damage components and the board.

• Keep the tool against the joint until the solder flows freely throughout the joint. When it flows freely, solder tends to form concave joints called "fillets" between the conductors. With insufficient heat solder does not flow freely; it forms convex shapes — blobs. Once the solder shape changes from convex to concave, remove the tool from the joint. If a fillet won't form, the joint may need additional cleaning. Look for that "Hershey's Kiss" shape to know if it is done correctly.

• Let the joint cool without movement at room temperature. It usually takes no more than a few seconds. If the joint is moved before it is cool, it may take on a dull, satin, or grainy appearance that is characteristic of a "cold" solder joint. Reheat cold joints until the solder flows freely and hold them still until cool. Using 63-37 solder will help reduce the time until the solder is solid again.

• When the iron is set aside, or if it loses its shiny appearance, wipe away any dirt with a damp cloth or sponge. If it remains dull after cleaning, tin it again.

• Adafruit has published a good collection of photographs showing many common soldering problems and how to repair them. See **learn.adafruit.com/adafruit-guide-excellent-soldering/common-problems**.

Overheating a transistor or diode while soldering can cause permanent damage, although as you get better at soldering, you'll be able to solder very quickly with little risk to the components. If the soldering iron will be applied for longer than a couple of seconds, use a small heat sink when you solder transistors, diodes, or components with plastic parts that can melt. Grip the component lead with a pair of needle-nose pliers up close to the unit so that the heat is conducted away (be careful — it is easy to damage delicate component leads). A rubber-band wrapped around the pliers handles will hold the pliers on the wire. A small alligator clip or a flat spring type paper clip also makes a good heat sink.

Mechanical stress can damage components. Mount components so there is no appreciable mechanical strain on the leads. Be especially careful with small glass diodes and small disc capacitors, as these components are easy to break when forming the leads.

Installing wire jumpers between connectors or modules and circuit boards can be straightforward, or it can lead to a rat's nest. For some excellent advice and guidance about this basic task, read "W8ZR's Tips for Wiring Circuit Board Jumpers" at **w8zr.net/stationpro/images/download%20files/Tips%20for%20Wiring%20Circuit%20Board%20Jumpers.pdf**. This is a common task in building equipment so why not learn to do it well?

Soldering to hollow pins, such as found on connectors, can be difficult, particularly if the connector has been used previously or has oxidized. Use a suitable small twist drill to clean the inside of the pin and then tin it. While the solder is still melted, clear the surplus solder from each pin with a whipping motion or by blowing through the pin from the inside of the connector. Watch out for flying hot solder — use safety goggles and protect the work surface and your arms and legs! A glass ashtray or small baking dish works great for catching the loose solder. Do not perform this operation near open electronic equipment because the loose solder can easily form short circuits. If the pin surface is plated, file the plating from the pin tip. Then insert the wire and solder it. After soldering, remove excess solder with a file, if necessary.

When soldering to the pins of plastic connectors or other assemblies, heat-sink the pin with needle-nose pliers at the base where it comes in contact with the plastic housing. Do not allow the pin to overheat; it will loosen and become misaligned. If you need to solder very quickly due to concerns about melting the housing, tin both the pin and wire first. Reheat the pin while inserting the wire. This melts the solder on both the pin and the wire, resulting in a solid connection without overheating the pin.

23.3.4 Desoldering

There are times when soldered components need to be removed. The parts may be bad, they may be installed incorrectly, or you may want to remove them for use in another project.

There are several techniques for desoldering. The easiest way is to use a desoldering braid. Desoldering braid is simply fine copper braid, often containing flux. It is available under a wide variety of trade names wherever soldering supplies are sold. A good rule of thumb is to choose a width and thickness of desoldering braid that matches the size of the connection being desoldered.

The soldering braid is placed against the joint to be desoldered. A hot iron is pressed onto the braid. If you have a variable temperature soldering station, you might get better results by turning up the temperature, as the combination of the wick and the connection absorbs more heat. As the solder melts, it is wicked into the braid and away from the joint. Copper is an excellent conductor of heat, and the braid can get quite hot, so watch your fingers when using braid for desoldering. After all the leads have been treated in this manner the part can be removed. (A thin film of solder may remain, but is easily broken loose through the use of needle-nose pliers.) The part of the braid that wicked up the solder is clipped off.

Do not allow the used portion of the braid to get too long, as it will absorb too much heat and not do as good of a job removing solder. When desoldering connections that have been made by using lead-free or "RoHS" solder, it can sometimes be difficult to achieve the high temperature needed to use solder wick, even with the iron heat turned all of the way up. A good tip in this case is to add a small amount of conventional leaded solder to the joint first, allowing it to mix. That lowers the melting point to a level that allows for much easier desoldering using solder wick, or other methods.

A desoldering vacuum pump can also be used. There are two types of desoldering vacuum tools, a simple rubber syringe bulb with a high temperature plastic tip and a desoldering pump. The desoldering pump is a simple manual vacuum pump consisting of a cylinder that contains a spring-loaded plunger attached to a metal rod inside a tip of high temperature plastic on the end of the pump. To desolder a joint, the plunger is pushed down and locked in place. The tip is placed against the joint to be desoldered along with a soldering iron. When the solder melts, a button on the pump releases the plunger, which pulls the rod back, creating a vacuum that sucks the molten solder through the tip. The part being desoldered can then be removed. Pushing the plunger again ejects the solder from the desoldering tip.

The desoldering bulb employs a similar concept: heat the joint to be desoldered, squeeze the bulb, place the tip on the joint and release it to suck up the solder. Remove the part that was desoldered. If the first application of the desoldering pump doesn't suck up all the solder, reheat the joint and suck up the rest.

If the desoldering tool doesn't seem to be sucking up solder, the tip may be clogged with solder. The tip can be unclogged by pushing the solder through. You may have to clear the tip several times when doing a job that requires desoldering many joints. One can purchase small desoldering irons that contain a bulb on the handle that leads to a tip adjacent to the iron tip. This desoldering iron combination is somewhat easier to handle than the separate bulb and iron and does an effective job. Use a glass ashtray for a container to blow out the excess solder from this tool before using it again, as an ashtray is usually designed to handle heat.

Desoldering stations are also available. One type contains a vacuum pump in a console much like a soldering station. A vacuum line is connected to the tip of the soldering iron. There is a valve trigger on the iron that is used to open the tip to the vacuum when the solder is melted. The solder is sucked up the line into a receptacle in the station or in the hand piece.

DESOLDERING SURFACE-MOUNT DEVICES

Special tools are available for desoldering and replacing surface-mount parts. (See the previous section on Tools for Soldering Surface-Mount Devices.) The two most common are "hot tweezers," which is really a special dual-tipped soldering iron that is squeezed to match the size of the component. Hot tweezers can be used to not only remove but to replace surface-mount parts. Another tool for this purpose is a hot-air tool with a small tip that focuses the hot air on a single component. This tool can be used to remove a surface mount part as well as to replace one. It heats the joint with hot air so the component can be lifted off with tweezers or needle-nose pliers.

Use surplus circuit boards to practice soldering and desoldering techniques. Old boards from all kinds of electronic items are often available at very low cost or free at many flea markets. Use these surplus boards to practice different soldering and desoldering techniques without having to worry about damaging a project. Computer boards are often made with lead-free solder, and are ideal for practicing desoldering methods by adding leaded solder to the connections before desoldering.

23.3.5 Soldering Safety

Soldering requires a certain degree of practice and, of course, the right tools. What potential hazards are involved?

Since the solder used for virtually all electronic components is a lead-tin alloy, the first thing in most people's minds is lead, a well-known health hazard. There are two primary ways lead might enter our bodies when soldering: we could breathe lead fumes into our lungs, or we could ingest (swallow) lead or lead-contaminated food. Inhalation of lead fumes is extremely unlikely because the temperatures ordinarily used in electronic soldering are far below those needed to vaporize lead. Nevertheless, since lead is soft and we may tend to handle it with our fingers, contaminating our food is a real possibility. For this reason, wash your hands carefully after any soldering (or touching of solder connections).

Using a small fan can keep the fumes away from your eyes and reduce your exposure to solder smoke. A small computer chipset fan, often only an inch or two wide, can be used. By reducing the voltage that feeds the fan, the speed and noise can be reduced, allowing the fumes to be blown away, without creating another problem with the airflow. Look in old computers for these little fans, often attached to video cards or to the bus chips on the motherboard. CPU or case fans can be also used, but the voltage supplying them will definitely need to be reduced to create the desired level of airflow. There are also commercially available specialized fans with built-in filters designed for this purpose — look for a soldering "fume extractor."

Soldering equipment gets *hot*! Be careful. Treat a soldering iron as you would any other hot object. A soldering iron stand is helpful, preferably one that has a cage that surrounds the hot tip of the iron. Here's a helpful tip — if the soldering iron gets knocked off the bench, train yourself not to grab for it because the chances are good that you'll grab the hot end!

When heated, the flux in solder gives off a vapor in the form of a light gray smoke-like plume. This flux vapor, which often contains aldehydes, is a strong irritant and can cause potentially serious problems to persons who suffer from respiratory sensitivity conditions such as asthma. In most cases it is relatively easy to use a small fan, like the small computer fans described previously, to move the flux vapor away from your eyes and face. Opening a window provides additional air exchange.

Solvents are often used to remove excess flux after the parts have cooled to room temperature. Minimize skin contact with solvents by wearing molded gloves resistant to the solvent. If you use a solvent to remove flux, it is best to use the mildest one that does the job. Isopropyl alcohol, or rubbing alcohol, is often sufficient. You can purchase alcohol ranging from 70% to 92% concentration at local drug stores that works well in removing most types of fluxes. Some water-soluble solder fluxes can be removed with water.

Observe these precautions to protect yourself and others:

- Properly ventilate the work area. If you can smell fumes, you are breathing them.
- Wash your hands after soldering, especially before handling food.
- Minimize direct contact with flux and flux solvents. Wear disposable surgical gloves when handling solvents.

For more information about soldering hazards and the ways to make soldering safer, see "Making Soldering Safer," by Brian P. Bergeron, MD, NU1N (Mar 1991 *QST*, pp. 28–30) and "More on Safer Soldering," by Gary E. Meyers, K9CZB (Aug 1991 *QST*, p. 42).

23.4 Surface Mount Technology (SMT)

Today, nearly all consumer electronic devices are made with surface mount technology. Hams have lagged behind in adopting this technology largely due to the misconception that it is difficult, requires extensive practice, and requires special equipment. In fact, surface mount devices can be soldered easily with commonly available equipment. There is no more practice required to become proficient enough to produce a circuit with surface-mount (SM) devices than there is with soldering through-hole (leaded) components.

There are several advantages to working with SMT:

• Projects are much more compact than if through-hole components are used

• SM parts are available that through-hole packages don't include

• Fewer and fewer through-hole parts are being produced

• Equivalent SM components are often cheaper than through-hole parts, and

• SM parts have less self-inductance, less self-capacitance, and better thermal properties.

There are several techniques that can be used effectively to work with SM components: conventional soldering iron, hot air reflow, and hot plate/hot air reflow. This section describes the soldering iron technique. On-line descriptions of reflow techniques are available for the advanced builder who wants to try them. As is the case for through-hole soldering, kits are available to teach the beginner how to solder SMT components.

The following material on working with surface-mount technology contains excerpts from a series of *QST* articles, "Surface Mount Technology — You Can Work with It! Part 1" by Sam Ulbing, N4UAU, published in the April 1999 issue. (The entire series of four articles is available with the online content. Additional information on SMT is available at **www.arrl.org/surface-mount-technology**.) Additional information and illustrations were contributed by George Heron, N2APB.

23.4.1 Equipment Needed

You do not need lots of expensive equipment to work with SM devices.

• A fundamental piece of equipment for SM work is an illuminated magnifying glass. You can use an inexpensive one with a 5 inch diameter lens, and it's convenient to use the magnifier for all soldering work, not just for SM use. Such magnifiers are widely available from about $25. Most offer a 3× magnification and have a built-in circular light. Be sure to use a light that is fluorescent or LED-based. Incandescent lights are much hotter and make it difficult to ascertain the correct color of component markings.

• A low-power, temperature-controlled soldering iron is necessary. Use a soldering iron with a grounded tip, as most SM parts are CMOS devices and are subject to possible ESD (static) failure.

• 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. Solder paste or cream can also be used.

• A flux pen comes in handy for applying just a little flux at a needed spot.

• Good desoldering braid is necessary to remove excess solder if you get too much on a pad. 0.100 inch wide braid works well.

• ESD protective devices such as wrist straps may be necessary if you live in a dry area and static is a problem.

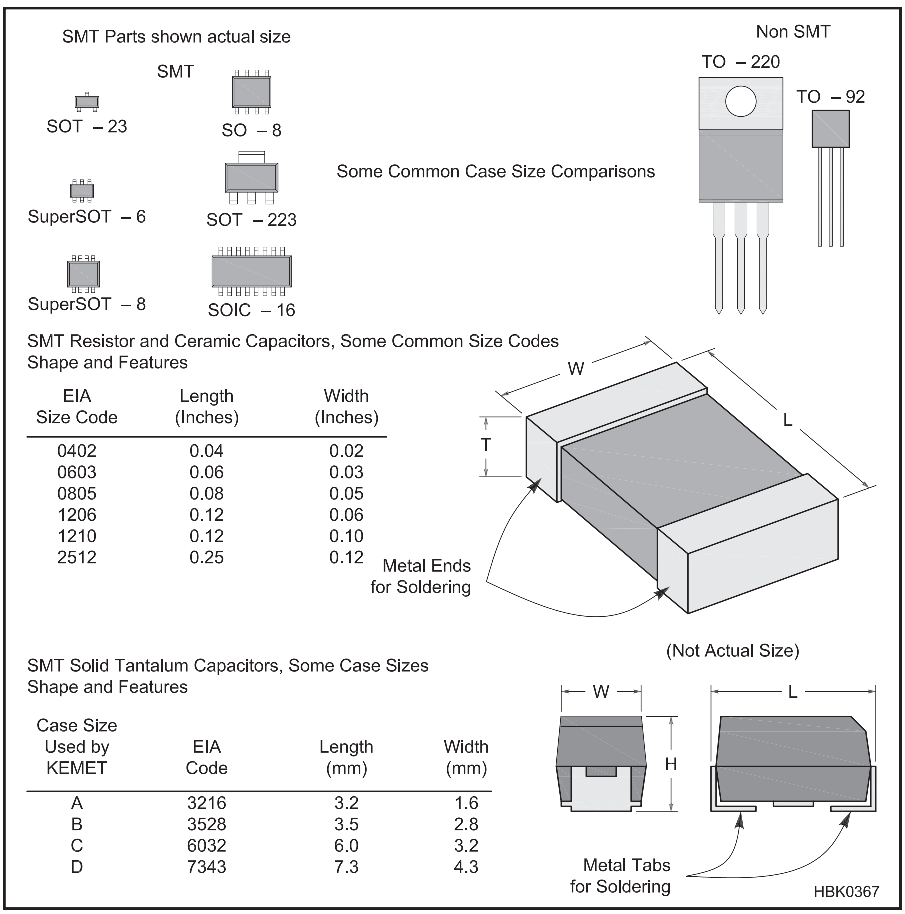
• Tweezers help pick up parts and position them. Nonmagnetic, stainless-steel drafting dividers also work well. The nonmagnetic property of stainless steel means the chip doesn't get attracted to the dividers.

• Some hams prefer to hold components in place with a temporary adhesive such as DAP Blue-Stik while soldering rather than holding the part with tweezers.

23.4.2 Surface-Mount Parts

Figure 23.3 shows some common SM parts. (The **Circuits and Components** chapter has more information on component packages.) 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.

Discrete component packaging has shrunk to 0.12 × 0.06 inches, as shown in the "1206" capacitor in **Figure 23.4** compared to a penny. Even smaller packages are common today, requiring much less PC board area for the same equivalent circuits. Integrated circuit packaging has also been miniaturized to create 10 ×



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 23.3 — Size comparisons of some surface-mount devices and their dimensions. See the Component Data and References chapter for more information about component packages and labeling.

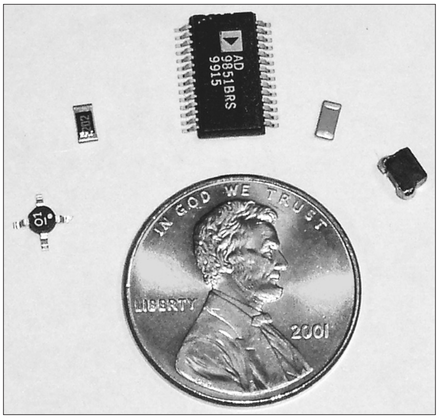


Figure 23.4 — SMT components are small. Clockwise from left: MMIC RF amp, 1206 resistor, SOIC integrated circuit, 1206 capacitor, and ferrite inductor.

Figure 23.5 — A magnifying visor is great for close-up work on a circuit board. These headsets are often available for less than $10 at hamfests, and some even come with superbright LEDs mounted on the side to illuminate the components being soldered.

5 mm SOIC packages with lead separations of 0.025 inch.

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. 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 an SM package.

If you are a beginner, it is probably best to start with the larger sizes — 1206 for resistors and capacitors, SOT-23 for transistors, and SO-8 for ICs. When you get proficient with these parts you can move to smaller ones.

PREPARING FOR SMT WORK

The key to success with any construction project is selecting and using the proper tools. A magnifying lamp is essential for well-lighted, close-up work on the components. **Figure 23.5** shows a convenient magnifying visor. Tweezers or fine-tipped pliers allow you to grab the small chip components with dexterity.

A clean work surface is of paramount importance because SMT components have a tendency to slip from pliers or tweezers and fly off even when held with the utmost care. You'll have the best chance of recovering your wayward part if your table is clear. When the inevitable happens, you'll have lots of trouble finding it if the part falls onto a rug. It's best to have your work area in a room without carpeting, for this reason as well as to protect static-sensitive parts. Using a cookie sheet with a raised lip helps to contain stray SMT parts.

23.4.3 SMT Soldering Basics

If the project contains both SM and conventional through-hole parts and you intend to use a heat-gun or oven for reflow soldering, always mount the SM parts first, as through-hole components are not always designed to handle the higher reflow heat levels. Use junk PC boards to practice your soldering and desoldering techniques for SM parts until you are comfortable beginning your own project.

SMT soldering techniques are covered in the March 2019 *QST* article "Soldering Surface-Mount Devices" by Dino Papas, KLØS. Reflow soldering is explained in the *QST* and *QEX* articles "Reflow Soldering for the Radio Amateur" (Jan 2011 *QST*) and "Reflow Soldering for the Radio Amateur — Revisited" (Mar 2019 *QEX*) by Jim Koehler, VE5FP. All three are included in the online content for this chapter.

USING A SOLDERING IRON

Let's look at how to solder a surface-mount IC with a soldering iron. Use a little solder to pre-tin the PC board pads if the board is not already pre-tinned. 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 (very) 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 to 3.5 seconds.

Place the part on the board and then use dividers (or 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 hold the chip down while tack-soldering 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. If you do it right, when the solder melts it will 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 perform 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. It's 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. The reflow method also works well for these parts.

Attaching wires that connect to points off the board can be a bit of a challenge because even #24 AWG 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. Pre-tin the pad, carefully place the wire on the pad, then heat it with the soldering iron until the solder melts.

USING REFLOW TECHNIQUES

While SMT projects can be built with conventional solder and a fine-point soldering iron, if you move on to reflow techniques you will need to use solder paste. Solder paste is a grayish looking substance made of a blend of flux and solder. A small dot of solder paste is put on the board at each location a component will need to be soldered. The components are then carefully placed on the board, with the paste loosely holding the parts in position. The whole board is then heated in an oven, or the area of the board being assembled is heated with a heat gun. With sufficient heat, the solder paste melts and flows onto the pad and component contacts, then the board is cooled leaving all of the components soldered in place.

A toaster oven can also be modified to perform reflow soldering as described in the article "Reflow Soldering for the Radio Amateur" by Jim Koehler, VE5FP, in the January 2011 issue of *QST* (this article is included with the online content). There are many online tutorials for adapting and using a toaster oven, such as **www.instructables.com/id/Hack-a-Toaster-Oven-for-Reflow-Soldering/**. SparkFun also shows how to add a temperature control to a toaster oven at **www.sparkfun.com/tutorials/60**. If you plan on doing a lot of SMT assembly, learning reflow techniques is well worth the effort.

For occasional SMT use, a heat gun is a better choice. Many hobby/crafts stores sell a special heat tool that is used for melting embossing inks used in scrapbooking. Look for an embossing heat tool in the scrapbooking department of these stores. Do not get the heat gun too close to the board, as the airflow may move your components out of place. Hold the tool steady a couple of inches above the board until you see the solder paste turn silver and the component appears to be soldered in place, then gradually remove the heat gun. Never use a heat gun designed for paint stripping, as the airflow is way too strong for this purpose and will blow the SM parts off the board.

Only a small amount of solder paste is required. Kester Easy Profile 256 is a good solder paste to start with. It is available at reasonable cost in a small syringe with a fine needle-point applicator. Since litte paste is needed for each solder joint, this small amount will last through several medium sized projects. Solder paste must be kept cold or it deteriorates. Kept in a household refrigerator or freezer it has a shelf life of at least a year.

Hot tweezers are a dual-tipped soldering iron used to solder and remove SMT parts. Hot tweezers can make very quick work of removing and replacing most common two- and three-terminal surface mount devices with a lot lower risk of board damage.

23.4.4 Removing SMT Components

The surface-mount ICs used in commercial equipment are not easy for experimenters to replace. They have tiny pins designed for precision PC boards. Sooner or later, however, you may need to replace one. If you do, don't try to get the old IC out in one piece! This will damage the IC beyond use anyway, and will probably damage the PC board in the process.

Although it requires a delicate touch and small tools, it's possible to change a surface mount IC at home. To remove the old one, use small, sharp wire cutters to cut the IC pins flush with the IC. This usually leaves just enough of the old pin to grab with tweezers. Heat the soldered connection with a small iron and use the pliers to gently pull the pin from the PC board. Use desoldering braid to remove excess solder from the pads and remove the flux with rubbing alcohol. Solder in the new component using the techniques discussed above.

You can also use the embossing heat gun previously described to remove SM parts, especially ICs. Keep in mind that not only the desired component, but some adjacent parts as well may be loosened by this process, so be sure to not move the board during reheating to allow the other components to stay in place. Use long-handled tweezers to remove a component once you see the solder become silvery again. Be careful to not disturb any adjacent components. Allow the board to thoroughly cool before moving it to prevent inadvertently allowing other components to shift before the solder solidifies again.

To remove individual components without a heat gun, first remove excess solder from the pads by using desoldering braid. Then the component will generally come loose from the pads if gently lifted with a hobby knife or dental tool. If the component remains attached to the pad, touch the pad with the soldering iron and lift the component off the pad. It may take one or two attempts to free the component from all pads. In extreme cases, it may be necessary to add solder to the pad to completely loosen the component. A product called CHIPQUIK (**www.chipquik.com**) reduces the melting point of the existing solder by mixing in a very low temperature solder. Using the flux and solder that comes with a CHIPQUIK kit lets you desolder an SMD easily without a hot-air tool.

23.4.5 SMT Construction Examples

The first project example is the DDS Daughtercard, a small module that generates precision RF signals for a variety of projects. This kit has become immensely popular in homebrew circles and is supplied with the chip components contained in color-coded packaging that makes an easy job of identifying the little parts, a nice touch by a kit supplier.

Figure 23.7A shows the DDS PC board, a typical layout for SMT components. All traces are on one side, since the component leads are not "through-hole." The little square pads are the places where the 1206 package-style chips will eventually be soldered. This project demonstrates the reflow technique using a soldering iron. **Figure 23.7B** illustrates how to use this technique,

(a) Pre-solder ("tin") one of the pads on the board where the component will ultimately go by placing a small blob of solder there.

(b) Carefully hold the component in place with small needle-nose pliers or sharp tweezers on the tinned pad.

(c) Reheat the tinned pad and component to reflow the solder onto the component lead, thus temporarily holding the component in place.

(d) Solder the other end of the component to its pad.

(e) Finally, check all connections to make sure there are no bridges or shorts.

Figure 23.7C shows the completed DDS board.

The second project example is the K8IQY Audio Amp, a discrete component audio amplifier that is homebrew-constructed "Manhattan-style" as described later in the chapter, in which small pads are glued or soldered to the copper-clad base board wherever you need to attach component leads or wires. See **Figure 23.7D**.

Instead of using little squares or dots of PC board material for pads, you might decide to create isolated connection points by cutting an "island" in the copper using an end mill or pad cutter. No matter how the pads are created, SMT components may be easily soldered from pad-to-pad, or from pad-to-ground plane to build up the circuit. **Figure 23.7E** shows the completed board, combining SMT and leaded components.

Homebrewing with SOIC-packaged integrated circuits is a little trickier and typically requires the use of an "SOIC carrier board," such as the one shown in **Figure 23.7F**, onto which you solder your surface-mount integrated circuit. You can then wire the carrier board onto your base board or whatever you're using to hold your other circuit components.

Full details on the DDS Daughtercard, the K8IQY Islander Audio Amp, and the Islander Pad Cutter may be found online at **www.njqrp.org**.

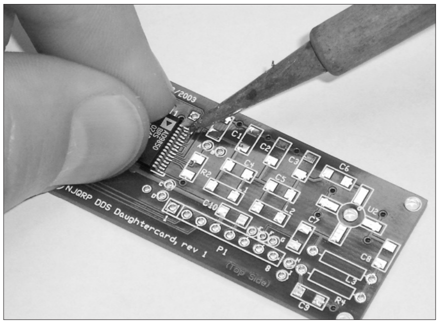


Figure 23.6A — This DDS Daughtercard has all interconnections on the top side. Connections to the ground plane on the backside of the board are made by the use of "vias," wires through the PC board. Pin 28 of the SMT IC is shown being tack-soldered to hold it to the board, keeping all other pins carefully aligned on their pads. Then the other pins are carefully soldered, starting with pin 14 (opposite pin 28). Finally, pin 28 is reheated to ensure a good connection there. If you bridge solder across adjacent pads or pins, use solder wick or a vacuum solder sucker to draw off the excess solder.

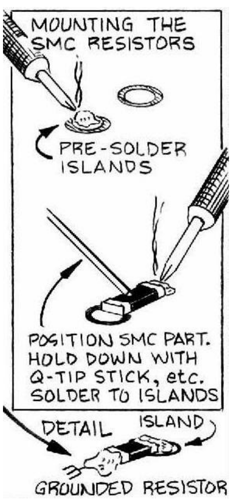


Figure 23.6B — Attaching an SMT part. It is a lot easier attaching capacitors, resistors, and other discrete components compared to multi-pin ICs. Carefully hold the component in place and properly aligned using needle-nose pliers or tweezers and then solder one end of the component. Then reheat the joint while gently pushing down on the component with the pliers or a Q-tip stick to ensure it is lying flat on the board. Finally, solder the other side of the component.

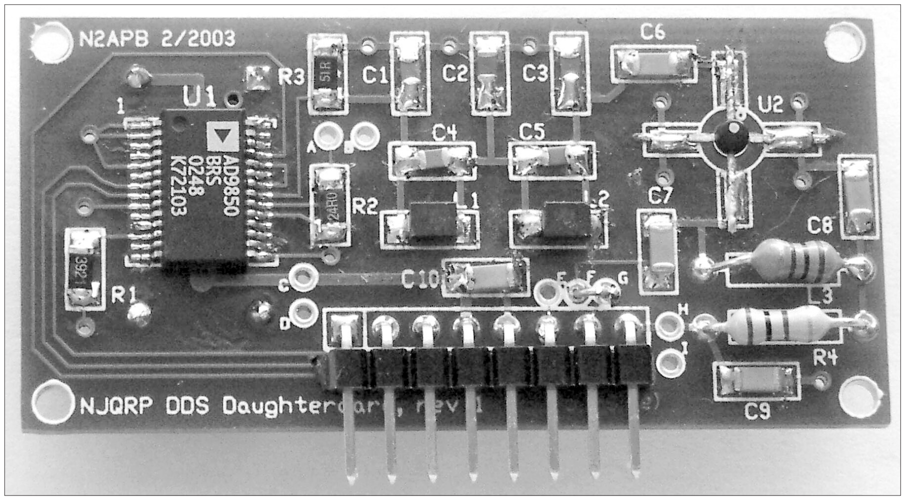


Figure 23.6C — The fully-populated DDS Daughtercard PC board contains a mix of SMT and through-hole parts, showing how both packaging technologies can be used together.

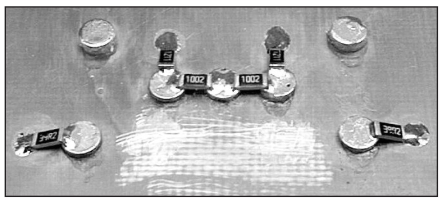


Figure 23.6D — SMT resistors soldered to base board of the Audio Amp in the beginning stages of assembly.

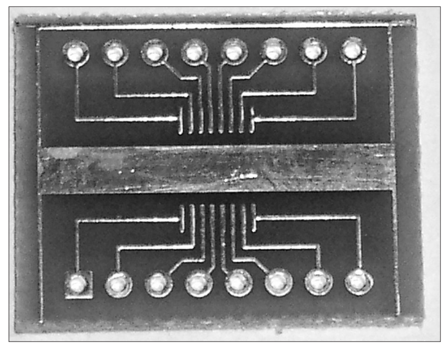


Figure 23.6F — Surface mount ICs can be mounted to general-purpose carrier boards, then attached as a submodule with wires to the base board of the homebrew project.

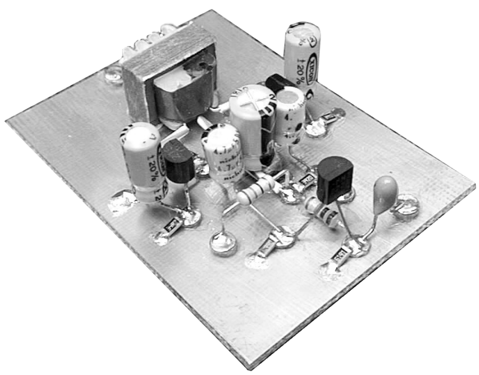


Figure 23.6E — The completed homebrew Audio Amp assembly shows simple, effective use of SMT components used together with conventional leaded components when constructed "Manhattan-style."

23.5 Constructing Electronic Circuits

Most of the construction projects undertaken by the average amateur involve electronic circuitry. The circuit is the "heart" of most amateur equipment. It might seem obvious, but in order for you to build it, the circuit must work! Don't always assume that a "cookbook" circuit that appears in an applications note or electronics magazine is flawless. These are sometimes design examples that have not always been thoroughly debugged. Many home-construction projects are one-time deals; the author has put one together and it worked. In some cases, component tolerances or minor layout changes might make it difficult to get a second unit working. Using a solderless breadboard can make it easier to test this type of circuit design. For RF circuits above a few MHz, a solderless breadboard is not always practical due to long lead lengths that result.

Take steps to protect the electronic and mechanical components you use in circuit construction. Some components can be damaged by rough handling. Dropping a ¼ W resistor causes no harm, but dropping a vacuum tube or other delicate subassemblies usually causes damage.

Some components are easily damaged by heat. Some of the chemicals used to clean electronic components (such as flux removers, degreasers, or control-lubrication sprays) can damage plastic. Check them for suitability before you use them.

23.5.1 Electrostatic Discharge (ESD)

Some components, especially high-impedance components such as FETs and CMOS gates, can be damaged by electrostatic discharge (ESD). Protect these parts from static charges. Most people are familiar with the static charge that builds up when one walks across a carpet and then touches a metal object; the resultant spark can be quite lively. Walking across a carpet on a dry day can generate 35 kV! A worker sitting at a bench can generate voltages up to 6 kV, depending on conditions, such as when relative humidity is less than 20%.

You don't need this much voltage to damage a sensitive electronic component; damage can occur with as little as 30 V. The damage is not always catastrophic. A MOSFET can become noisy, or lose gain; an IC can suffer damage that causes early failure. To prevent this kind of damage, you need to take some precautions.

The energy from a spark can travel inside a piece of equipment to affect internal components. Protection of sensitive electronic components involves the prevention of static build-up together with the removal of any existing charges by dissipating any energy that does build up.

MINIMIZING STATIC BUILD-UP

Several techniques can be used to minimize static build-up. Start by removing any carpet in your work areas. You can replace it with special antistatic carpet, but this is expensive. It's less expensive to treat the carpet with antistatic spray, which is available from electronics wholesalers. Adding humidity to the air can help reduce the presence of static charges as well.

Even the choice of clothing you wear can affect the amount of ESD. Polyester has a much greater ESD potential than cotton.

Many builders who have their workbench on a concrete floor use a rubber mat to minimize the risk of electric shocks from the ac line. Unfortunately, the rubber mat increases the risk of ESD. An antistatic rubber mat can serve both purposes.

Many components are shipped in antistatic packaging. Leave components in their conductive packaging. Other components, notably MOSFETs, are shipped with a small metal ring that temporarily shorts all of the leads together. Leave this ring in place until the device is fully installed in the circuit.

Use antistatic bags to transport susceptible components or equipment. Keep your workbench free of objects such as paper, plastic, and other static-generating items. Use conductive containers with a dissipative surface coating for equipment storage. Storing partially assembled projects in antistatic bags is also a good idea.

These precautions help reduce the build-up of electrostatic charges. Other techniques offer a slow discharge path for the charges or keep the components and the operator handling them at the same ground potential.

DISSIPATING STATIC

One of the best techniques is to connect the operator and the devices being handled to earth ground, or to a common reference point. It is not a good idea to directly ground an operator working on electronic equipment, though; the risk of shock is too great. If the operator is grounded through a high-value resistor such as 100 kΩ to 1 MΩ, ESD protection is still offered but there is no risk of shock.

The operator is usually grounded through a conductive wrist strap. This wrist band is equipped with a snap-on ground lead. A 1 MΩ resistor is built into the snap of the strap to protect the user should a live circuit be contacted. Build a similar resistor into any homemade ground strap.

The devices and equipment being handled are also grounded, by working on a charge-dissipating mat that is connected to ground. The mat should be an insulator that has been impregnated with a resistance material. Suitable mats and wrist straps are available from most electronics supply houses. **Figure 23.7** shows

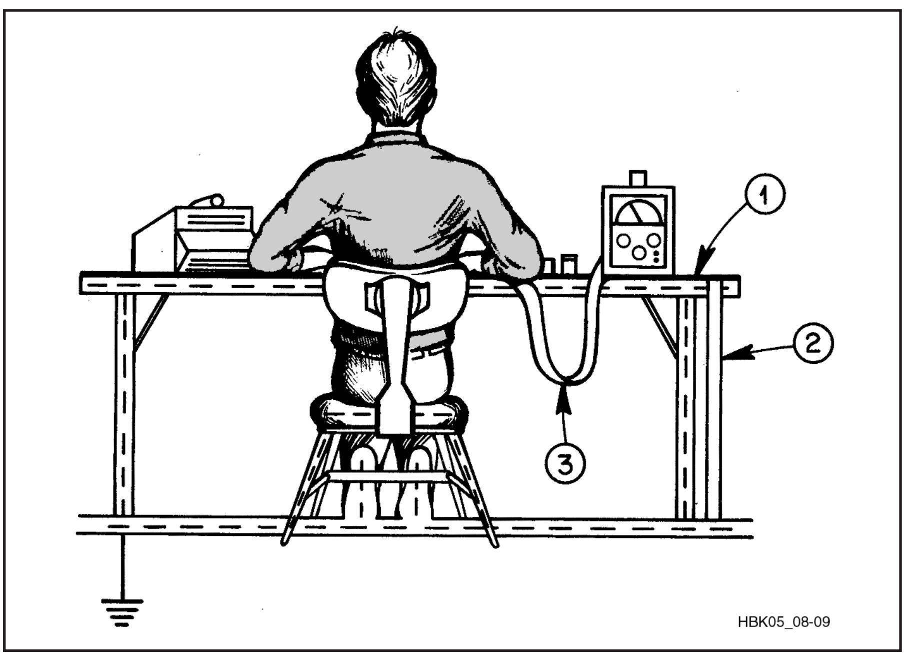


Figure 23.7 — A work station that has been set up to minimize ESD features with (1) a grounded dissipative work mat and (2) a wrist strap that (3) grounds the worker through high resistance.

a typical ESD-safe work station.

The work area should also be grounded, directly or through a conductive mat. Use a soldering iron with a grounded tip to solder sensitive components. Most irons that have three-wire power cords are properly grounded. When soldering static-sensitive devices, use two or three jumpers: one to ground you, one to ground the work, and one to ground the iron. If the iron does not have a ground wire in the power cord, clip a jumper from the metal part of the iron near the handle to the metal box that houses the temperature control. Another jumper connects the box to the work. Finally, a jumper goes from the box to an elastic wrist band for static grounding.

23.5.2 Sorting Parts

When building a project, especially one packaged as a kit, finding the appropriate container to sort your components can be a problem. There are a number of things that make this task a lot easier and at a low cost.

Using a plastic egg carton or poking component leads into Styrofoam works quite well for sorting parts, but both methods can lead to ESD damage of sensitive components. Use this method for components such as resistors, capacitors, and inductors that are relatively immune to ESD.

Metal cupcake trays are ideal, as the tray itself can be grounded through a 1 MΩ resistor to prevent static buildup. Cupcake trays typically come in three sizes: 6, 12 and 16 cups. The best sorting technique is to place the most common or first-used components in the cups closest to the builder (usually resistors),

Common Standard Parts

When building a project or repairing equipment, it is helpful to have an assortment of standard and common parts on hand for use in modifying circuit designs and fine-tuning performance. Making repairs or completing a kit that is missing a part are also good reasons to keep an assortment of parts on hand. It is not possible to have every possible needed part, but the majority of components in any project are usually one of the common standard values.

Assortments of new parts such as resistors, capacitors, and semiconductors are available from electronic distributors such as DigiKey, Mouser, Newark, and Jameco, and from parts companies such as Velleman. These are often available with a storage container or cabinet. When taking into account the cost of the parts cabinet and the parts themselves, it is very economical to buy standard parts in this manner. If parts cabinets are not available, craft stores as well as fishing supply stores often have low-cost compartmentalized containers ideal for sorting and storing small parts. Tackle boxes are also useful for storing components and materials, particularly if purchased during end-of-season sales.

Another recommendation is to buy in quantity when ordering parts for a project. Not only can you often get a price break on the individual components but you will be building up your store of components along the way.

Many vendors also have bags of one or more values of surplus parts from electronic manufacturing. It is often less expensive to buy an entire bag of surplus parts than it is to buy even one or two of them elsewhere. Some come as truly random assortments, often called “grab bags,” and others may be sold as “tapes” that were prepared for a parts-placement machine. You will have to sort out the components but the price makes it worthwhile!

You can accumulate a good selection of parts at ham radio flea markets. It is common to see parts cabinets available with entire collections of components and hardware! Grab bags and parts junk boxes are also common here and generally a very good value if you are willing to sort through the components.

The tables in this sidebar list common parts used in many projects. It is a good idea to accumulate these parts and keep them on hand. Keep in mind that you do not have to have every one of these parts, but have the list in mind as you shop for parts. (See the **Circuits and Components** chapter for more information on part types.)

Resistors (values in ohms)

10	15	18	22	27	33	39	47	56	68	82	100	120
150	180	220	270	390	470	560	680	820				
1.0k	1.2k	1.5k	1.8k	2.2k	2.7k	3.3k	3.9k	4.7k	5.6k	6.8k	8.2k	
10k	12k	15k	18k	22k	27k	33k	39k	47k	56k	68k	82k	
100k	120k	150k	180k	220k	470k	560k	680k	820k	1M	4.7M		

Potentiometers (values in ohms)

500	1k	5k	10k	100k	1M

Ceramic disc capacitors (values in pF)

5	10	18	22	27	33	39	47	56	68	82	100	120
150	220	390	470	560	680	1000	4700					

Ceramic disc capacitors (values in μF)

0.001	0.005	0.01	0.022	0.05	0.1

Electrolytic and tantalum capacitors (values in μF)

1	2.2	4.7	10	22	33	47	100	220	470	1000	2200

Diodes and rectifiers

1N34A	1N914	1N4001	1N4007	1N4148	1N5401	1N5819

Transistors

2N2222	2N3055	MJ2955	2N3904	2N3906	2N4401	2N4404	2N7000	IRF510	TIP31C	TIP32C

Voltage regulators

78L05	7805	7812	7815	LM317	723

Operational amplifiers and miscellaneous ICs

324	741	747	TL081
555 (timer)			
ULN2001 (driver array)			

with the least used parts (usually mounting hardware) in the farther cups.

Another great idea for parts sorting is to use a fishing tackle box with removable trays. Inexpensive tackle boxes are available at outdoor supply houses and department stores, and you can often find them on sale. The tackle box has the distinct advantage of allowing you to sort your parts into different compartments within each movable tray, and some trays have pieces that allow you to resize the compartments to better fit the parts for your project. Many tackle boxes have a larger open space in the top that allows you to store your partially finished boards, with the remaining components in the closeable trays below it. This arrangement is ideal to protect your project from damage and can be securely stored between work sessions.

23.5.3 Construction Order

When building a kit or DIY project, the question often arises as to what order the parts need to be mounted. In a kit, the manual often is very explicit, requiring the builder to construct the project stage by stage. Other kit manuals offer only a minimum of directions, leaving it up to the builder. Have the manual handy, either printed or on a laptop or tablet nearby for reference.

In stage by stage construction, each stage in the project is completed in order to allow the builder to test and troubleshoot that area of the project without having a more complex problem to solve with all components mounted. This method also allows the builder to learn the principles involved in the project and how each part of the circuit works from the power supply to the output. This is a great aid to future modification and repair.

When building a kit in stages, it is often better to sort the parts by stages as well, placing the parts from each stage in their own space. That way, when each stage is completed, there should be no extra parts left over in that stage's container. Number the stages, if they are not already numbered in the manual, and place a small piece of paper with that number in each compartment, indicating the stage that those parts belong to.

When given a minimum of assembly instructions, the best approach is to mount the resistors first, then the capacitors, and then the semiconductors, followed by the more unique components. The majority of parts are then mounted early in the process, so finding the remaining part locations is a lot easier. This technique also allows the builder to double-check the usage of parts. Try to mount large parts after the smaller surrounding parts so as not to possibly block your ability to properly mount all of them.

Inventory the parts before commencing construction. In kits or DIY projects, if a change is introduced after a number of kits have been assembled, there is a chance of errors so that the parts list or board layouts do not reflect changes in the design of the circuit. Sometimes, a number of extra parts are supplied with a kit to facilitate different options, such as the choice of bands covered. Sometimes parts are eliminated or substituted with a change of other components in the circuit. Be sure to ask the kit supplier if you are not sure as to why you have an empty space or surplus parts. Sometimes new parts values are added to substitute for old values already packed in the kit, making the old values surplus. Resolve questions about component placement before powering up a completed project.

23.5.4 Component Mounting

When working with a large number of components, there are a few techniques that can be helpful should troubleshooting be required. Although resistors are not polarized, it is a good idea to mount them with the color codes reading in the same direction to make it easier to spot a part that is not in its correct position. Polarity-sensitive parts, such as diodes, electrolytic capacitors, and ICs, must be placed in their specified direction. In general, mount components so their values are readable without having to remove the component from the board or bending it, causing possible damage.

Axial-lead components such as resistors are mounted in one of two methods, upright and flat. To save space on a PC board, resistors are often mounted upright with one lead bent double in a manner resembling a hairpin. The best practice is to make this bend so that the color stripes denoting the resistor value begin at the top and the precision stripe (often silver or gold) is at the bottom, making it easier to read the values once mounted. Components with alphanumeric markings, such as diodes, should be mounted with the markings visible.

Non-polarized capacitors are best placed with markings facing in the same direction, unless the markings would be blocked by another component.

For polarized parts, always double-check its positioning before soldering. A commercially prepared PC board often has stripes on the diode labels, indicating which end to place the cathode stripe on the diode. A "+" sign on the board inside or next to the circle for a capacitor denotes the positive lead, which will often be longer. LEDs will have a flat spot or a notch on a lead to identify the cathode. See the **Component Data and References** chapter and manufacturers' data sheets for more information on component body styles.

When mounting ICs, using a pin straightener helps align the leads for insertion. If one is not available, use a flat surface to align them all at once. Be sure a pin does not get bent inward and that all pins go into the socket or PC board holes.

Straight-pin "header" connectors are commonly used in projects and kits. Unless you have a specialized jig for this purpose, soldering this type of connector can appear to be difficult. An easy technique is to simply solder one pin to hold the part on the board, without regard to how exactly straight the connector is. Using a finger, apply pressure on an unsoldered pin or pins, and reheat the solo pin that was previously soldered. You can feel when the connector is moved into place, straight and vertical. Be sure to only touch a pin that is not being heated as it can become very hot! (KØNEB has contributed photos that are available in the book's online material as additional guidance.)

Once the connector is in the correct position with only one pin soldered, the other pins can be soldered, completing the connector. This same process can also be used for soldering things like plugs and jacks to a PC board. Temporarily or "tack" solder one pin, then reheat it while adjusting its position, and then complete the other pins on the connector.

23.5.5 Electronics Construction Techniques

Several different point-to-point wiring techniques or printed-circuit boards (PC boards) can be used to construct electronic circuits. Most circuit projects use a combination of techniques. The selection of techniques depends on many different factors and builder preferences.

For one-time construction, PC boards are really not necessary. It takes time to lay out, drill, and etch a PC board. Alterations are difficult to make if you change your ideas or make a mistake.

The simple audio amplifier shown in **Figure 23.8** will be built using various point-to-point or PC-board techniques. This shows how the different construction methods are applied to a typical circuit. (Surface-mount techniques are discussed in the previous section.)

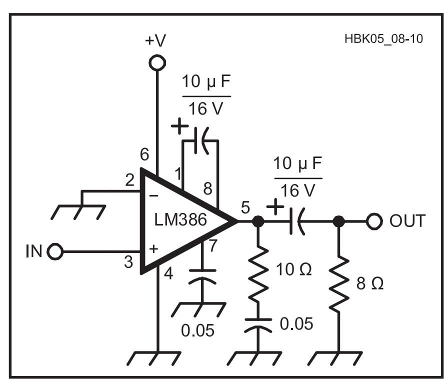


Figure 23.8 — Schematic diagram of the audio amplifier used as a design example of various construction techniques.

POINT-TO-POINT TECHNIQUES

Point-to-point techniques include all circuit construction techniques that rely on tie points and wiring, or component leads, to build a circuit. This is the technique used in most home-brew construction projects. It is sometimes used in commercial construction, such as old vacuum-tube receivers and modern tube amplifiers.

Point-to-point wiring is also used to connect the "off-board" components used in a printed-circuit project. It can be used to interconnect the various modules and printed- circuit boards used in more complex electronic systems. Most pieces of electronic equipment have at least some point-to-point wiring.

GROUND-PLANE CONSTRUCTION

A point-to-point construction technique that uses the leads of the components as tie points for electrical connections is known as "ground-plane," "dead-bug" or "ugly" construction. "Dead-bug construction" gets its name from the appearance of an IC with its leads sticking up in the air. In most cases, this technique uses copper-clad circuit-board material as a foundation and ground plane on which to build a circuit using point-to-point wiring, so in this chapter it is called "ground-plane construction." An example is shown in **Figure 23.9**.

Ground-plane construction is quick and simple: You build the circuit on an unetched piece of copper-clad circuit board. Wherever a component connects to ground, you solder it to the copper board. Ungrounded connections between components are made point-to-point. Once you learn how to build with a ground-plane board, you can grab a piece of circuit board and start building any time you see an interesting circuit.

A PC board has strict size limits; the components must fit in the space allotted. Ground-plane construction is more flexible; it allows you to use the parts on hand. The circuit can be changed easily — a big help when you are experimenting. The greatest virtue of ground-plane construction is that it is fast.

Ground-plane construction is something like model building, connecting parts using solder almost — but not exactly — like glue. In ground-plane construction you build the circuit directly from the schematic, so it can help you get familiar with a circuit and how it works. You can build subsections of a large circuit on small ground-plane modules and string them together into a larger design.

Circuit connections are made directly, minimizing component lead length. Short lead lengths and a low-impedance ground conductor help prevent circuit instability. There is usually less inter-component capacitive coupling than would be found between PC-board traces, so it is often better than PC-board construction for RF, high-gain, or sensitive circuits.

Use circuit components to support other circuit components. Start by mounting one component onto the ground plane, building from there. There is really only one two-handed technique to mount a component to the ground plane. Bend one of the component leads at a 90° angle, and then trim off the excess. Solder a blob of solder to the board surface, perhaps about 0.1 inch in diameter, leaving a small dome of solder. Using one hand, hold the component in place on top of the soldered spot and reheat the component and the solder. It should flow nicely, soldering the component securely. Remove the iron tip and hold the component perfectly still until the solder cools. You can then make connections to the first part.

Connections should be mechanically secure before soldering. Bend a small hook in the lead of a component, then "crimp" it to the next component(s). Do not rely only on the solder connections to provide mechanical strength; sooner or later one of these connections will fail, resulting in a dead circuit.

In most cases, each circuit has enough grounded components to support all of the components in the circuit. This is not always possible, however. In some circuits, high-value resistors can be used as standoff insulators. One resistor lead is soldered to the copper ground plane; the other lead is used as a circuit connection point. You can use ¼ or ½ W resistors in values from 1 to 10 MΩ. Such high-value resistors permit almost no current to flow, and in low-impedance circuits they act more like insulators than resistors. As a rule of thumb, resistors used as standoff insulators

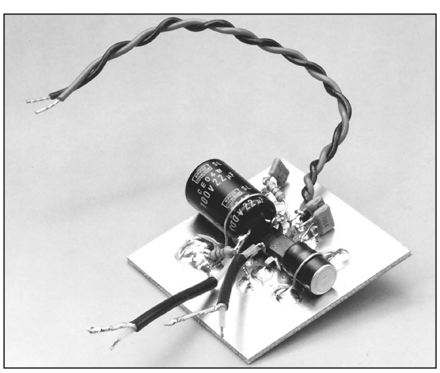


Figure 23.9 — The example audio amplifier of Figure 23.9 built using ground-plane construction.

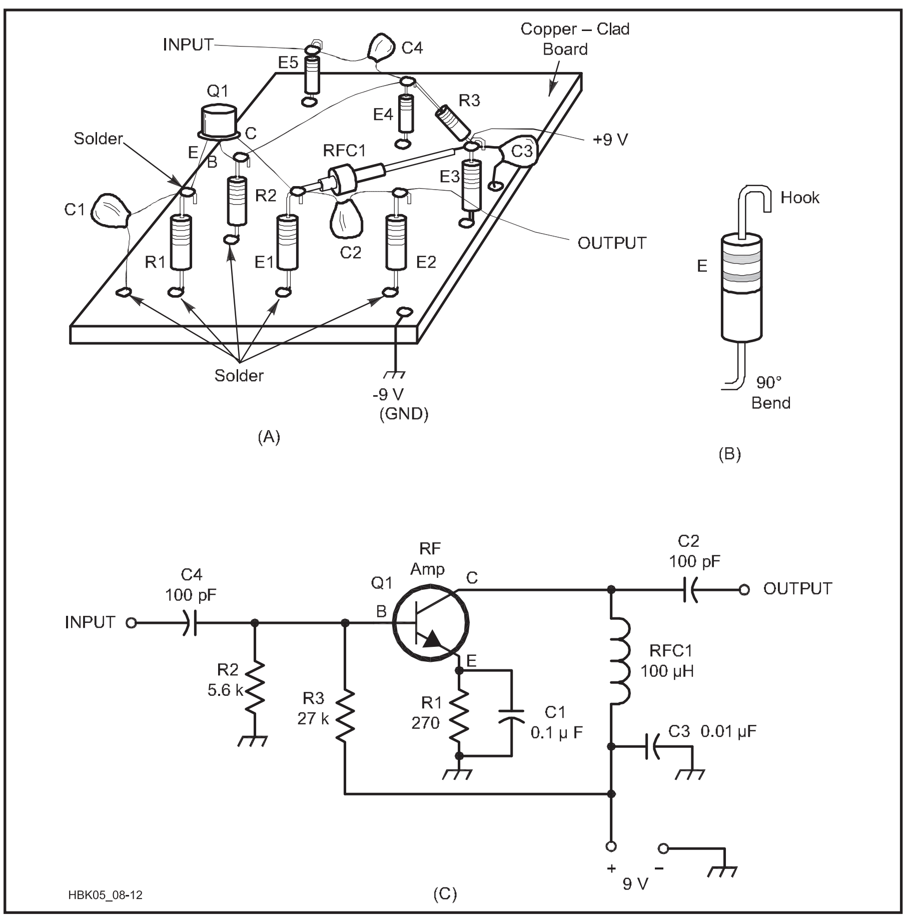


Figure 23.10 — Pictorial view of a circuit board that uses ground-plane construction is shown at A. A close-up view of one of the standoff resistors is shown at B. Note how the leads are bent. The schematic diagram at C shows the circuit displayed at A.

should have a value that is at least 10 times the circuit impedance at that point in the circuit.

Figure 23.10A shows how to use the standoff technique to wire the circuit shown at Figure 23.10C. Figure 23.10B shows how the resistor leads are bent before the standoff component is soldered to the ground plane. Components E1 through E5 are resistors that are used as standoff insulators. They do not appear in the schematic diagram. The base circuitry at Q1 of Figure 23.10A has been stretched out to reduce clutter in the drawing. In a practical circuit, all of the signal leads should be kept as short as possible. E4 would, therefore, be placed much closer to Q1 than the drawing indicates.

No standoff posts are required near R1 and R2 of Figure 23.11. These two resistors serve two purposes: They are not only the normal circuit resistances but function as standoff posts as well. Follow this practice wherever a capacitor or resistor can be employed in the dual role.

"MANHATTAN" CONSTRUCTION

Another solution to building up a circuit is called "Manhattan" construction, shown previously in the Surface-Mount Components section as Figure 23.6E. This method got its name from the appearance of the finished product, resembling the tall buildings in a city. Manhattan construction uses plain, unetched copper clad PC board material to make both the main board and the component connection points. The PC board material used may be single or double-sided.

After cutting the desired size and shape of the board required for the project, use the scraps left over to make the insulated contact pads. These pads can be made a number of ways, the most common being cutting the material into tiny squares about ¼ or ⅜ inch across. Another method to create the pads is to use a heavy-duty hole punch. This kind of punch often has changeable dies to create various sizes of round holes in materials such as sheet metal, and is available at many tool dealers. Once cut, the pads can be glued to the board in a pattern that accommodates the lead lengths of the parts to be connected.

Pads can be glued to the base board or soldered if the pads are double-sided PC board material. Use a tiny drop of instant glue, such as a cyanoacrylate "super glue" to mount the pads to the board. When soldering to the pads, use the minimum amount of heat required to avoid loosening them.

Component leads are soldered to the pads, and additional leads can be added to a pad by simply reheating the connection already there. The main board is used as the ground plane with all ground leads soldered to the board. This method of construction works well with RF circuits up to UHF.

WIRED TRACES — THE LAZY PC BOARD

If you already have a PC-board design but don't want to copy the entire circuit — or you don't want to make a double-sided PC board — then the easiest construction technique is to use a bare board or perfboard and hard-wire the traces.

Drill the necessary holes in a piece of single-sided board, remove the copper ground plane from around the holes, and then wire up the back using component leads and bits of wire instead of etched traces (**Figure 23.11**).

To transfer an existing board layout, make a 1:1 photocopy and tape it to your piece of PC board. Prick through the holes with an automatic (one-handed) center punch or by firm pressure with a sharp scriber, remove the photocopy, and drill all the holes. Holes for ground leads are optional — you generally get a better RF ground by bending the component lead flat to the board and soldering it down. Remove the copper around the rest of the holes by pressing a drill bit lightly against the hole and twisting it between your fingers. A drill press can also be used, but either way, don't remove too much board material. Then wire up the circuit beneath the board. The results look very neat and tidy — from the top, at least!

Circuits that contain components originally designed for PC-board mounting are good candidates for this technique. Wired traces would also be suitable for circuits involving multi-pin RF ICs, double-balanced mixers, and similar components. To bypass the pins of these components to ground, connect a miniature ceramic capacitor on the bottom of the board directly from the bypassed pin to the ground plane.

A wired-trace board is fairly sturdy, even though many of the components are only held in by their bent leads and blobs of solder. A drop of cyanoacrylate "super glue" can hold down any larger components, components with fragile leads, or any long leads or wires that might move.

PERFORATED CONSTRUCTION BOARD

A simple approach to circuit building uses a perforated phenolic or epoxy resin board known as perfboard. Perfboard is available with many different hole patterns. Choose the one that suits your needs. Perfboard is usually unclad, although it is made with pads that facilitate soldering.

Circuit construction on perforated board is easy. Start by placing the components loosely on the board and moving them around until a satisfactory layout is obtained. Most of the construction techniques described in this chapter can be applied to perfboard. The audio amplifier of Figure 23.8 is shown constructed with this technique in **Figure 23.12**.

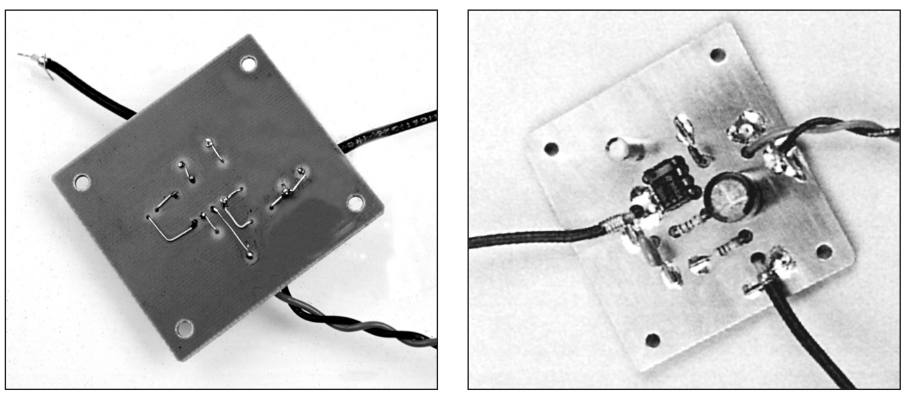


Figure 23.11 — The audio amplifier built using wired-traces construction.

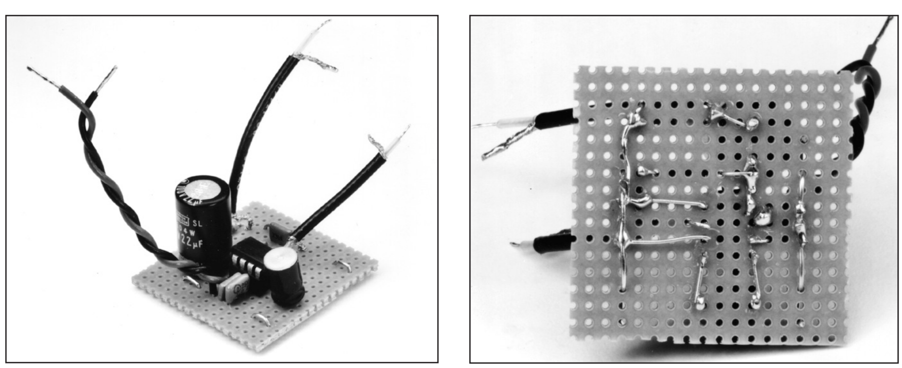


Figure 23.12 — The audio amplifier built on perforated board. Top view at left; bottom view at right.

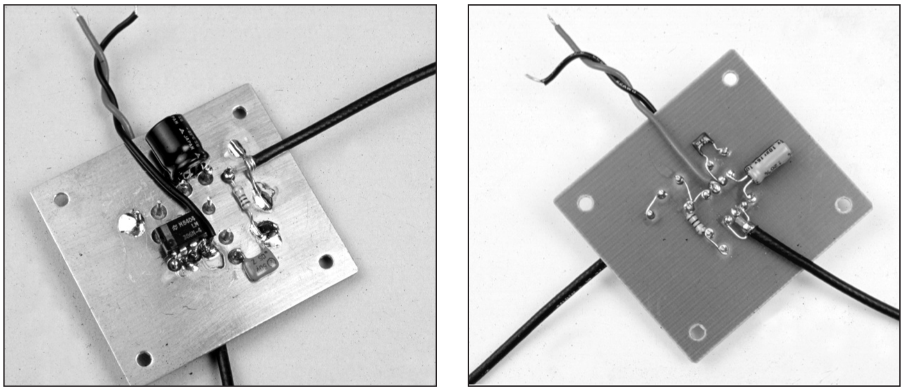


Figure 23.13 — The audio amplifier built using terminal-and-wire construction.

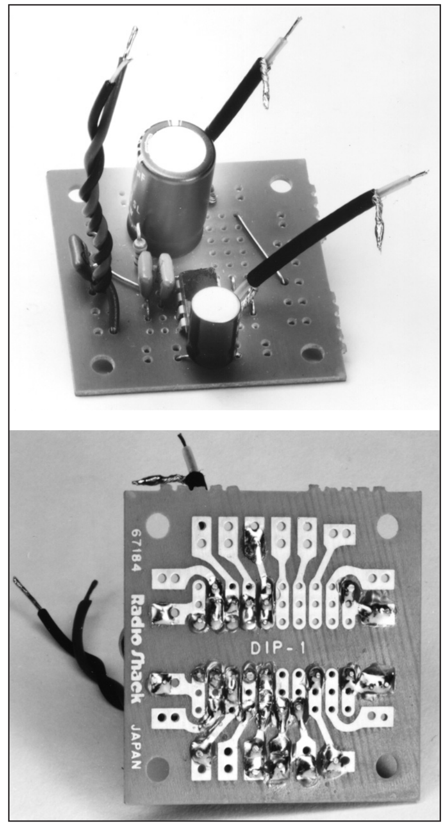


Figure 23.15 — The audio amplifier built on a multipurpose PC breadboard. Top view at A; bottom view at B.

Perfboard and accessories are widely available. Accessories include mounting hardware and a variety of connection terminals for solder and solderless construction.

TERMINAL AND WIRE

A perfboard is usually used for this technique. Push terminals are inserted into the holes in a perfboard. Components can then be easily soldered to the terminals. As an alternative, drill holes into a bare or copper-clad board wherever they are needed (**Figure 23.13**). The components are usually mounted on one side of the board and wires are soldered to the bottom of the board, acting as wired PC-board "traces." If a component has a reasonably rigid lead to which you can attach other components, use that instead of a push terminal — a modification of the ground-plane construction technique.

If you are using a bare board to provide a ground plane, drill holes for your terminals with a high-speed PC-board drill and drill press. Mark the position of the hole with a center punch to prevent the drill from skidding. The hole should provide a snug fit for the push terminal.

Mount RF components on top of the board, keeping the dc components and much of the interconnecting wiring underneath. Make dc feed-through connections with terminals having bypass capacitors on top of the board. Use small solder-in feedthrough capacitors for more critical applications.

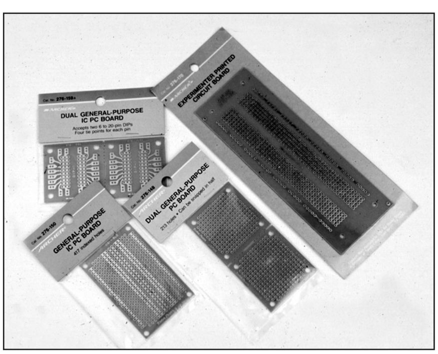


Figure 23.14 — Utility PC boards like these are available from many suppliers.

UTILITY PC BOARDS

"Utility" PC boards are an alternative to custom-designed etched PC boards. They offer the flexibility of perforated board construction and the mechanical and electrical advantages of etched circuit connection pads. Utility PC boards can be used to build anything from simple passive filter circuits to computers.

Circuits can be built on boards on which the copper cladding has been divided into connection pads. Power supply voltages can be distributed on bus strips. Boards like those shown in **Figure 23.14** are commercially available.

An audio amplifier constructed on a utility PC board is shown in **Figure 23.15**. Component leads are inserted into the board and soldered to the etched pads. Wire jumpers connect the pads together to complete the circuit.

Utility boards with one or more etched plugs for use in computer-bus, interface, and general purpose applications are widely available. Connectors, mounting hardware, and other accessories are also available. Check with your parts supplier for details.

23.5.6 Solderless Prototype Board

When designing circuits, building a prototype is often necessary to test the design. Laying out and fabricating a circuit board as soon as a schematic is available is usually premature for the home builder. Similarly, soldering components together directly or with wires makes substituting values more difficult. An easier solution is to use a solderless prototype board or "protoboard," which allows for the rapid assembly and modification of a circuit with no soldering. This section touches on important aspects of protoboards used for amateur radio circuits. A detailed guide to protoboards is available at **protosupplies.com/guide-to-solderless-breadboards**.

A protoboard makes it possible to quickly substitute parts to vary component values in a design. Having a variety of standard parts available can make experimenting or fine-tuning a design very easy and effective. Protoboards come in a variety of sizes (specified by the number of "tie points" or contacts), ranging from very small to accommodate very simple circuits up to multiple large protoboards that are placed together on a host board.

There are also a number of small circuit board adapters for mounting types of components that cannot be attached easily to a protoboard. Components like a small surface-mount IC can be used with an adapter board that has pins below it to connect to a protoboard. Some small controllers, such as an Arduino Nano or a Raspberry Pi Zero, can have connecting pins soldered to the bottom of the board intended for connecting to a protoboard. There are also protoboards with controllers built-in.

PROTOBOARD LAYOUT

Protoboard contacts (see **Figure 23.16**) are spaced by 0.1 inches in rows or grids.

Parallel groups of five contacts are connected together with a metal strip to form a common point of contact. These groups are insulated from each other to form common connection points. Components and jumpers are connected together by plugging them into the same group of five contacts.

Along one or both long edges of the protoboard are strips of contacts called *rails* that are connected together. The rails are often marked with a stripe. These are intended to be used as the positive (+) or negative (–) dc current power supply connections or the circuit common (ground) connection. If there is a stripe, the holes running down that entire length of the protoboard are all connected together along that stripe. Some boards also divide the long bus lines in the center of the board to create isolated power supply sections, so check with an ohmmeter to see if the rails go all the way down the side of the protoboard.

Across the centerline of the protoboard, groups of contacts are separated by 0.3 inches to fit most 8-14-16-18-, and 20-pin DIP ICs. Most through-hole style components, including discrete semiconductors, have leads spaced to fit the 0.1-inch hole spacing of the protoboard grid, either as manufactured or by bending the leads to standard width. The leads of components like resistors and capacitors can be shaped and bent to fit in the desired five-position row of contacts.

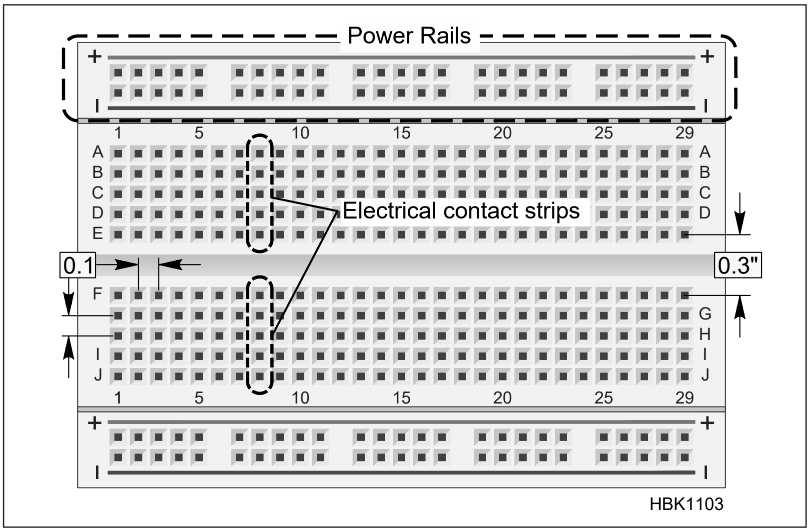


Figure 23.16 — Protoboard contacts are laid in out in groups that form common contacts and power supply rails. Contacts are laid out on a 0.1-inch grid. A 0.3-inch gap between rows of 5-contact groups accommodates standard DIP package ICs.

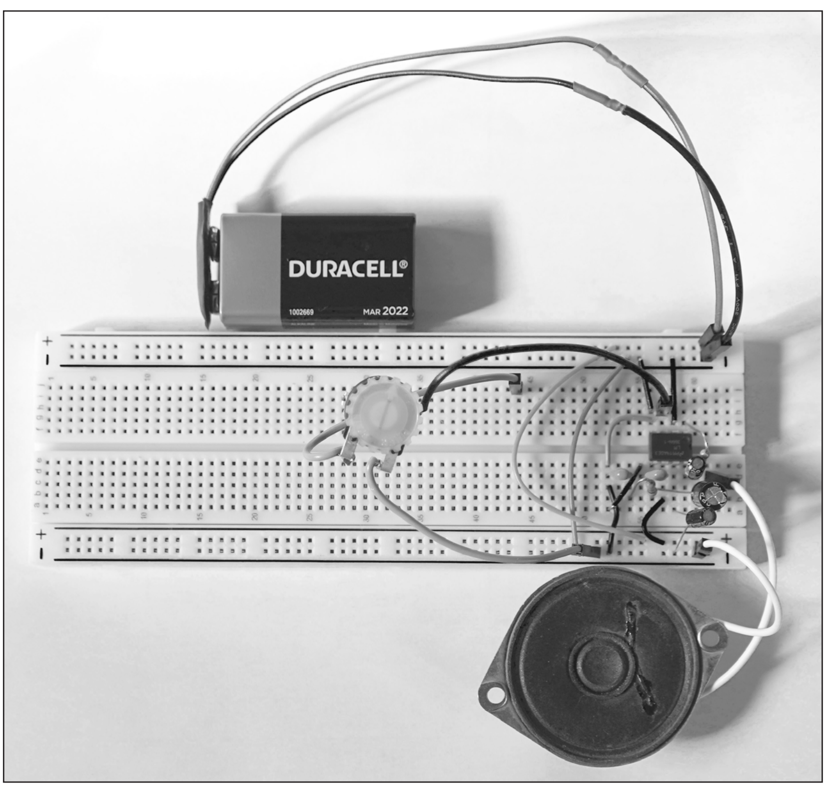


Figure 23.17 — The audio amplifier circuit of Figure 23.9 constructed on a protoboard.

VOLTAGE AND CURRENT RATINGS

Protoboards are not intended for high-power use and should never be used with non-isolated connections to utility ac power. The maximum voltage rating for most protoboards is less than 50 V dc (35 V_{RMS} ac). Higher voltages may result in arcs between adjacent contact strips or to a metal ground plane, if there is one. Maximum current rating for the spring contacts is typically a few amperes. Higher currents will cause the contact to heat and lose its temper, resulting in poor contact in future use. Heat can even melt the plastic insulation of the protoboard. A protoboard should be connected to separate assemblies that control high-power circuits. Be sure to read the board specifications before beginning to build your circuit.

WIRING JUMPERS

A typical protoboard can accommodate solid wires from #20 to #28 AWG. It may be possible to force larger wires into the holes, but that may force the underlying spring contacts apart so that they make an unreliable connection in the future. Smaller wires may not be gripped tightly by the contacts, leading to intermittent connections.

Wire jumpers can be home-made by simply cutting the appropriate gauge of solid wire to the desired length and stripping the ends. Stranded wire does not work well on protoboards unless carefully tinned. A good source of jumper wires is 8-wire Ethernet cable or round 4-conductor telephone cable. Custom jumpers can be made of any length.

Commercially prepared assortments of pre-made wire jumpers of a few different lengths and colors are available. Some are simple solid wires with stripped ends. Another type is stranded wire with a solid pin attached to each end. Pre-made jumpers have the advantage of being easier to use but are also more likely to be affected by the effects of added capacitance, resistance, and inductance caused by the addition of a longer lead length. For many circuits, this is no problem at all, but these factors need to be kept in mind when analyzing the performance of the circuit.

Pre-made cables are also available in pin-to-pin, pin-to-socket, and socket-to-socket configurations. This accommodates connect-

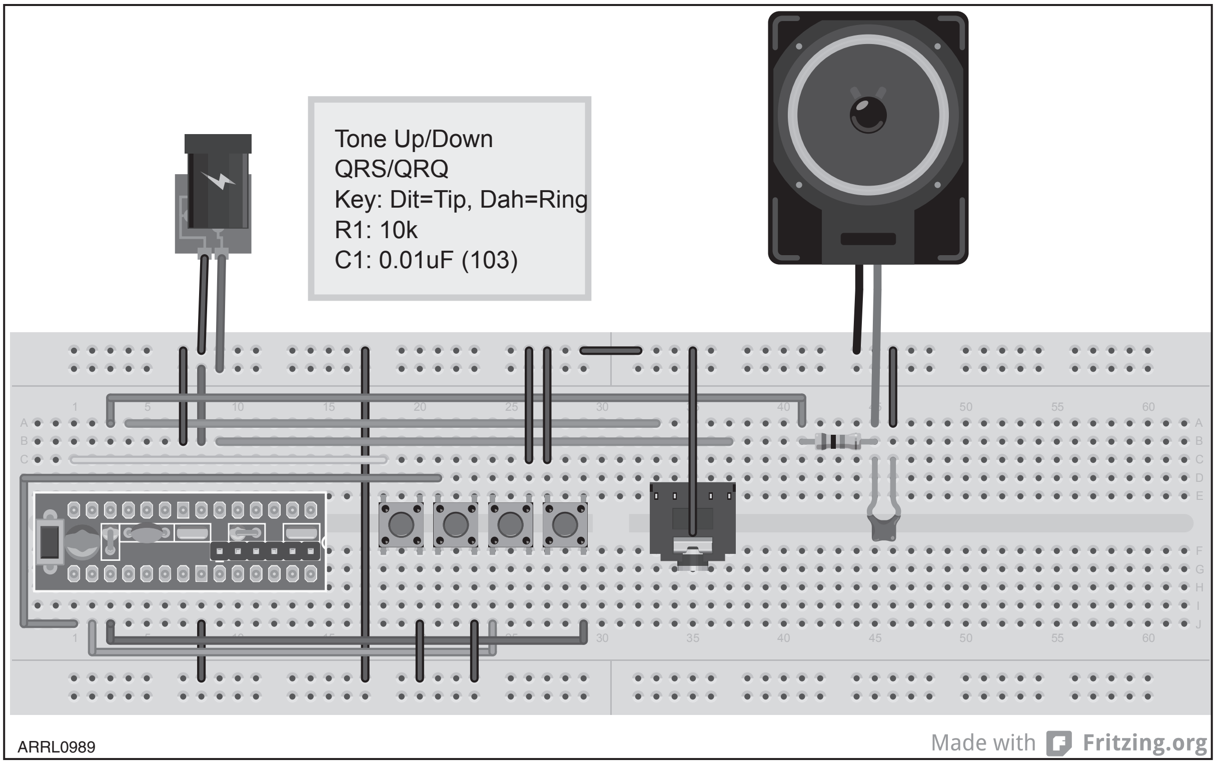


Figure 23.18 — A Fritzing diagram of a protoboard layout for a CW keyer using a small version of the Arduino (at left).

ing the protoboard to pins on single-board computer/CPU boards, such as an Arduino, Raspberry Pi, or other types of programmable controllers.

ASSEMBLY AND LAYOUT NOTES

Figure 23.17 shows a circuit similar to the audio amplifier in Figure 23.9 constructed on a protoboard. Simple protoboard circuits can be constructed directly from a regular schematic. Print a copy of the schematic and use a colored marker to highlight each connection as you install components and jumpers. It might also help to make a sketch of the protoboard showing each component with its schematic designator (R1, D2, C3, IC4, etc.)

If ICs are used, it is best to install them first. When installing components and jumpers, work around ICs from pin 1 to the last pin. This allows you to be sure all of the wires and components needed are connected to each pin.

Before you connect the power supply leads to the protoboard, measure resistance between the power rails and from the rails to any circuit common connection. Be sure there are no short circuits in the circuit or power supply lines before applying power to the board. It can be helpful for testing and troubleshooting to connect meter probes to a jumper wire for probing the circuit and making reliable contact.

It is also common for the schematic of a circuit intended to be constructed on a protoboard to be drawn as a pictorial as in **Figure 23.18**. This is called a *Fritzing diagram,* which displays individual wires and components as their physical packages would be laid out on the protoboard. Individual wires are drawn showing the contact groups they connect. The program for Fritzing is available for a small fee from **www.fritzing.org**.

Because the contacts are laid out in a grid, components such as resistors and capacitors may need to be placed at an angle between the desired contacts. This lets larger components be used without damaging the parts by bending the leads too much. It is usually okay to use any contact within the same contact group.

Parts such as resistors and diodes can also be mounted in the vertical "hairpin" style to minimize space needed (see **Figure 23.19**). This does create a small additional inductance. The alternative is to place the component flat against the board, connected to an unused contact group, and using additional wire jumpers to complete the connection.

LIMITATIONS OF PROTOBOARDS

One limitation is that the layout of these boards and the interconnections are not ideal when working with RF circuits, especially at higher frequencies. The capacitance between adjacent rows of contacts can cause problems at HF and higher frequencies. Long jumpers and indirect signal routes contribute extra inductance to the circuit. Digital circuits with high clock frequencies or extremely fast signal rise and fall times can be difficult to build reliably on a protoboard.

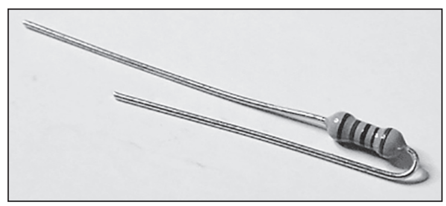


Figure 23.19— A leaded component can be mounted in a protoboard or printed-circuit board "hairpin" style by bending one lead parallel to the component.

Low-frequency audio or digital circuits work well with protoboards, although there can still be problems if the circuit is highly sensitive to stray capacitance or inductance from the long leads. Additional capacitance and inductance are also created by the arrangements of the components. Very high-impedance circuits may be sensitive to stray capacitance.

High-gain circuits can oscillate if coupling between input and output circuits is not controlled. Careful layout and using the minimum jumper length helps avoid these problems.

Be aware that after a lot of use, the connections inside a protoboard can become worn and will not work as well as when new. If you notice that the contacts have become loose or erratic, it is best to replace the entire protoboard to avoid unintended problems that can be frustrating to diagnose.

23.5.7 Printed-Circuit (PC) Boards

PC boards are everywhere — in all kinds of consumer electronics, in most of your amateur radio equipment. They are also used in most kits and construction projects. A newcomer to electronics might think that there is some unwritten law against building equipment in any other way!

The misconception that everything needs to be built on a printed-circuit board is often a stumbling block to easy project construction. In fact, a PC board is probably the worst choice for a one-time project. In actuality, a moderately complex project (such as a QRP transmitter) can be built in much less time using other techniques such as those described in the preceding section. The additional design, layout, and manufacturing is usually much more work than it would take to build the project by hand.

So why does everyone use PC boards? The most important reason is that they are reproducible. They allow many units to be mass-produced with exactly the same layout, reducing the time and work of conventional wiring and minimizing the possibilities of wiring errors. If you can buy a ready-made PC board or kit for your project, it can save a lot of construction time. This is true because someone else has done most of the real work involved — designing the PC board layout and fixing any "bugs" caused by inter-trace capacitive coupling, ground loops, and similar problems. In most cases, if a ready-made board is not available, ground-plane construction is a lot less work than designing, debugging, and then making a PC board.

Using a PC board usually makes project construction easier by minimizing the risk of wiring errors or other construction blunders. Inexperienced constructors usually feel more confident when construction has been simplified to the assembly of components onto a PC board. One of the best ways to get started with home construction (to some the best part of amateur radio) is to start by assembling a few kits using PC boards. A list of kit manufacturers can be found on the QRP ARCI website, **www.qrparci.org,** under "Links." Then click on "QRP Kits Bits and Supplies."

PC-BOARD ASSEMBLY TECHNIQUES

Cleanliness

Make sure your PC board and component leads are clean. Clean the entire PC board before assembly; clean each component before you install it. Corrosion looks dark instead of bright and shiny. Don't use sandpaper to clean your board. Use a piece of fine steel wool or a Scotchbrite cleaning pad to clean component leads or PC board before you solder them together.

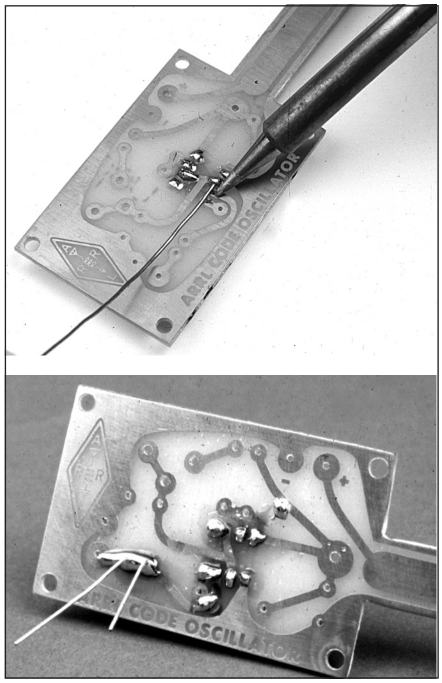


Figure 23.20 — The top photo shows how to solder a component to a PC board. Make sure that the component is flush with the board on the other side. Below is a solder bridge has formed a short circuit between PC board traces.

Installing Components

In a construction project that uses a PC board, most of the components are installed on the board. Installing components is easy — stick the components in the right board holes, solder the leads, and cut off the extra lead length. Most construction projects have a parts-placement diagram that shows you where each component is installed.

Getting the components in the right holes is called "stuffing" the circuit board. Inserting and soldering one component at a time takes too long. Some people like to put the components in all at once, and then turn the board over and solder all the leads. If you bend the leads a bit (about 20°) from the bottom side after you push them through the board, the components are not likely to fall out when you turn the board over.

Start with the shortest components — horizontally mounted diodes and resistors. Larger components sometimes cover smaller components, so these smaller parts must be installed first. If building a kit, follow the suggested order of mounting your parts if provided. Use adhesive tape to temporarily hold difficult components in place while you solder.

PC-Board Soldering

To solder components to a PC board, bend the leads at a slight angle; apply the soldering iron to one side of the lead and flow the solder in from the other side of the lead (see **Figure 23.20**). Too little heat causes a bad or "cold" solder joint; too much heat can damage the PC board. Practice a bit on some spare copper stock before you tackle your first PC board project. After the connection is soldered properly, clip the lead flush with the solder.

Make sure you have the components in the right holes before you solder them. Components that have polarity, such as diodes, ICs, and some capacitors must be oriented as shown on the parts-placement diagram.

Inspect solder connections. A bad solder joint is much easier to find before the PC board is mounted to a chassis. Look for any damage caused to the PC board by soldering. Look for solder "bridges" between adjacent circuit-board traces. Solder bridges (Figure 23.20) occur when solder accidentally connects two or more conductors that are supposed to be isolated. It is often difficult to distinguish a solder bridge from a conductive trace on a tin-plated board. If you find a bridge, re-melt it and the adjacent trace or traces to allow the solder's surface tension to absorb it. Double check that each component is installed in the proper holes on the board and that the orientation is correct. Make sure that no component leads or transistor tabs are touching other components or PC board connections. Check the circuit voltages before installing ICs in their sockets. Ensure that the ICs are oriented properly and installed in the correct sockets.

23.5.8 From Schematic to Working Circuit

Turning a schematic into a working circuit is more than just copying the schematic with components. One thing is usually true — you can't build it the way it looks on the schematic. The schematic describes the electrical connections, but it does not describe the mechanical layout of the circuit. Many design and layout considerations that apply in the real world of practical electronics don't appear on the schematic.

HOW TO DESIGN A GOOD CIRCUIT LAYOUT

A circuit diagram is a poor guide toward a proper layout. Circuit diagrams are drawn to be readable and to describe the electrical connections. They follow drafting conventions that have very little to do with the way the circuit works. On a schematic, ground and supply voltage symbols are scattered all over the place. The first rule of RF layout is — *do not lay out RF circuits as their schematics are drawn!* How a circuit works in practice depends on the layout. Poor layout can ruin the performance of even a well-designed circuit.

The easiest way to explain good layout practices is to take you through an example.

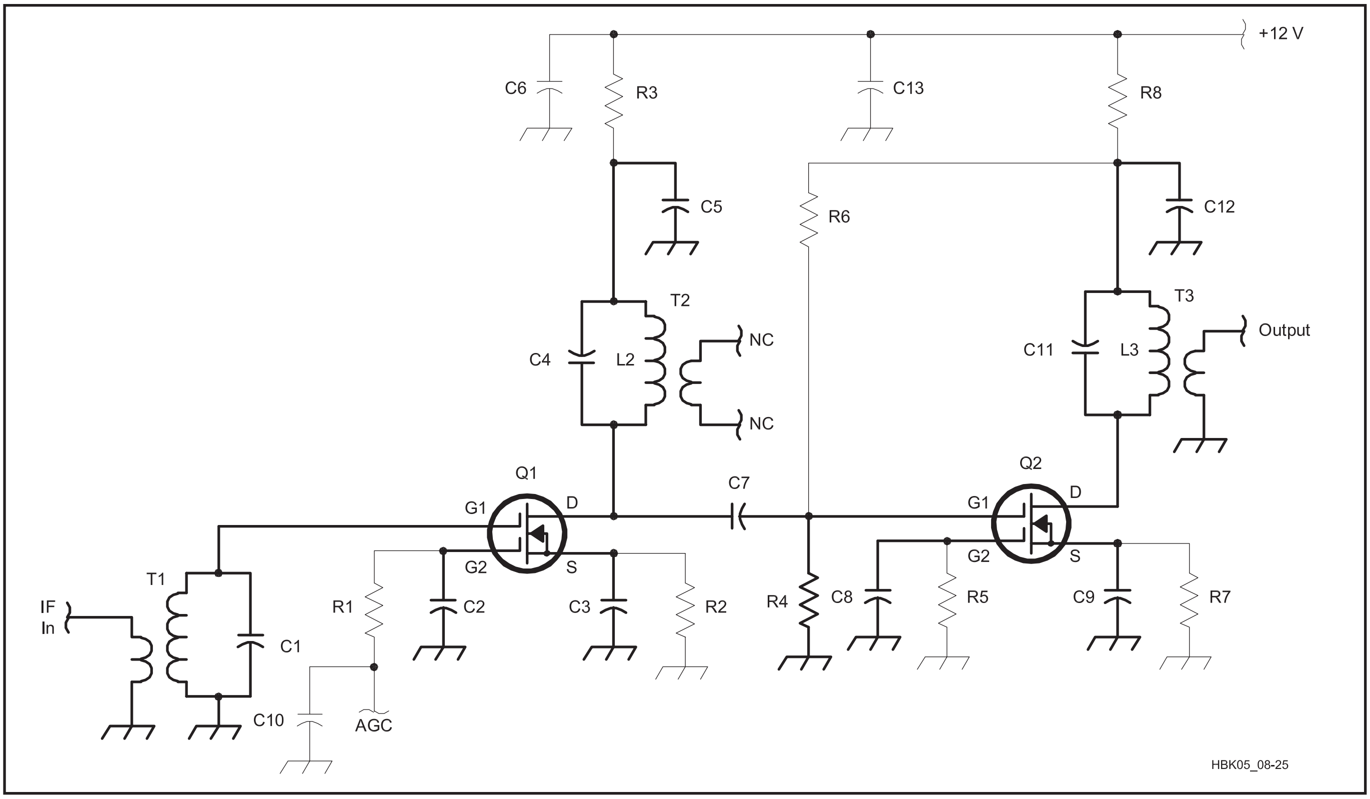


Figure 23.21 — The IF amplifier used in the design example. C1, C4 and C11 are not specified because they are internal to the IF transformers.

Figure 23.21 is the circuit diagram of a two-stage receiver IF amplifier using dual-gate MOSFETs. It is only a design example, so the values are only typical. To analyze which things are important to the layout of this circuit, ask these questions:

- Which are the RF components, and which are only involved with AF or dc?
- Which components are in the main RF signal path?
- Which components are in the ground return paths?

Use the answers to these questions to plan the layout. The RF components that are in the main RF signal path are usually the most critical. The AF or dc components can usually be placed anywhere. The components in the ground return path should be positioned so they are easily connected to the circuit ground. Answer the questions, apply the answers to the layout, and then follow these guidelines:

- Avoid laying out circuits so their inputs and outputs are close together. If a stage's output is too near a previous stage's input, the output signal can feed back into the input and cause problems.
- Keep component leads as short as practical. This doesn't necessarily mean as short as possible, just consider lead length as part of your design.
- Remember that metal transistor cases conduct, and that a transistor's metal case is usually connected to one of its leads. Prevent

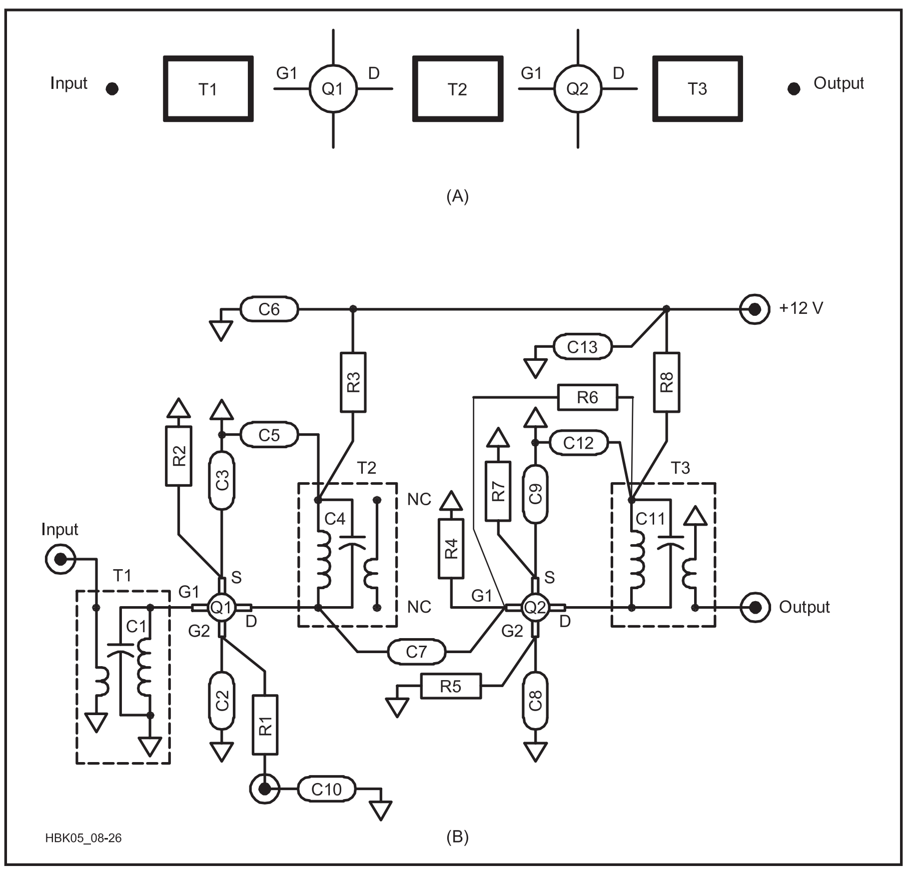


Figure 23.22 — Layout sketches. The preliminary line-up is shown in A; the final layout in B.

cases from touching ground or other components, unless called for in the design.

In our design example, the RF components are shown in heavy lines, though not all of these components are in the main RF signal path. The RF signal path consists of T1/C1, Q1, T2/C4, C7, Q2, T3/C11. These need to be positioned in almost a straight line, to avoid feedback from output to input. They form the backbone of the layout, as shown in **Figure 23.22A**.

The question about ground paths requires some further thought — what is really meant by "ground" and "ground-return paths"? Some points in the circuit need to be kept at RF ground potential. The best RF ground potential on a PC board is a copper ground plane covering one entire side. Points in the circuit that cannot be connected directly to ground for dc reasons must be bypassed ("decoupled") to ground by capacitors that provide ground-return paths for RF.

In Figure 23.23, the components in the ground-return paths are the RF bypass capacitors C2, C3, C5, C8, C9 and C12. R4 is primarily a dc biasing component, but it is also a ground return for RF so its location is important. The values of RF bypass capacitors are chosen to have a low reactance at the frequency in use; typical values would be 0.1 µF at LF, 0.01 µF at HF, and 0.001 µF or less at VHF. Not all capacitors are suitable for RF decoupling; the most common are disc ceramic capacitors. RF decoupling capacitors should always have short leads. Surface mount capacitors with no leads are ideal for bypassing.

Almost every RF circuit has an input, an output, and a common ground connection. Many circuits also have additional ground connections, at both the input and the output side. Maintain a low-impedance path between input and output ground connections. The input ground connections for Q1 are the grounded ends of C1 and the two windings of T1. The two ends of an IF transformer winding are generally not interchangeable; one is designated as the "hot" end, and the other must be connected or bypassed to RF ground.) The capacitor that resonates with the adjustable coil is often mounted inside the can of the IF transformer, leaving only two component leads to be grounded, as shown in Figure 23.22B.

The RF ground for Q1 is its source connection via C3. Since Q1 is in a plastic package that can be mounted in any orientation, you can make the common ground either above or below the signal path in Figure 23.23B, although the circuit diagram shows the source at the bottom. The practical circuit works much better with the source at the top, because of the connections to T2.

It's a good idea to locate the hot end of the main winding close to the drain lead of the transistor package, so the other end is toward the top of Figure 23.23B. If the source of Q1 is also toward the top of the layout, there is a common ground point for C3 (the source bypass capacitor) and the output bypass capacitor C5. Gate 2 of Q1 can safely be bypassed toward the bottom of the layout.

C7 couples the signal from the output of Q1 to the input of Q2. The source of Q2 should be bypassed toward the top of the layout, in exactly the same way as the source of Q1. R4 is not critical, but it should be connected on the same side as the other components. Note how the pinout of T3 has placed the output connection as far as possible from the input. With this layout for the signal path and the critical RF components, the circuit has an excellent chance of working properly.

DC Components

The rest of the components carry dc, so their layout is much less critical. Even so, try to keep everything well separated from the main RF signal path. One good choice is to put the 12 V connections along the top of the layout and the AGC connection at the bottom. The source bias resistors R2 and R7 can be placed alongside C3 and C9. The gate-2 bias resistors for Q2, R5, and R6 are not RF components so their locations aren't too critical. R7 has to cross the signal path in order to reach C12, however, and the best way to avoid signal pickup would be to mount R7 on the opposite side of the copper ground plane from the signal wiring. Generally speaking, ⅛ W or ¼ W metal-film or carbon-film resistors are best for low-level RF circuits.

Actually, it is not quite accurate to say that resistors such as R3 and R8 are not "RF" components. They provide a high impedance to RF in the positive supply lead. Because of R8, for example, the RF signal in T2 is conducted to ground through C5 rather than ending up on the 12 V line, possibly causing unwanted RF feedback. Just to be sure, C6 bypasses R3 and C13 serves the same function for R8. Note that the gate-1 bias resistor R6 is connected to C12 rather than directly to the 12 V supply, to take advantage of the extra decoupling provided by R8 and C13.

If you build something, you want it to work the first time, so don't cut corners! Some commercial PC boards take liberties with layout, bypassing and decoupling. Don't assume that you can do the same. Don't try to eliminate "extra" decoupling components such as R3, C6, R8, and C13, even though they might not all be absolutely necessary. If other people's designs have left them out, put them in again. In the long run it's far easier to take a little more time and use a few extra components, to build in some insurance that your circuit will work. For a one-time project, the few extra parts won't hurt your pocket too badly; they may save untold hours in debugging time.

A real capacitor does not work well over a large frequency range. A 10-µF electrolytic capacitor cannot be used to bypass or decouple RF signals. A 0.1-µF capacitor will not bypass UHF or microwave signals. Choose component values to fit the range. The upper frequency limit is limited by the series inductance, L_S. In fact, at frequencies higher than the frequency at which the capacitor and its series inductance form a resonant circuit, the capacitor actually functions as an inductor. This is why it is a common practice to use two capacitors in parallel for bypassing, as shown in **Figure 23.23**. At first glance, this might appear to be unnecessary. However, the self-resonant frequency of C1 is usually 1 MHz or less; it cannot supply any bypassing above that frequency. However, C2 is able to bypass signals up into the lower VHF range. (This technique should not be applied under all circumstances, as discussed in the section on Bypassing in the **RF Techniques** chapter.)

Let's summarize how we got from Figure 23.22 to Figure 23.23B:

- Lay out the signal path in a straight line.
- By experimenting with the placement and orientation of the components in the RF signal path, group the RF ground connections for each stage close together, without mixing up the input and output grounds.
- Place the non-RF components well clear of the signal path, freely using decoupling components for extra measure.

Practical Construction Hints

Now it's time to actually construct a project. The layout concepts discussed earlier can be applied to nearly any construction technique. Although you'll eventually learn from your own experience, the following guidelines give a good start:

- Divide the unit into modules built into separate shielded enclosures — RF, IF, VFO, for example. Modular construction improves RF stability and makes the individual modules easier to build and test. It also means that you can make major changes without rebuilding the whole unit. RF signals between the modules can usually be connected using small coaxial cable.
- Use a full copper ground plane. This is your largest single assurance of RF stability and good performance.
- Keep inputs and outputs well separated for each stage, and for the whole unit. If possible, lay out all stages in a straight line. If an RF signal path doubles back or re-crosses itself, it usually results in instability.
- Keep the stages at different frequencies well separated to minimize interstage coupling and spurious signals.
- Use interstage shields where necessary, but don't rely on them to cure a bad layout.
- Make all connections to the ground plane short and direct. Locate the common ground

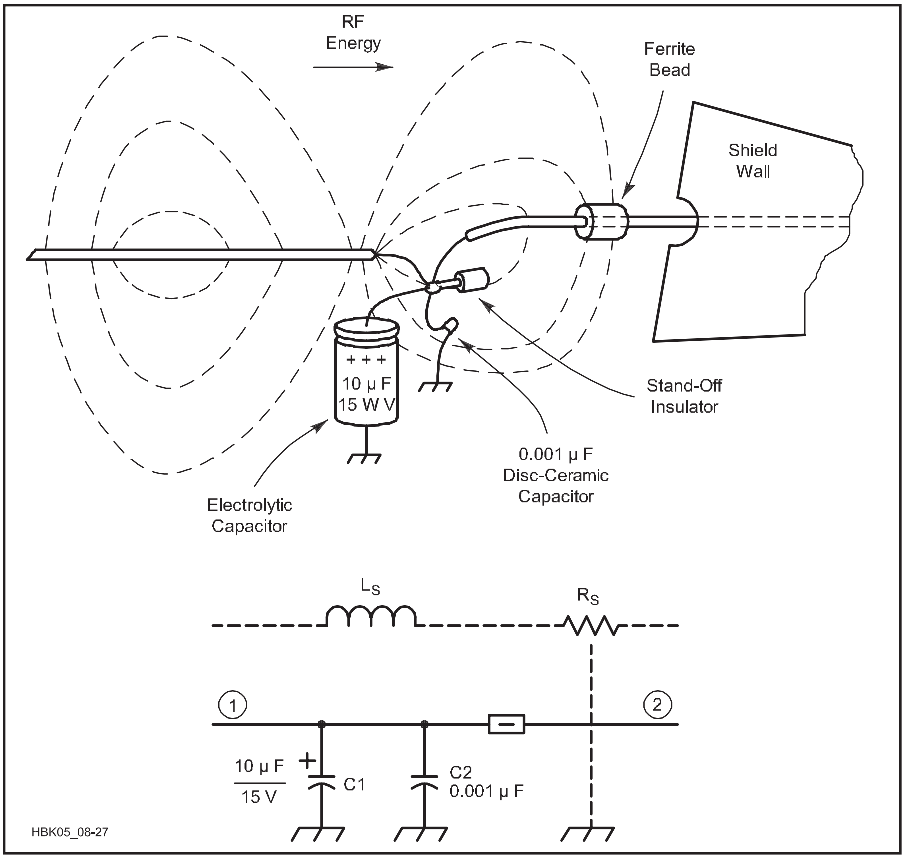


Figure 23.23 — Two capacitors in parallel afford better bypassing across a wide frequency range.

for each stage between the input and the output ground. Single-point grounding may work for a single stage, but it is rarely effective in a complex RF system.

• Locate frequency-determining components away from heat sources and mount them so as to maximize mechanical strength.

• Avoid unwanted coupling between tuned circuits. Use shielded inductors or toroids rather than open coils. Keep the RF high-voltage points close to the ground plane. Orient air-wound coils at right angles to minimize mutual coupling.

• Use lots of extra RF bypassing, especially on dc supply lines.

• Try to keep RF and dc wiring on opposite sides of the board, so the dc wiring is well away from RF fields.

• Compact designs are convenient, but don't overdo it! If the guidelines cited above mean that a unit needs to be bigger, make it bigger.

COMBINING TECHNIQUES

You can use a mixture of construction techniques on the same board, and in most cases you probably should. Even though you choose one style for most of the wiring, there will probably be places where other techniques would be better. If so, do whatever is best for that part of the circuit. The resulting hybrid may not be pretty (these techniques aren't called "ugly construction" for nothing), but it will work!

Mount dual-in-line package (DIP) ICs in an array of drilled holes, then connect them using wired traces as described earlier. It is okay to mount some of the components using a ground-plane method, push pins, or even wire wrap. On any one board, you may use a combination of these techniques, drilling holes for some ICs or gluing others upside down, then surface mounting some of the pins, and other techniques to connect the rest. These combination techniques are often found in a project that combines audio, RF, and digital circuitry.

A Final Check

No matter what construction technique is chosen, do a final check before applying power to the circuit! Things do go wrong, and a careful inspection minimizes the risk of a project beginning and ending its life as a puff of smoke! Check wiring carefully. Make a photocopy of the schematic and mark each connection and lead on the schematic with a red X, or use a highlighter to mark the circuit when you've verified that it's connected properly.

23.5.9 Tuning and Alignment

The task of performing adjustment or alignment can be difficult unless you are using the proper tools. There are variable inductors of various sizes and types and different types of variable capacitors, as well as specialized potentiometers to adjust. Each different type of adjustable component requires its own specialized tool for making the adjustment. Failure to use the proper tool in the proper manner can result in damage to the component.

In RF circuits, the proximity of your hand or a metal object can greatly influence the apparent value of the variable component and make it difficult if not impossible to adjust. For this reason, a number of plastic or ceramic tools are available to make this task as easy as possible and at a very low cost. As example is the Velleman "Plastic Tuning Needle Set," which can be found online (**www.velleman.eu**) and is widely available for a few dollars. Sources such as Mouser, Digi-Key, Newark, and Allied sell both individual tools and assortments of tools. Specialized tools, if required, are usually available from the same vendor selling the adjustable component.

When choosing a tuning tool, use the tool that best fits the adjustment hole or slot. Tips that are too small can end up damaging the inside of the adjustment slot or hole, and tools that are too large can also damage these small components or their adjustment mechanism. Choose a tuning tool with a tip that exactly matches the component.

VARIABLE INDUCTORS

Most variable inductors used in low-power RF circuits are a tiny coil of wire wound around a plastic form, with a threaded ferrite (or sometimes brass) slug in the center. For ferrite slug coils, inserting the slug into to coil increases the inductance and vice versa. (Brass slugs work oppositely and are uncommon.) Some variable inductors are inside metal cans, which act as a shield to reduce coupling to nearby components. The slug either has a hexagonal hole or a slot for a tuning tool to adjust the position of the slug like turning a screw.

Do not use small metal screwdrivers or hex keys (Allen wrenches) to adjust these coils. The metal tool will alter the inductance when inserted, making adjustment unpredictable and frustrating. A metal tool can also damage the slug by cracking it as ferrite slugs are quite brittle, ruining the inductor. Plastic or plastic-tipped tools allow you to make the adjustment while keeping your hand far away and not damaging the slug. Most tuning tools also have a mark on them in the form of a dimple, a logo/identification, or a stripe to help you count the number of times you rotate the tool.

VARIABLE CAPACITORS

Variable capacitors not tuned with a shaft and knob usually have a screw-slot for adjustment. These include ceramic and mica trimmers, piston trimmers, and small air variables. Try using a plastic tool first, but these adjustments are sometimes too stiff for a plastic tool to make. Ceramic and plastic-tipped tools are available but not common in most electronics stores. Metal screwdrivers introduce enough capacitance to alter the value of these small capacitors but may be used in a series of small adjustments.

POTENTIOMETERS

Multi-turn miniature potentiometers (a.k.a "trimpots") usually have small metal adjusting screws. Resistance values are rarely sensitive to metal tools and so a miniature jeweler's screwdriver can be used. For RF circuits, however, use a plastic or ceramic screwdriver to avoid any possible interactions with the signals.

Larger potentiometers that are found inside equipment and that do not have a shaft are designed to be adjusted with a screwdriver-style tool. While a metal screwdriver can be used, a plastic tool eliminates any chance that the metal screwdriver could slip and come in contact with nearby components or leads. The metal screwdriver's shaft can also make accidental contact with wiring inside equipment.

ADJUSTING HIGH VOLTAGE CIRCUITS

If high voltages are involved, such as during tube neutralization or bias voltage adjustment, safety precautions are important. If the adjustable component is located near an RF stage, the presence of your hand can influence the circuit. Use plastic or ceramic screwdrivers for both slotted and Philips-type screws. Be sure to use a tool that is long enough to keep your hands well away from any high voltage source! There are extra long tools available for use in high voltage areas. Shops that repair vacuum tube musical instrument amplifiers may be able to help locate suitable tools — safety first!

RESPECT ADJUSTMENT LIMITS

When making adjustments on any variable component, be sure not to force an adjustment beyond its range, as that can cause permanent damage. For example, turning a slug all the way into or out of an adjustable inductor can begin to strip the tiny threads that hold the slug, making it difficult to perform future adjustments. If you feel resistance, stop, then look to see if the adjustment is at a limit. When an adjustment reaches a component's limit, that is usually an indication of a problem somewhere else in the circuit or, less frequently, that the component value has changed. Determine whether either is the case before proceeding.

In some circuits, adjustment is quite smooth and easy while others can be quite touchy, requiring patience and a steady hand to get the circuit adjusted properly. Take your time and watch your meter or oscilloscope for the desired changes. Use the dimples or marks on the tool to give a visual indication of how the adjustment is set, or to count turns of a multi-turn adjustment. Keep notes if you are making multiple adjustments or if different adjustments interact.

Finally, don't be a "screwdriver technician" who adjusts components seemingly at random, hoping to get lucky and correct a problem. That usually results in more problems and even a completely non-functional piece of equipment requiring professional realignment or repair. Make small adjustments and if you don't see the results you were expecting, stop and figure out why before proceeding.

23.5.10 Other Construction Techniques

WIRING

Select the wire used in connecting amateur equipment by considering the maximum current it must carry, the voltage its insulation must withstand, and its use.

To minimize leakage of RF that causes EMI, the power wiring and low-level signal wiring of all transmitters should use shielded wire or coaxial cable. Receiver and audio circuits may also require the use of shielded wire at some points for stability or the elimination of coupling to adjacent circuits. Coaxial cable is recommended for all 50 Ω circuits. It can also be used for *short* runs of high-impedance audio wiring.

When choosing wire, consider how much current it will safely carry, called *ampacity*. (See the **Antennas** chapter for wire tables.) See **www.electricaltechnology.org/2021/01/standard-wire-gauge-swg-calculator.html** for a handy online calculator and table. Stranded wire is usually preferred over solid wire because stranded wire better withstands the inevitable bending that is part of building and troubleshooting a circuit. Solid wire is more rigid than stranded wire; use it where mechanical rigidity is needed or desired.

Wire with typical plastic insulation is good for voltages up to about 500 V. Use Teflon-insulated or other high-voltage wire for higher voltages. Teflon insulation does not melt when a soldering iron is applied. This makes it particularly helpful in tight places or large wiring harnesses. Although Teflon-insulated wire is more expensive, it is often available from industrial surplus houses.

Solid wire is often used to wire RF circuits in both receivers and transmitters. Bare soft-drawn tinned wire, #22 to #12 AWG (depending on mechanical requirements) is suitable. Avoid kinks by stretching a piece 10 or 15 feet long and then cutting it into short, convenient lengths. Run RF wiring directly from point to point with a minimum of sharp bends and keep the wire well-spaced from the chassis or other grounded metal surfaces. Where the wiring must pass through a chassis wall or shield, cut a clearance hole and line it with a rubber grommet. If insulation is necessary, slip spaghetti insulation or heat-shrink tubing over the wire. For power-supply leads, bring the wire through walls or barriers via a feedthrough capacitor.

In transmitters where the peak voltage does not exceed 500 V, shielded wire is satisfactory for power circuits. Shielded wire is not readily available for higher voltages — use point-to-point wiring instead. In the case of filament circuits carrying heavy current, it is necessary to use #10 or #12 AWG bare or enameled wire. Slip the bare wire through spaghetti and cover it with copper braid pulled tightly over the spaghetti. Slide the shielding back over the insulation and flow solder into the end of the braid; the braid will stay in place, making it unnecessary to cut it back or secure it in place. Clean the braid first so solder will flow into the braid with a minimum of heat.

ENAMELED WIRE

When connecting enameled wire leads, care must be taken to be sure a good connection is made. There are two methods that can be used to remove the insulation. With Thermaleze-type enamel, the heat from a soldering iron can be used to remove the insulation. You will first need to turn up the heat on your soldering iron for best results. After adding some melted solder to your soldering tip, move the drop slowly along the desired length of the wire lead. Moving it slowly gives the insulation time to melt and the solder a chance to tin the now-exposed wire. The tinned leads will then easily solder to a PC board or other mounting system.

If using other kinds of enameled wire, an emery board or small file can be used to remove the insulation. (Using a knife to scrape off the enamel usually nicks the wire, which will eventually break at that point.) Be sure you have removed it completely from the desired lead. Follow that up with your soldering iron and solder to tin the wire using a thin coating of solder. Be sure to not make the tinned surface so thick that the lead cannot fit through the PC board holes. Preparing the leads in this manner will give the best possibility for a clean and secure connection.

HIGH-VOLTAGE TECHNIQUES

High-voltage wiring and construction requires special care. Read and follow the guidelines for high-voltage construction in the **Power Sources** chapter. You must use wire with insulation rated for the voltage it is carrying. Most standard hookup wire is inadequate above 300 or 600 V. High-voltage wire is usually insulated with Teflon or special multilayer plastic. Some coaxial cable is rated at 3700 V_{RMS} (or more) *internally* between the center conductor and shield, but the outer jacket rating is usually considerably lower.

CABLE ROUTING

Where power or control leads run together for more than a few inches, they present a better appearance when bound together in a single cable. Plastic cable ties or tubing cut into a spiral are used to restrain and group wiring. Check with your local electronic parts supplier for items that are in stock.

To give a commercial look to the wiring of any unit, route any dc leads and shielded signal leads along the edge of the chassis. If this isn't possible, the cabled leads should then run parallel to an edge of the chassis. Further, the generous use of the tie points mounted parallel to an edge of the chassis, for the support of one or both ends of a resistor or fixed capacitor, adds to the appearance of the finished unit. In a similar manner, arrange the small components so that they are parallel to the panel or sides of the chassis.

Tie Points

When power leads have several branches in the chassis, it is convenient to use fiber-insulated terminal strips as anchors for junction points. Strips of this kind are also useful as insulated supports for resistors, RF chokes, and capacitors. Hold exposed points of high-voltage wiring to a minimum; otherwise, make them inaccessible to accidental contact.

WINDING COILS

A detailed tutorial for winding coils by Robert Johns, W3JIP, titled "Homebrew Your Own Inductors!" from August 1997 *QST* can be found in the Radio Technology section of the ARRL TIS at **www.arrl.org/radio-technology-topics** under Circuit Construction. Understanding these techniques greatly simplifies coil construction.

Close-wound coils are readily wound on the specified form by anchoring one end of the length of wire (in a vise or to a doorknob) and the other end to the coil form. Straighten any kinks in the wire and then pull to keep the wire under slight tension. Wind the coil to the required number of turns while walking toward the anchor, always maintaining a slight tension on the wire.

To space-wind the coil, wind the coil simultaneously with a suitable spacing medium (heavy thread, string, or wire) in the manner described above. When the winding is complete, secure the end of the coil to the coil-form terminal and then carefully unwind the spacing material. If the coil is wound under suitable tension, the spacing material can be easily removed without disturbing the winding. Finish space-wound coils by judicious applications of RTV sealant or hot-melt glue to hold the turns in place.

The "cold" end of a coil is the end at (or close to) chassis or ground potential. Wind coupling links on the cold end of a coil to minimize capacitive coupling.

Winding Toroidal Inductors

Toroidal inductors and transformers are specified for many amateur radio projects. The advantages of these cores include compactness and a self-shielding property. **Figures 23.24** and **23.25** illustrate the proper way to wind and count turns on a toroidal core.

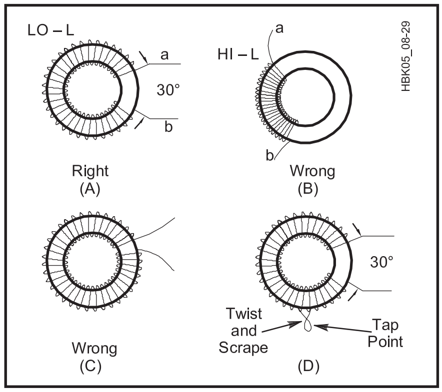


Figure 23.24 — The maximum-Q method for winding a single-layer toroid is shown at A. A 30° gap is best. Methods at B and C have greater distributed capacitance. D shows how to place a tap on a toroidal coil winding.

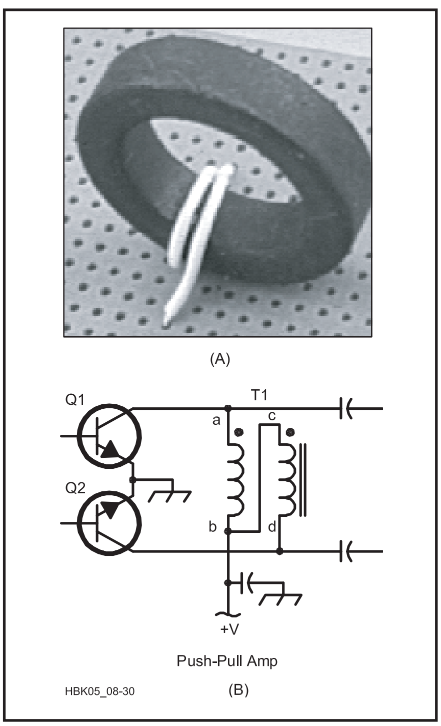


Figure 23.25 — A shows a toroidal core with two turns of wire (see text). Large black dots, like those at T1 in B, indicate winding polarity (see text).

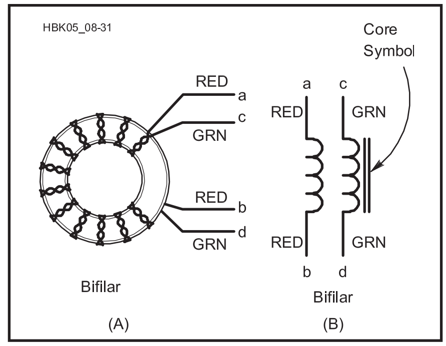


Figure 23.26 — Schematic and pictorial presentation of a bifilar-wound toroidal transformer.

The task of winding a toroidal core, when more than just a few turns are required can be greatly simplified by the use of a homemade bobbin upon which the wire is first wound. A simple yet effective bobbin can be fashioned from a wooden popsicle stick. Cut a "V" notch at each end and first wind the wire coil on the popsicle stick lengthwise through the notches. Once this is done, the wound bobbin can be easily passed through the toroid's inside diameter. While firmly grasping one of the wire ends against the toroidal core, the bobbin can be moved up, around, and through the toroidal core repeatedly until the wire has been completely transferred from the bobbin. The choice of bobbin used is somewhat dependent on the inside diameter of the toroid, the wire size, and the number of turns required.

Another method is to create a holder that allows you to grip the core, yet thread the wire easily around it. When winding toroids, be sure to seat the wire so it hugs the shape of the core, and do not allow turns to overlap unless part of a twisted group of wires as in bifilar or trifilar windings. Do not pull the wire too tight because the thin wire used in some toroids can break if pulled too tightly.

When you wind a toroid inductor, count each pass of the wire through the toroid center as a turn. You can count the number of turns by counting the number of times the wire passes through the center of the core (see Figure 23.26A).

Multiwire Windings

A bifilar winding is one that has two identical lengths of wire, which when placed on the core result in the same number of turns for each wire. The two wires are wound on the core side by side at the same time, just as if a single winding were being applied. An easier and more popular method is to twist the two wires (8 to 15 turns per inch is adequate), then wind the twisted pair on the core. The wires can be twisted handily by placing one end of each in a bench vise. Tighten the remaining ends in the chuck of a small hand drill and turn the drill to twist the pair.

A trifilar winding has three wires, and a quadrifilar winding has four. The procedure for preparation and winding is otherwise the same as for a bifilar winding. **Figure 23.26** shows a bifilar toroid in schematic and pictorial form. The wires have been twisted together prior to placing them on the core. It is helpful, though by no means essential, to use wires of different color for multifilar windings. It is more difficult to identify multiple windings on a core after it has been wound. Various colors of enamel insulation are available, but it is not easy for amateurs to find this wire locally or in small-quantity lots. This problem can be solved by taking lengths of wire (enameled magnet wire), cleaning the ends to remove dirt and grease, then spray painting them. Ordinary aerosol-can spray enamel works fine. Spray lacquer is not as satisfactory because it is brittle when dry and tends to flake off the wire.

You may also identify bifilar and trifilar toroid lead pairs of identical colors by using a continuity checker. It is a good idea to check all toroids with an ohmmeter or continuity tester to be sure there are no shorts between different windings and that there is a good connection on the ends of each winding. Testing the leads after mounting can be difficult due to the circuit layouts, so be sure to test all toroids before mounting.

The winding sense of a multifilar toroidal transformer is important in most circuits. Figure 23.26B illustrates this principle. The black dots (called phasing dots) at the top of the T1 windings indicate polarity. That is, points a and c are both start or finish ends of their respective windings. In this example, points a and d are of opposite polarity to provide push-pull voltage output from Q1 and Q2.

23.6 PCB CAD and Fabrication

[With numerous PCB design software packages available and low quantity, low-cost PCB manufacturing services accepting orders electronically, the development of PCBs has never been easier for the amateur. As with any assembly or manufacturing process, it is important to understand the vocabulary and technology in order to achieve the desired result. Thus, this section provides a detailed description of the entire process of PCB design. — Ed.]

The primary goal of using software for printed circuit board (PCB) design is the production of PCB "artwork" — the graphic designs used to create the patterns of traces that establish connectivity on the PCB. Historically, PCB artwork was created by hand on clear film using black tape and special decals that were then photographically reduced. However, free and low-cost programs specifically for the PCB design process are now widely available. These programs not only allow the creation of artwork efficiently and accurately, but produce the required ancillary files for commercial production, exchange information with schematic capture software, produce Bills of Materials (parts lists), and even include such features as three-dimensional visualization of the finished board. While artwork files can be shared with other people for PCB production, the "source" files used by the CAD program can typically only be used by other people who share the same program.

The decision to produce a PCB must take into account the nature of the circuit itself (for example, high frequency, low noise, and high current circuits require additional care). Other considerations are time available; expense; available alternatives; quantity required; ability to easily share, replicate, and improve the design; and non-electrical characteristics such as thermal and mechanical properties and physical robustness. For complicated designs, the typical PCB process ensures the connectivity of the schematic are properly reflected in the PCB, which can save a lot of debugging.

23.6.1 Overview of the PCB Design Process

The PCB design process begins with establishing the list of components in the circuit, the connections between the components, the physical outline/size of the board, and any other physical, thermal, and electrical constraints or design goals. Much of the connectivity and component information is reflected in the schematic for a circuit, so in many cases the PCB layout process begins by entering the schematic in a schematic capture program which may be integrated with the PCB CAD program or standalone. (Schematic capture is not required for PCB layout but offers so many advantages that it is almost always done.) Once the schematic is entered, there may be other options possible such as simulating the circuit as described in the preceding sections. A clean, well-organized schematic that is easily modified is an asset regardless of the circuit production and construction methods.

With input from the schematic and other information, the board outline is created, mounting and other holes placed, the components positioned, and the pattern of traces created. Once the layout is complete, in many cases it is possible to run a design rules check — the equivalent of a "spell checker." Design rules include component connections and other information to check for problems related to connectivity and manufacture. This step can save a great deal of time and expense by catching errors that could be fixed by hand, but would otherwise negate some of the benefits of a PCB.

The final step in the PCB layout program is to produce the collection of up to a dozen or so different files required for PCB production. In brief, the list includes the artwork for the pattern of traces, files for producing the board outline, solder masks, silk screens, and holes.

The user then uploads the set of files to a PCB manufacturer. As quickly as two to three days later a package will be delivered with the freshly minted boards ready for assembly! Alternatively, the user may create the board "in house" using photomechanical, CNC machining, or other processes based on the output files from the software.

23.6.2 Options for PCB Design Software

PCB software varies in features, function, and cost. The good news is that the radio amateur has excellent no-cost options that include both open-source and commercial PCB software. Needless to say, the availability and terms of PCB design software are continually in flux; check for the current terms, particularly for free software.

The first time designing and ordering a PCB can be daunting, so keep the initial job simple and pay attention to details (and read the instructions). When starting to use a specific software package, join a user's support group or forum if available. Request sample designs from other users and experiment with them to see how they are constructed and what files are required in the output data set. Once you are comfortable with the tools, you can begin on a design of your own.

FREE AND OPEN SOURCE (FOSS) SOFTWARE

This software is no-cost, and the source is available for the user to read and improve upon. Support is generally through online forums, and there are no artificial limitations on the PCB design. The functionality can't be taken away from the user, but at the same time there is no guarantee the software will continue to be improved. Two popular examples, *KiCad* (**kicad.org**) and *GNU PCB* (part of a suite of tools at **geda-project.org**), have been developed over decades, and *KiCad* in particular is quite popular and powerful. *KiCad* and other FOSS software may often be available on different operating systems (i.e., Linux, Mac, and Windows); commercial software is often limited to the Windows platform.

FREE COMMERCIAL SOFTWARE

Commercial software may be made available in a free version for several reasons, including:

• the limited function of the free offering encourages upgrade to the paid version

• the free offering is licensed only for non-commercial (e.g., educational/hobbyist) use

• output is limited to the manufacturer's PCB production

• older versions are sometimes made available as-is with limited or no support

Because there is an income stream supporting development, commercial software may have more polish or advanced features than its open-source counterparts. Letting a license (even if free) lapse, however, may mean losing access to one's designs for software that uses a subscription model. Changes in company ownership or direction have in the past resulted in changes to use policy, often in the direction of limitation. Limitations on size or layers could be annoying as one's designs increase in sophistication. Examples of free commercial software include:

Altium CircuitMaker (**www.altium.com/circuitmaker**). Few limits, but only five projects can be private; the rest are shared. Commercial use okay.

DesignSpark (**www.rs-online.com/designspark/pcb-software**). No artificial limitations; commercial use okay. From RS-Components, a major electronic component vender, which provides an easy way to order parts through the software.

ExpressPCB (**www.expresspcb.com/pcb-cad-software**). Four to six layers. Free when used to order boards through the ExpressPCB manufacturing service; otherwise there is a cost per design.

AutoDesk Eagle (**www.autodesk.com/products/eagle).** The free version is limited in size and layers, is licensed for non-commercial use, and the user must have an active license for the free version of Fusion 360 CAD (e.g., student, teacher, hobbyist).

PAID COMMERCIAL SOFTWARE

Here the sky is the limit in terms of features (and cost!), but many of the advanced features are likely of less value to the occasional amateur radio PCB designer than to someone who uses the software day in and day out, or is engaged in cutting-edge PCB designs.

Nearly every piece of software will be available at least as a trial download so that the potential user can do a test drive. For the open-source software, of course, the full version is freely available. PCB design software manuals and tutorials discuss the basic operation but also special keystrokes and other shortcuts that make operations such as routing traces much more efficient.

23.6.3 Schematic Capture

The first step in PCB design is to create a schematic. It is possible to design a PCB directly from a paper schematic, but it is much easier if the schematic is entered (or "captured") in electronic form. Schematic capture software has two outputs — the visual schematic and the component and connectivity data for subsequent PCB layout. These two separate requirements can make some operations during schematic entry more complicated than what would seem at first glance necessary. Bear in mind, however, that the user is creating not only a clear graphic representation of the circuit but of the underlying electrical connectivity.

Schematics are generally entered on a (virtual) page usually corresponding to common paper sizes — for example, 11 × 17 inches. More complicated schematics can span multiple pages using special labels or components to indicate both visual and electrical connectivity. Often one can group logically related elements into a module that can then be referenced as a "black box" on a higher-level schematic. For complex circuits, these features are extremely useful and make the difference between a jumbled diagram that is difficult to use and an organized, compact diagram that efficiently communicates the function and operation of the circuit.

COMPONENTS

The components (resistors, capacitors, etc.) on a schematic are either selected from an existing library or created by the user and stored in a custom library. It is also possible to find components and/or additional libraries on the internet, although each program has its own specific format.

Each schematic component includes a great deal more than shape and pin numbers. A typical component library entry includes:

Symbol — This is the graphic representation shown on the schematic. Many components may have the same symbol (e.g., the op amp symbol may be shared by many different types of op amps)

Pins — For each pin or point of electrical connection, the component model may specify the pin number, label (e.g., "V_{DD}"), pin type (inverting, noninverting), or pin functions (common).

PCB footprint — A given component may be available in a number of different packages (e.g., DIP or surface mount). Many components may have the same physical footprint (e.g., op amps, comparators, and optoisolators could all map to the same eight-pin DIP footprint). Footprints include the electrical connections (pins) as well as mechanical mounting holes and pad sizes, and the component outline. Note that in some software, the association of schematic symbol to PCB footprint is done as a separate step, so the schematic symbol is independent of the PCB footprint.

Value — Many components such as resistors and capacitors will have identical information except for a difference in value. For example, all ¼-W resistors may be instances of the same component, differing only in value and designator.

Designator — The unique reference to the component, such as R1, C7, D3. Designators can be assigned manually, or usually there is a provision to automatically assign them sequentially once the design is done.

Source information — Part number, vendor, cost, etc. This information is for the Bill of Materials (discussed below).

Components are typically placed on the schematic by opening a library and searching for the desired component. It may be tempting for the beginner to select a component that looks "about right" when faced with a long list of components in some libraries. There are two caveats to keep in mind. First, the schematic will be easier to read if symbols are consistent (e.g., consistently using either the US or European style of resistor symbol, but not both). Second, in many schematic capture programs the component is associated with a particular PCB footprint. In that case, during schematic capture one must also decide on the desired package for the component.

(In other cases, this mapping from schematic symbol to physical package is a later step.) For example, either "1/8W Resistor, Axial" or "1W Resistor, Upright" will result in the same neatly drawn resistor symbol on the schematic, but the PCB footprints will be dramatically different.

It is not at all uncommon to add new components to the library in the course of creating a schematic. Since many components are closely related to existing devices, the process often consists of selecting an existing schematic symbol, editing the shape and/or component data, creating a new label, and associating the part with an existing footprint. Adding a specific type of op amp is an example. This usually only needs to be done once, since symbols can be saved in a personal library to be reused in other designs and/or shared with others. It is often faster to modify a part that is close to what is desired than to "build" a new part from scratch.

Component symbols can generally be rotated and flipped when placing the component instance on the schematic.

CONNECTIONS

The schematic software will have a mode for making electrical connections, usually called "nets." For example, one might click on the "draw net" symbol and then draw a line using the mouse from one pin to another pin, using intermediate mouse clicks to route the line neatly with 90° turns on a uniform grid (e.g., 0.1 inch). Internally, the software must not only draw the visual line but recognize what electrical connectivity that connection represents. So, one must click (exactly) on a component pin to start or end a line. When making a connection between two lines that intersect, one must explicitly indicate a net-to-net connection (often with a special "dot" component). The connections on a schematic can often be assigned additional information, such as the desired width of the trace for this connection on the PCB or a name assigned by the user, such as "input signal." For most simple designs this is not needed.

Not all connections on a schematic are drawn. To make any schematic — electronic or hand drawn — more readable, conventions are often employed such as ground or power symbols or grouping similar connections into busses. Schematic capture software often supports these conventions. In some cases, components may be created with implicit power connections; in these cases, the connections may not even be noted on the schematic but will be exported to the PCB software.

Since it is often possible for component pins to be assigned attributes such as "power input," "output," "input," and so on, some schematic entry programs allow one to do an early design check. The program can then flag connections between two outputs, inputs that are missing connections, and so on. This is not nearly as important as the Design Rule Check for the PCB, discussed below.

Most schematic capture software should allow one to mark unused pins as "no connect" or "not connected" (NC or N/C), which then makes it easier to see those pins that have been unintentionally left without a connection.

Text can be placed freely on the schematic to capture important notes or to label sections, connectors, etc., and there will be a text block in a corner for date, designer, version, title, and the other information that identifies the schematic. The information in the text block is sometimes maintained as data for an entire project, or it may be entered manually.

NETLISTS

Once the components are placed and connections made, the schematic may be printed and any output files for the PCB layout software produced. The connectivity and component information needed for PCB layout is captured in a *netlist* file. The flow from schematic entry to PCB may be tightly integrated, in which case the user may switch between schematic and PCB like two views of the same design (which they are) and not have to worry about explicitly generating the netlist. The netlist can also be exported to an external circuit simulation program or be used by an integrated simulator program. (See the **Electronic Design Automation (EDA)** chapter for more information on circuit simulation.)

ANNOTATION AND BILL OF MATERIALS

The important features of *forward* and *backward annotation* enter at the interface between schematic entry and PCB layout. It is not uncommon during the PCB layout process to either come across some design deficiency or realize that a change to the schematic could produce a design that would be easier to lay out. Likewise, a review of the schematic partway through the PCB layout process could reveal some needed design change. In the case of changes to the PCB (perhaps changing some pins on a connector to make routing easier), back annotation can propagate the changes "backward" to the schematic. The connectivity data will be updated; however, the user may need to manually route the connection lines to neaten up the schematic. Likewise, changes to the schematic when the PCB is already (partially) routed are known as forward annotations and like the schematic, while the connectivity is updated the user will likely need to manually route the traces. Neither forward nor back annotation is absolutely necessary but is very useful in keeping the schematic and PCB consistent. In their absence, the user is strongly urged to keep the schematic and PCB up to date manually to avoid time-consuming problems later on.

Finally, the underlying data in the schematic can be used to produce a *Bill of Materials* (BOM). A BOM lists all the components of the schematic, typically ordered by reference designator(s), and may even be exportable for online ordering. The entry for each component usually includes a part number, cost per part, distributor, and other information used by the manufacturing process.

Association of Schematic Symbols and PCB Footprints

As was noted earlier, the selection of a schematic symbol may or may not be associated with a particular PCB footprint. If the footprints have not automatically been assigned, there is an intermediate step between schematic capture and PCB layout wherein PCB footprints are assigned to each symbol in the schematic. Unfortunately, the description of component footprints may be confusing. The use of highly condensed industry standard or non-uniform naming conventions often means the user needs to browse through the component library to see the different types of components. Resistors, diodes, and capacitors seem particularly prone to a propagation of perplexing options. One solution is to open an example PCB layout and see what library element that designer used for resistors, LEDs, and so on. Here, the PCB layout software directed at hobbyists may be superior in that there are fewer options than in professional programs.

23.6.4 PCB Characteristics

PCB CONSTRUCTION

It is useful to know a little bit about PCB construction in order to make sense of the PCB design process. **Figure 23.27** shows the basic structure of a PCB and some of its design elements (discussed in later sections). The laminate material provides a stable, electrically insulating substrate with other known characteristics (thermal, dielectric, etc.). Copper is bonded to one or both sides and selectively removed (usually chemically) to leave traces and pads. The pads provide points of connection for components. Though electrical connectivity is crucial, it is important to remember that the solder and pads provide mechanical and thermal connectivity as well. Thermal properties will be critical to components that must dissipate a lot of heat, especially if no other means (such as a heat sink) are provided. Components such as switches and potentiometers may transmit a lot of force to the solder joints, which could fail with repeated cycling, and even heavy components could exert a lot of force if the device is dropped, for example. Pads may be drilled for mounting *through-hole* components or left undrilled for *surface-mount* components.

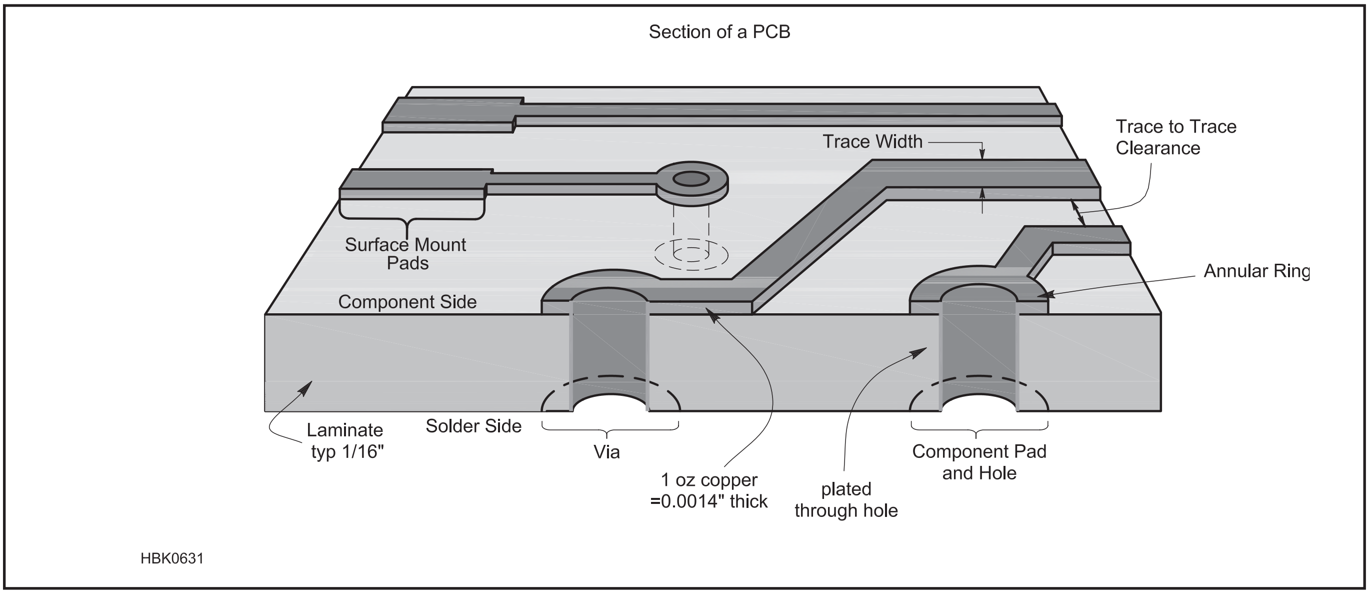


Figure 23.27 — The various elements of PCB construction and specification.

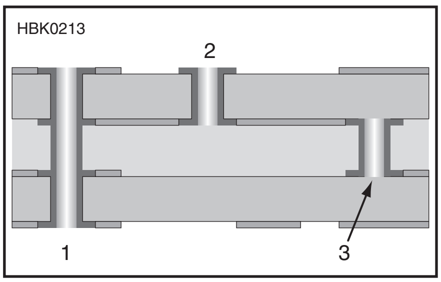


Figure 23.28 — Via refers to a plated-through hole that connects one board layer to another. Vias are used for signal, power, or ground connections and even for ventilation. Different via types include through-hole (1); blind (2), and buried (3).

A separate electrochemical process plates the inside surface of *plated-through holes* to provide connectivity between upper and lower pads. Plated-through holes whose sole purpose is to provide electrical connectivity between layers of a PCB are known as *vias*, shown in **Figure 23.28**. Since they do not need to accommodate a component lead, their hole and pad size can be smaller.

While two-layer boards can mount components on either side, most PCBs will have a primary side called the *component side* upon which most of the components will be placed, and a *solder side* dominated by soldered pins and traces. Where high density is required, surface mount (and sometimes through-hole) devices are mounted on both sides, but this is considerably more complex.

Multi-layer boards are essentially a stack of two or more two-layer boards, with an insulating layer between each board. Plated-through holes make connections possible on every layer, and the laminate material is proportionally thinner so the entire multi-layer board is roughly the same thickness as a regular two-layer board. Vias that join selected, adjacent copper layers without connecting the entire stack of layers are called "buried" or "blind" vias and are typically only needed for very dense designs. Multi-layer boards provide much more flexibility in routing signals and some other benefits such as dedicated layers for power distribution and grounding, but at often substantial additional cost and fabrication time.

PCB MANUFACTURING SPECIFICATIONS

Unless the board is manufactured by the hobbyist, the PCB files are sent out to be manufactured by a *board house*. The most important issue for the amateur may be the pricing policies of the board house. Board size, quantity, delivery time, number of layers, number and/or density of holes, presence of solder masks and silk screens, minimum trace/separation width, type of board material, and thickness of copper may all influence pricing. One cost-saving option of the past, a single-layer board, may not be offered with low-cost, low-volume services — two layers may be the simplest option, and it results in a more robust board. *[Note that most ordering specifications use English units of inches and ounces. Offshore board houses may use both English and metric units, or be metric-only. English units are used here because they are the most common encountered by hobbyists in the US. — Ed.]*

The second issue to consider is manufacturing capabilities and ordering options. These will vary with pricing and delivery times but include the following:

Board material and thickness — FR-4 is the most popular board material for low volume PCBs; it consists of flame-resistant woven fiberglass with epoxy binder. (G-10 is similar but lacks the self-extinguishing property.) Typical thickness is 0.062 inch (1/16 inch), but thinner material is often available. Flexible laminates are also available at greater cost and longer delivery time. Special board laminates for microwave use or high-temperature applications are also available.

Copper thickness — Expressed in ounces per square foot, typical values are 1 to 2 oz (1 oz corresponds to 0.0014 inch of thickness). Other values may not be available inexpensively for small volumes. Inner layers on multi-layer boards may be thinner — check if this is important. Most board designs can assume at least 1 oz copper for double-sided boards; trace width is then varied to accommodate any high current requirements.

Layers — Two-layer boards are the most common. Because of the way multi-layer PCBs are manufactured, the number of copper layers will be multiples of two. For quick-turn board houses, usually only two- or four-layer boards are available. PCBs with more than two layers will always be more expensive and often take longer to manufacture.

Minimum hole size, number, and density of holes — Minimum hole size will rarely be an issue, but unusual board designs with high hole density or many different hole sizes may incur additional costs. Be sure to include vias when specifying minimum hole size. Some board houses may have a specific list of drill sizes they support. Note that you can often just hand edit the drill file to reduce the number of different drill sizes.

Minimum trace width and clearance — Often these two numbers are close in value. Most board houses are comfortable with traces at least 0.010 inch in width, but 0.008 and 0.006 inch are often available, sometimes at a higher cost.

Minimum annular ring — A minimum amount of copper is required around each plated-through hole, since the PCB manufacturing process has variations. This may be expressed as the ratio of the pad size to hole size, but more commonly as the width of the ring.

Edge clearances — Holes, pads, and traces may not be too close to the edge of the board.

Board outline and copies — There may be options to route the outline of the board in other shapes than a rectangle, perhaps to accommodate a specific enclosure or optimize space. If multiple copies of a board are ordered, some board houses can *panelize* a PCB, duplicating it multiple times on a single larger PCB (with a reduction in cost per board). These copies may be cut apart at the board house, or small tabs or v-grooves may be left to connect the boards so assembly of multiple boards can be done as a single unit.

Tin plating — Once the traces and pads have been etched and drilled, tin plating is usually applied to the exposed copper surfaces for good soldering. (Lead-free finishes are almost always available.) Gold plating may be available for edge connectors.

Solder mask — This is a solder-resistant coating applied after tin plating to both sides of the board covering everything except the component mounting pads. It prevents molten solder from bridging the gaps between pads and traces. Solder mask is offered except by the quickest turn services. Green is the most common color, but other colors may be available with an additional cost or small delay.

Silkscreen — This is the ink layer, usually white, on top of the solder mask that lays out component shapes and designators and other symbols or text. A minimum line width may be specified — if not specified, try to avoid thin lines. All but the quickest turn services typically offer silk screening on one or both sides of the PCB.

23.6.5 PCB Design Elements

The schematic may or may not note the specific package of a part, nor the width or length of a connection. The PCB, being a physical object, is composed of specific instances of components (not just "a resistor," but a "1/4 W, axial-lead resistor mounted horizontally," for example) plus traces — connections between pins of components with a specific width and separation from other conductors. Before discussing the process of layout, we briefly discuss the nature of components and connections in a PCB.

COMPONENTS

A component in a PCB design is very similar to its counterpart on the schematic. **Figure 23.29** shows the PCB footprint of an opto-interrupter, including graphics and connectivity information. The footprint of a component needs to specify what the footprint is like on all applicable copper layers, any necessary holes including non-electrical mounting holes or slots, and any additional graphics such a silkscreen layer.

Take a common ¼-W axial-lead resistor as an example. This footprint will have two pins, each associated with a pad corresponding to the resistor's two leads. This pad will appear on both the top and bottom layers of the board, but will also have a smaller pad associated with inner layers, should there be any. The hole's size will be based on the nominal lead diameter, plus some allowance (typically 0.006 inch). The pad size will be big enough to provide a manufacturable annular ring, but is usually much larger so as to allow good quality soldering. The pins will be labeled in a way that corresponds to the pin numbering on the schematic symbol (even though for this component, there is no polarity). A silkscreen layer will be defined, usually a box within which the value or designator will appear. The silkscreen layer is particularly useful for indicating orientation of parts with polarity.

More complicated parts may require additional holes which will not be associated with a schematic pin (mounting hole, for example). These are usually added to the part differently than adding a hole with a pad — in this case, the hole is desired without any annular ring or plating. The silkscreen layer may be used to outline the part above and beyond what is obvious from the pads, for example, a TO-220 power transistor lying on its back, or the plastic packaging around the opto-interrupter in Figure 23.29.

As with schematic entry, it is not uncommon to have to modify or create a new PCB footprint. Good technical drawings are often available for electronic parts; when possible, the user should verify these dimensions against a real part with an inexpensive digital caliper. It is also useful to print out the finished circuit board artwork at actual size and do a quick check against any new or unusual parts.

TRACES

Traces are the other main element of PCB construction — the copper pathways that connect components electrically. PCB traces are merely flat wires — subject to all the properties as an equivalent thickness of copper wire in the same circumstance (resistance, capacitance, inductance, etc.). At VHF/UHF/microwave frequencies and for high-speed

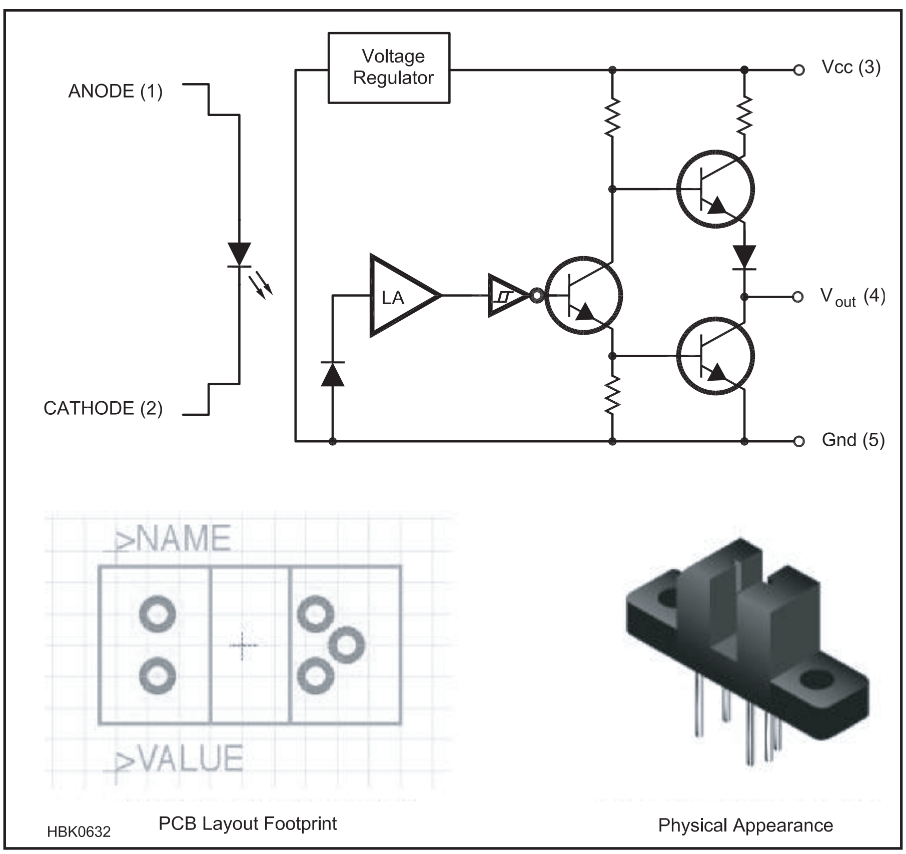


Figure 23.29 — The PCB footprint for a component, such as the opto-interrupter shown here, combines electrical connectivity as defined in the part's schematic and the part's physical attributes.

digital signals, PCB traces act as transmission lines, and these properties need to be accounted for, and can be used to advantage, in the design. At sufficiently high current they heat up, and have even been used (or abused) as heating elements.

There are few constraints on traces apart from those such as minimum width and clearance imposed by the board house. They are created by chemical etching and can take arbitrary shapes. In fact, text and symbols may be created on copper layers which may be handy if a silkscreen is not included. Traces may be of any length, vary in width, incorporate turns or curves, and so on. (In fact, one can arrange traces to produce a simple set of windings for a motor, or create a strain gage.) However, most traces will be a uniform width their entire length (a width they will likely share with other traces carrying similar signals), make neat 45° corners, and on two-layer boards have a general preference for either horizontal travel on the component side or vertical travel on the solder side.

The same considerations when building a circuit in other methods applies to PCB design, including current capacity (width of trace, thickness of copper), voltage (clearance to other signals), noise (shielding, guarding, proximity to other signals), impedance of ground and power supplies, and so on.

23.6.6 PCB Layout

With a schematic and netlist ready and all of the PCB characteristics defined, the actual layout of the PCB can begin.

BOARD SIZE AND LAYERS

The first step in PCB layout is to create the board outline to contain not only the circuit itself but also any additional features such as mounting holes. For prototype or one-off designs, the board is often best made a bit larger to allow more space between components for ease in testing and debugging. (Some low cost or freeware commercial PCB software imposes limits on board size and number of components.) The board outline may be provided in a default size that the user can modify, or the user may need to enter the outline from scratch.

As discussed above, rectangular board shapes are generally acceptable, but many board houses can accommodate more complex outlines, including curves. The author even produced a small "learn to solder" board with the shape of Cthulhu (a tentacled monster) at an accommodating board house. These outlines will be routed with reasonable accuracy and may save an assembly step if the PCB needs a cutout or odd shape to fit in a specific location.

While the software may not require deciding at the start how many layers the PCB will use, this is a decision the user should make as early as possible, since the jump from two to four or more will have a big impact on routing the traces as well as cost!

For your initial design, start with a two-layer board for a simple circuit that you have already built and tested. This will reduce the number of decisions you have to make and remove some of the unknowns from the design process.

COMPONENT PLACEMENT

Good component placement is much more than half the battle of PCB layout. Poor placement will require complicated routing of traces, make assembly difficult, and even introduce noise or other degradations, while good placement can lead to clean, easily routed, easy-to-assemble designs.

The first elements placed should be mounting holes or other fixed location features. These are often placed using a special option selected from a palette of tools in the software rather than as parts from a library. Holes sufficient for a #4 or #6 screw are often fine; be sure to leave room around them for the heads of the screws and nut driver or standoff below. These will be often be "non-plated-through holes" (NPTH) with no annular pad (though the board house may plate all the holes in a board, regardless).

(A brief aside on plated-through holes: The normal flow of PCB manufacture has holes drilled early in the process and all the insides plated. If for some reason one needs a hole to specifically not be plated through, this requires a separate drilling step, and perhaps extra cost or time in manufacturing. Although a mechanical element such as a mounting screw does not need a plated-through hole, it is often more economical to lump all the holes together and make all of them plated-through. Hence the option in some PCB software to combine PTH and NPTH holes into the same file.)

Depending on the software and whether schematic capture was performed, the board outline may already contain the footprints of all the circuit components (sometimes stacked in a heap in one corner of the board) and the netlist will already be loaded. In this case, components may be placed by clicking and dragging the components to the desired location on the board. Otherwise, one may need to manually import the netlist.

Most PCB programs have a "rat's nest" option that draws a straight line for each netlist connection of a component, and this is a great aid in placement because the connections between components are apparent as the components are moved around. (See **Figure 23.30**) However, connections are shown to the nearest pin sharing that electrical connection; thus, components such as decoupling capacitors (which are often meant to be near a specific component) will show rats nest connections to the nearest power and ground pins and not the pins the designer may have intended. In this case, the component must be placed by referring to the schematic.

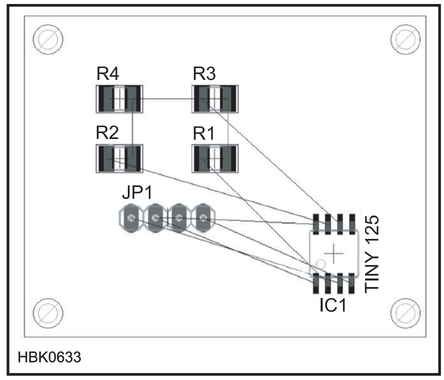


Figure 23.30 — The rat's nest view during PCB layout shows the direct connections between component pins. This helps the designer with component placement and orientation for the most convenient routing.

The PCB layout software may offer *auto-placement* in which the components are initially arranged automatically. The beginner should certainly feel free to experiment and see how well this tool performs, but is cautioned that manual tweaking will likely be needed.

PCBs need not be arranged to precisely mimic the schematic, but it is appropriate to place components in a logical flow, when possible, so as to minimize the length of traces in the signal path. Sensitive components may need to be isolated or shielded from other components, and grounding and decoupling attended to, just as one would do with a point-to-point soldered version.

During placement the user will find that different orientations of components simplify routing (for example, minimizing the number of traces that have to cross over each other or reducing the trace lengths). Components are generally rotated in increments of 90°, although free rotation may be an option (for functional or artistic intent). The user is strongly urged to maintain the same orientation on similar devices as much as possible. Mixing the position of pin 1 on IC packages, or placing capacitors, diodes, transistors, and LEDs with random orientation invites time-consuming problems during assembly and testing that can be minimized by consistent, logical layout.

Placement and orientation of components can also affect how easily the final PCB can be assembled. Allow plenty of space for sockets, for example, and for ICs to be inserted and removed. Components with a mechanical interface such as potentiometers and switches should be positioned to allow access for adjustment. Any components such as connectors, switches, or indicators (e.g., LEDs) should be positioned carefully, especially if they are to protrude through a panel. Often this will involve having the component overhang the edge of the PCB. (Beware of the required clearance between copper traces and pads to the edge of the PCB.)

Components ideally should include a silk-screen outline that shows the size of the whole component — for example, a transistor in a TO-220 package mounted flat against the PCB should have an outline that shows the mounting hole and the extent of the mounting tab. The user should also consider the clearance required by any additional hardware for mounting a component, such as nuts and bolts or heatsinks — including clearance for nut drivers or other assembly tools. (It is a good policy to avoid running traces under mounting hardware, since the only protection the trace has from shorting out is the solder mask, which could become abraded.)

Take care to minimize the mechanical stress on the PCB, since this can result in cracked traces, separated pads, or other problems. Utilize mounting holes or tabs, when possible, for components such as connectors, switches, and pots. Use two-layer boards with plated-through holes even if the design can be single-layer. Component leads soldered to plated-through holes produce much stronger mechanical connections than single-layer boards in which the soldered pad is held only by the bond between copper and laminate and is easily lifted if too much heat or stress is imposed.

When prototyping a new design, add a few unconnected pads on the circuit board for extra components (e.g., a 16 pin DIP, 0.4-inch spaced pads for resistors and other discrete components). Include test points and ground connections. These can be simply pads to which cut off leads can be soldered to provide convenient test points for ground clips or to monitor signals.

Wires or cables can be directly soldered to the PCB, but this is inconvenient when swapping out boards, and is not very robust since stress is concentrated at the solder joint, leading to fatigue breaks. Connectors are much preferred when possible and often provide strain relief for the wire or cable. However, if a wire is directly soldered to the PCB, the user should consider adding a (non-plated) hole nearby just large enough to pass the wire, including insulation. The wire can be passed from the solder side through the non-plated hole, then soldered into the regular plated-through hole. This provides some measure of strain relief, which can be augmented with a dollop of glue if desired.

ROUTING TRACES

After placing components, mounting holes, and other fixed location features that limit component or trace placement, traces can be *routed* in order to complete all the connections between pins without producing short circuits.

Most PCB design programs allow components and traces to be placed on a regular grid, similarly to drawing programs. There may be two grids — a visible coarse pitch grid and a "snap" fine pitch grid, to which components and other objects will be aligned when placed. It is good practice to use a 0.1- or 0.050-inch grid for component placement and to route traces on a 0.025-inch grid. While the "snap to grid" feature can usually be turned off to allow fine adjustment of placement, a board routed on a grid is likely to look cleaner and be easier to route.

The trace starts at a component pin and wends its way to any other pins to which it should be connected. Traces should start and end at the center of pads, not at the edge of a pad, so that the connection is properly recorded in the program's database. If a netlist has been loaded, most PCB software will display a rat's nest line showing a direct connection between pads. Once the route is completed, the rat's nest line for that connection disappears. The rat's nest line is rarely the desired path for the trace, since is a straight shot and often not the best destination. For example, when routing power traces, the user should use good design sense rather than blindly constructing a Byzantine route linking pins together in random order. For this reason, routing the power and ground early is a good practice.

High speed, high frequency, and low noise circuits will require additional care in routing. In general, traces connecting digital circuits such as microprocessors and memories should not cross or be in close proximity to traces carrying analog or RF signals. (Please refer to the **RF Techniques** chapter and earlier sections of this chapter, and the references listed at the end of this section.)

Manual routing is a core skill of PCB design, whether or not auto-routing is used. The process is generally made as simple as possible in the software, since routing will take up most of the PCB design time. A trace will be routed on the copper layer currently selected. For a single-sided board, there is only one layer for routing; for a two-layer board the component side and solder side can have traces; and for multi-layer boards additional inner layers can have traces. Often a single keystroke can change the active copper layer (sometimes automatically inserting a via if a route is in progress). The trace is drawn in straight segments and ends at the destination pin. When routing, 90° corners are normally avoided — a pair of 45° angles is the norm. **Figure 23.31** shows some sample traces.

It is good practice on a double-sided board to have one side of the board laid out with

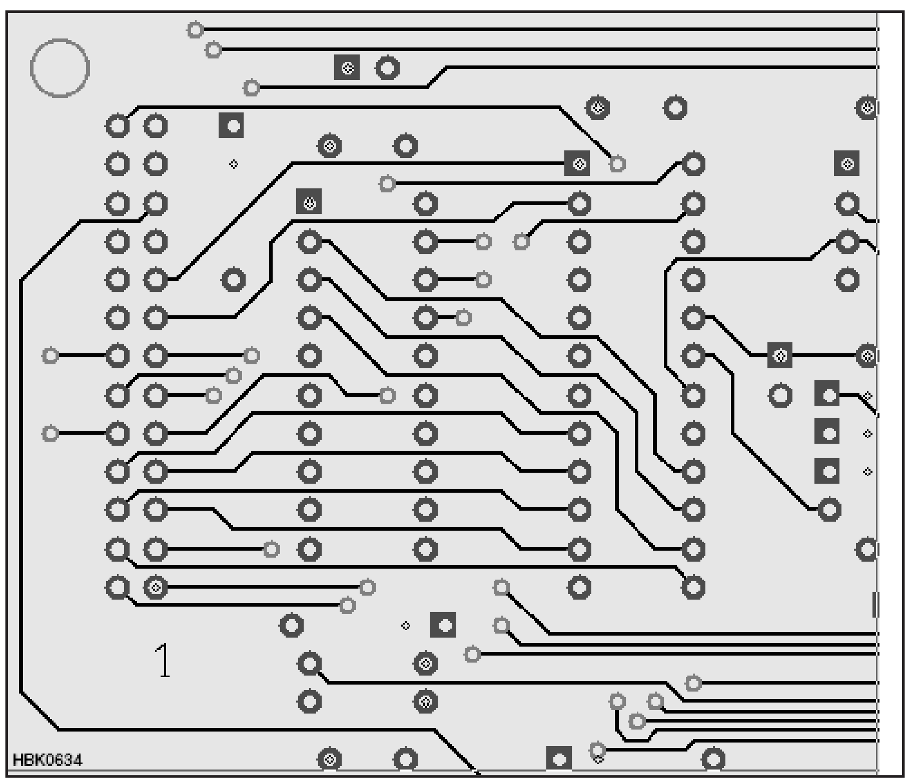


Figure 23.31 — This example shows traces on the side of the PCB for horizontal routing. Traces are routed between pins of ICs. The smaller pads are for vias to a different layer of the PCB.

mostly horizontal traces, and the other side laid out with mostly vertical traces. A trace that needs to travel primarily vertically can do so on the side with vertical traces and use a via to move to the other side to complete the horizontal part of the route.

It is easiest during testing and debugging if most traces are routed on the bottom (solder) side of the board — traces on the component side often run under ICs or other components, making them hard to access or follow. It is often much clearer to connect adjacent IC or connector pins by routing a trace that leaves one pad, moves away from the IC or connector, then heads back in to the adjacent pad to connect. This makes it clear the connection on the assembled board is not a solder bridge, which a direct connection between the two pads would resemble.

It may be the case that no amount of vias or wending paths can complete a route (particularly for single-sided boards). The one remaining tool for the PCB designer is a jumper — a wire added as a component during assembly just for the purpose of making a connection between two points on the board. Jumpers are most often required for single-sided boards; when the "jump" is rather small, uninsulated wire can be used. Jumpers are usually straight lines, and can be horizontal or vertical. Professional production PCBs may use machine-insertable zero-ohm resistors as jumpers. (The radio amateur may as well — all the major distributors will carry zero-ohm resistors in a range of packages.) Jumpers on double-sided boards are usually not viewed very favorably, but this is an aesthetic and efficiency issue, not a functional one.

Multi-layer boards (i.e., four or more) clearly offer additional routing options, but again having some dominant routing direction (vertical or horizontal) on each layer is recommended, since mixing directions tends to cause routing problems. However, it is not uncommon to devote one or two inner layers to power and ground, rather than merely be additional layers for routing signals. This allows power and ground to be routed with minimal resistance and exposes the traces carrying interesting signals on the component and solder sides, where they are available to be probed or modified. It is very difficult to modify traces on inner layers, needless to say!

Before routing too many traces, it is helpful to run the *Design Rule Check (DRC)* on the board. (See the section on Design Rule Checking below.) Applied early and often, DRC can identify areas of concern when they are easiest to correct. For example, a given trace width may provide insufficient clearance when passing between two IC pads. Waiting until the end of the initial layout process to run a DRC can lead to time-consuming rework if problems aren't easily corrected.

Some PCB design packages offer *auto-router* capability in which the software uses the component and connectivity data of the netlist and attempts to route the traces automatically. There are some circumstances when they save time, but view these tools with some caution. Auto-routers are good at solving the routing puzzle for a given board, but merely connecting all the pins correctly does not produce a good PCB design. Traces carrying critical signals may take "noisy" routes; components that should have short, low resistance connections to each other may have lengthy traces instead, and so on. More sophisticated auto-routers can be provided with extensive lists of "hints" to minimize these problems. For the beginner, the time spent conveying this design information to the auto-router is likely better spent manually routing the traces.

If an auto-router is used, at a minimum critical connections should be first routed manually. These include sensitive signals, connections whose length should be minimized, and often power and ground (for both RFI and trace width reasons). Better still is to develop a sense of what a good layout looks like (which will come with practice and analyzing well designed boards), and learn at what stage the auto-router can be "turned loose" to finish the routing puzzle.

TRACE WIDTH AND SPACING

All traces will have some width — the width may be the default width, the last width selected, or a width provided from data in the netlist. It may be tempting to route all but the power traces using the smallest trace width available from the board house (0.008 inch or smaller), since this allows the highest density of traces and eases routing. A better design practice is to use wider traces to avoid hard-to-detect trace cracking and improve board reliability. The more common 0.012-inch wide traces can be run in parallel on a 0.025-inch grid and can pass between many pads on 0.1-inch centers. Even wider traces will make the board easier to produce "in house," though the exact process used (CNC routing, chemical etching, etc.) will limit the resolution. Note that it is possible to "neck down" traces where they pass between IC or connector pads — that is, the regular, thick trace is run up close to the narrow gap between the pads, passes between the pads with a narrow width, then expands back to the original width. There is little reason to use traces wider than 0.030 inch or so for most signals (see **Table 23.4**), but power and ground trace widths should be appropriate for the current.

All traces have resistance, and this resistance is a function of the cross section of the trace (width times thickness) and the length. This resistance will convert electrical power to heat. If the heat exceeds a relatively high threshold, the trace becomes a fairly expensive and difficult-to-replace fuse. The trace width should be selected such that for the worst case expected current, heat rise is limited to some threshold, often 10 °C. In practice, power traces (especially grounds) are often made as wide as practical to reduce resistance, and they greatly exceed the width required by heat rise limits alone.

Table 23.4 summarizes maximum currents for external (component and solder side) and internal traces for some common trace widths. Internal traces (on inner layers of multi-layer boards) can carry only about half the current of external traces for the same width, since the internal layers do not dissipate heat to the ambient air like external traces can. (Note that trace widths are also sometimes expressed in "mils." 1 mil = 0.001 inch; it is shorthand for "milli-inch," not millimeter!)

There is no upper bound on the effective trace width. It is common to have large areas of the board left as solid copper. These *copper fill* areas can serve as grounds, heat sinks, or may just simplify board production (especially homemade boards). It is not a good idea

Table 23.4
Maximum Current for 10 °C Rise, 1 oz/ft^2 Copper

Based on IPC-2221 standards (not an official IPC table)

Trace Width (inches)	*Max. Current (External Trace) (A)*	*Max. Current (Internal Trace) (A)*	*Resistance (ohms/inch)*
0.004	0.46	0.23	0.13
0.008	0.75	0.38	0.063
0.012	1.0	0.51	0.042
0.020	1.5	0.73	0.025
0.040	2.4	1.2	0.013
0.050	2.8	1.4	0.010
0.100	4.7	2.4	0.0051
0.200	7.8	3.9	0.0025
0.400	13	6.4	0.0013

IPC-2221 Generic Standard on Printed Circuit Design,
Institute for Interconnecting and Packaging Electronic Circuits, **www.ipc.org**

to place a component hole in the middle of a copper fill — the copper is a very efficient heat sink when soldering. Instead, a "wagon wheel" pattern known as a thermal relief is placed (sometimes automatically) around the solder pad, providing good electrical connectivity but reducing the heat sinking. Often, copper fill areas can be specified using a polygon, and the fill will automatically flow around pads and traces in that area.

In practice, most boards will have only two or perhaps three different trace widths: narrow widths for signals, and a thicker width for power (usually with a healthy margin).

One final note on trace width — vias are typically one size (i.e., small), but multiple vias can be used to create low resistance connections between layers. Spacing the vias so their pads do not touch works well; the pads are then shorted on both top and bottom layers with a trace.

Voltage also figures into the routing equation, but instead of trace width, higher voltages should be met with an increased clearance between the trace and other copper. The IPC-2221 standard calls for a clearance of 0.024 inch for traces carrying 31-150 V (peak) and 0.050 inch for traces carrying 151-300 V (peak); these are external traces with no coating. (With the appropriate polymer solder mask coating, the clearances are 0.006 inch for 31-100 V and 0.016 inch for 101-300 V. Internal traces also have reduced clearance requirements.) Fully addressing the safety (and regulatory) issues around high voltage wiring is outside the scope of this brief review, however, and the reader is urged to consult UL or IPC standards.

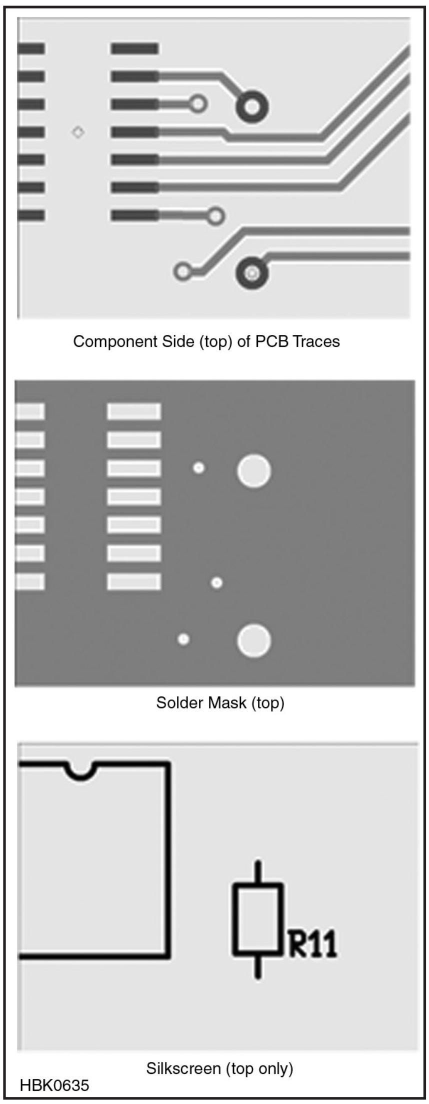


Figure 23.32 — The relationship between the layout's top copper layer with traces and pads, the solder mask that covers the copper (a separate solder mask is required for the top and bottom layers of the PCB), and the silkscreen information that shows component outlines and designators.

SILKSCREEN AND SOLDER MASK

The silkscreen (or "silk") layer contains the text and graphics that will be silkscreened on the top (and bottom) of the board, shown in **Figure 23.32**. Component footprints will generally have elements on the silkscreen layer that will automatically appear, such as designators and values, but other elements must be created and placed manually. Common silkscreen elements include circuit name, date, version, designer (and call sign), company name, power requirements (voltage, current, and fusing), labels for connections (e.g., "Mic input"), warnings and cautions, labels for adjustments, and switches. A solid white rectangle on the silkscreen layer can provide a good space to write a serial number or test information.

The board house will specify the minimum width for silkscreen lines, including the width of text. Text and graphics can be placed anywhere on the solder mask, but not on solder pads and holes.

In the past, some quick-turn board houses omit the silkscreen for prototype boards. And if the PCB is made in-house, it is unlikely to be silkscreened. As noted earlier, many of the text elements above can be placed on the external copper layers. Component outlines are not possible since the resulting copper would short out traces, but component polarization can be noted with symbols such as a hand-made "+" made from two short traces, or a "1" from a single short trace. (Note that some component footprints already follow a practice of marking the pad for pin 1 with a square pad, while others are round or oval.)

The solder mask is a polymer coating that is screened onto the board before the silkscreen graphics. As shown in Figure **23.33**, it covers the entire surface of the board except for pads and vias. Solder masking prevents solder bridges between pads and from pads to traces during assembly, and is particularly important for production processes that use wave soldering or reflow soldering. There is one solder mask layer for the top layer and another for the bottom layer. Internal layers do not need a solder mask. For prototype boards without solder masking, care must be taken to keep solder from creating unwanted bridges or short circuits.

During the PCB layout process, solder mask layers are generally not shown because they do not affect connectivity. Figure 23.33 shows a typical PCB as it appears when the PCB layout process is complete.

DESIGN RULE CHECK

If a netlist has been provided from the schematic capture program, a DRC can be made of the board's layout. The PCB software

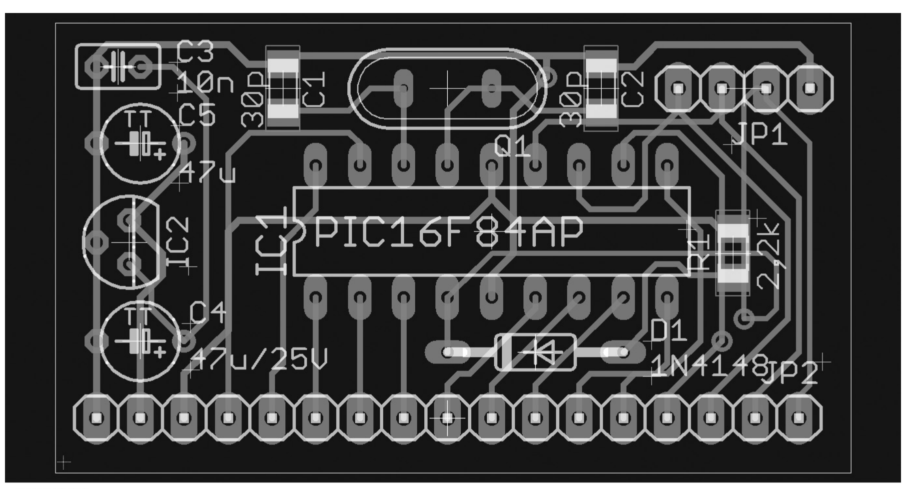


Figure 23.33 — A completed microprocessor board as it is seen in a typical PCB layout editor (Eagle). Solder mask layers are omitted for visibility. Traces that appear to cross each other are on different sides of the board and are in different colors in the layout software. The silkscreen layer is shown in white.

will apply a list of rules to the PCB, verifying that all the connections in the netlist are made, that there is sufficient clearance between all the traces, and so on. These rules can be modified based on the specific board house requirements. As stated above, it is useful to run the DRC even before all the traces have been routed — this can identify clearance or other issues that might require substantial re-routing or a different approach.

If the user has waited until all the routing is done before running the DRC, the list of violations can be daunting. However, it is often the case that many if not all of the violations represent issues that may prevent the board from operating as wished. Whenever possible, all DRC violations should be rectified before fabrication. In some cases, the board house may reject designs that violate design rules.

23.6.7 Preparation for Fabrication

LAYOUT REVIEW

Once the board has passed DRC, the electrical connectivity and basic requirements for manufacture have likely been satisfied. However, the design may benefit from an additional review pass. Turn off all the layers but one copper layer and examine the traces — often simplifications in routing will be apparent without the distractions of the other copper layers. For example, a trace can be moved to avoid going between two closely spaced pins. Densely spaced traces could be spaced farther apart. There may be opportunities to reduce vias by routing traces primarily on one layer even if that now means both vertical and horizontal travel. Repeat the exercise for all the copper layers.

Review the mechanical aspects of the board as well, including the proximity of traces to hardware. If your prototype PCB does not have a solder mask, traces that run underneath components such as crystals in a conductive case or too close to mounting hardware can form a short circuit. An insulator must be provided, or the trace can be re-routed.

GENERATING OUTPUT FILES

Once the PCB design is complete, the complete set of design description files can be generated for producing the PCB:

Copper layers — One file per copper layer. These are known as *Gerber files* and are text files of commands that were originally intended to drive a *photoplotter*. Gerber was the primary manufacturer of photoplotters, machines that moved a light source of variable width (apertures) from one location to another to draw patterns on photographic film. While photoplotters have been replaced by digital technology, the format used by Gerber has been standardized as RS-274X and is universally used except by PCB software tied to a specific manufacturer. RS-274X is related to the RS-274 ("G-Code") used by machinists to program CNC machinery but is an *additive* description (essentially saying "put copper here"), rather than describing the movements of a tool to remove material. A program is thus required to translate between Gerber and G-Code if a CNC machine is used to make a PCB by mechanically removing copper.

(Another format for PCB files, ODB++, is sometimes also supported by software and board houses, but Gerber is still all but universal.)

Drill file — The file containing the coordinates and drill sizes for all the holes, plated or not. Also called the *NC* or *Excellon* file, some board houses may require a specific format for the coordinates, but these are usually available to be set as options in the PCB program. There is only one drill file for a PCB, since the holes are drilled from one side. (Exotic options such as buried vias will require more information.) Like RS-274X apertures, the drill file will generally contain a drill table. (If non-plated-through-holes (NPTH) are required, these will need a separate NC file.)

Silkscreen — Also in RS-274 format. Some board houses can provide silkscreen on both sides of the board, which will require two files.

Solder mask — The solder mask file is used by the board house to create the solder mask. One file per side is required.

A Gerber preview program such as *Gerbv* (**gerbv.gpleda.org**, open source, *Linux*, Mac OS X), *GerbView* (found in KiCad), or *GC-Prevue* (**www.graphicode.com**, *Windows*) can be used to review the trace layout Gerber files. This is a good test — the board house will make the boards from the Gerber files, not the PCB design file. Gerber previewers can import the copper layers, silkscreen, and drill files to verify they correspond and make sense.

Any of the layers can usually be printed out within the PCB program (and/or Gerber preview program) for reference and further inspection.

In addition to files for PCB production, PCB layout programs can also generate assembly diagrams, and in some cases can provide 3D views of what the assembled board will look like. These can be useful for documentation as well as verification of mechanical issues such as height clearance.

Sending the files to the PCB manufacturer or board house and ordering PCBs is explained on the manufacturing website, or a customer service representative can walk you through the process. Some firms may also offer a design review service for first-time customers or on a fee basis.

Finally, many PCB layout programs can generate a "pick & place" or "component placement list" file, which is used by automatic pick and place machines to populate a circuit board with surface mount components. While this would obviously make sense at large scale, some boards with a large number of components could be candidates for board assembly, a service offered by many board houses. Automated assembly may require some additional PCB features for alignment.

23.6.8 Checklist

SCHEMATIC CAPTURE

Verify all non-connected inputs can be left disconnected (versus, for example, needing to tie some inputs to GND or VCC). Unused pins should be marked as no-connect (NC).

In some programs, it may be useful to include mounting hardware in the schematic so they can be imported into the layout program. If mounting holes are to be grounded, make these connections in the schematic.

Verify any necessary decoupling capacitors are placed.

Especially for prototypes, add plenty of test points to signals of interest, including ground points for test equipment.

LEDs (with appropriate current limiting resistors) are cheap, take up little space, and provide visual indication of slow (and sometimes fast!) signals and power status.

Again, for prototypes add extra components (resistors, capacitors, various IC packages) to the schematic, not connected to anything else. These will allow adding RC filters, op-amps, etc., should the need arise. Bring unused pins of microcontrollers to nearby pads to allow cleaner modifications if needed later.

Check that connections to potentiometers are such that clockwise adjustment produces the correct effect.

PRINTED CIRCUIT BOARD LAYOUT

Run the design rule check (DRC) if available, using design rules from the board house you'll use.

Verify decoupling caps are located near the components they are associated with.

Are all the traces routed? In some cases, programs will have a status line showing how many traces remain unrouted. Some rats-nest lines may be difficult to see, especially short ones between adjacent pins.

Verify mounting holes are located properly. (0.125-inch diameter works well for 4-40 hardware.) Allow clearance around holes for screw heads or standoffs to traces and other components.

Check side clearance for board guides or slotted enclosure.

Verify clearance to components and traces around connectors, modules, or parts with heatsinks. Large heat sinks may require separate mounting holes.

Adjust silk-screened component references (e.g., "R8") so they will be visible after the component is installed, and are clearly associated with the correct footprint. Component values (if used) should also be properly associated, but in tight quarters could be underneath a component.

Verify that connectors, power components (i.e., with thermal requirements), and other components are located properly--for example, a right-angle connector may need to stick out past the edge of the circuit board for proper mounting in a panel.

Check that the board outline is on the proper layer (e.g., "edge cuts") as specified by the board house.

Verify ground and power traces are thick enough to carry the required current without voltage drops. Use multiple vias for less resistance (electrical and thermal) than a single via.

Check for sufficient clearance for high voltage traces.

Make sure the orientation of components is noted for connectors, IC's, diodes, electrolytic capacitors, and other components with polarity or specific orientation. These notations should be visible after the component is installed.

Avoid traces (e.g., between adjacent pins) that could resemble solder bridges.

On the silk screen layer, add descriptions to connectors (e.g., "serial port," "test 1").

Add tick marks on the silk screen every 5 or 10 pins for high-pin count components to aid in debugging.

Add identification on the front (and/or back) silk screen, including

Name/function of board

• Date
• Board version
• Company/designer
• Call sign
• Website or email
• Copyright (as needed)
• Power requirements (e.g., "+12VDC 1.5A")
• A solid rectangular block on the front silk screen allows for a hand-written serial number

Several PCB layout checklists can be found online, including the comprehensive one at **aqdi.com/wordpress/PCBChecklistLive.htm**.

PCB CAD REFERENCES

Analog Devices, *High Speed Design Techniques*, Analog Devices, 1996.

Johnson, Howard, and Graham, Martin, *High Speed Digital Design: A Handbook of Black Magic*, Prentice Hall, 1993.

Ott, Henry, *Electromagnetic Compatibility Engineering*, Wiley Press, 2009.

Pease, Robert A, *Troubleshooting Analog Circuits*, Butterworth-Heinemann, 1991.

Silver, W. NØAX, "Hands-On Radio Experiments 107-110: PCB Layout," *QST*, Dec 2011 through Mar 2010.

23.7 Microwave Construction

Paul Wade, W1GHZ, updated this short tutorial to construction practices suitable for microwave frequency operation, originally written by ARRL Laboratory Engineer Zack Lau, W1VT. For more information on microwave equipment, read Paul's series of columns, "Microwavelengths," in *QST.*

Microwave construction is not only fun, but within the capabilities of most amateurs. To get on the air requires some degree of construction, since you can't buy a ready to go box. At a minimum, a few modules must be connected together with coax and power cables. At the other extreme, the whole station may be homebrewed from scratch, or from a kit.

The growth of mobile phones and wireless networking has caused a proliferation of microwave integrated circuits offering high performance at low cost. Some of these are also useful for ham applications, so that microwave construction has evolved from traditional waveguide "plumbing" to printed circuitry requiring surface-mount assembly of tiny components. Waveguide techniques are still valuable for some components, such as filters and antennas, so proficiency in both types of construction is valuable.

A problem we all face when building new microwave equipment is finding a nearby station to try it out. One solution is to convince a buddy to build a similar system, so that you will both have someone to work with. If you have complementary skills, say one with metal and the other with surface-mount soldering, you can help each other. A larger group or club effort can be even better — someone with experience and test equipment can be an Elmer for the group.

In addition to the material here and in magazine columns, several excellent references for amateurs interested in working on microwave frequencies include:

• *Microwave Know How*, by Andy Barter, G8ATD (RSGB)

• *International Microwave Handbook*, by Andy Barter, G8ATD (RSGB)

• *The ARRL UHF/Microwave Experimenter's Manual* (out of print but available used)

• *The ARRL UHF/Microwave Projects* CD (out of print but available used)

23.7.1 Lead Lengths

Microwave construction is becoming more popular, but at these frequencies the size of physical component leads and PC-board traces cannot be neglected. Microwave construction techniques either minimize these stray values or make them part of the circuit design.

The basic consideration in microwave circuitry is short lead lengths, particularly for ground returns. Current always flows in a complete loop with a return path through the "ground" (see **Figure 23.34**). The loop must be much smaller than a wavelength for a circuit to work properly — as the frequency gets higher, dimensions must get smaller. One area that requires particular care to ensure good ground return continuity is the transition from coax cable outside a cabinet to a PC board inside the cabinet.

At microwave frequencies, the mechanical aspects and physical size of circuits become very much a part of the design. A few millimeters of conductor has significant reactance at these frequencies. This even affects VHF and HF designs in which the traces and conductors resonate on microwave frequencies. If a high-performance FET has lots of gain in this region, a VHF preamplifier might also function as a 10 GHz oscillator if the circuit stray reactances were just right (or wrong!). You can

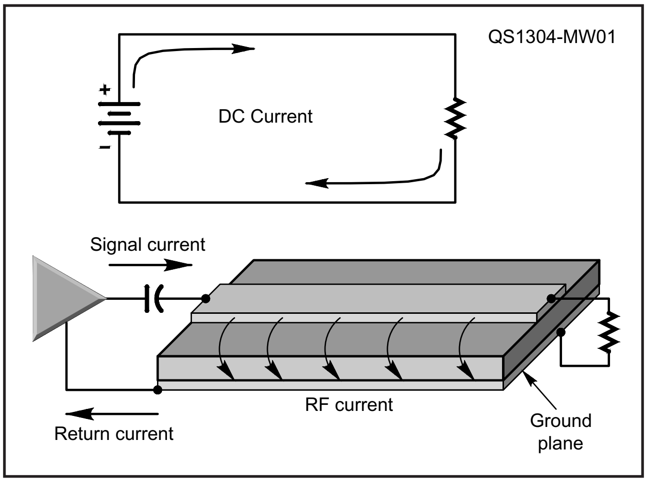


Figure 23.34— Current must always flow in a complete loop. This is particularly important at microwave frequencies where wavelengths are very short.

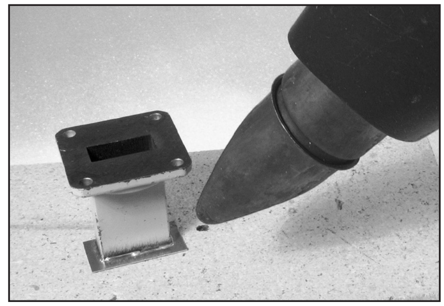


Figure 23.35 — Larger assemblies may be soldered with a heat gun. A torch is better for high-temperature brazing.

prevent this by using shields between the input and output, or by adding microwave absorptive material to the lid of the shielded module. (SHF Microwave sells absorptive materials.)

23.7.2 Metalworking

Waveguide construction requires some basic metalworking skills, which are also useful for assembly and packaging of microwave systems. Some minimal tools would be a hacksaw, drills, files, and layout tools. While a hand drill can suffice, an inexpensive drill press will make work more precise as well as safer. In metals, Dewalt Pilot Point drill bits make clean, accurate holes. Files and nibbling tools can make rectangular or odd-shaped holes.

Solder has a poor reputation for microwave losses — but, of course, it is essential. It does have higher resistance than copper or aluminum, but that only matters in locations of high current, for instance, the ground end of a high-Q quarter-wave resonator. Horn antennas and most waveguide structures have low Q so a clean solder joint, without excess blobs, has no effect. Traditional tin-lead solder works well; the silver solder with 1% silver has no performance advantage but looks pretty.

For things like waveguide and antennas that are too large for a soldering iron, a hot air gun does an excellent job. **Figure 23.35** shows the backshort of a waveguide transition as it is being soldered with a hot air gun. A small amount of paste flux (Kester SP-44) was applied to the waveguide end, and a ring of solder placed along the joint. Then heat is applied to the whole area until the solder melts and flows into the joint.

A torch can also be used for soldering, but tends to cause more oxidation of the metal. However, when a high-temperature brazed joint is desired, a torch is required. A MAPP gas torch and Silvaloy 15 (or Harris Stay-silv 15) silver brazing material, which requires no flux, are suitable for small and medium-sized components.

Many hams seem to think that silver plating is desirable, but it rarely makes a difference — it just looks much prettier. The only application in which silver plating has been shown to make a significant difference is in UHF cavities with sliding contacts. In most other places, copper losses are low enough so that little improvement is possible, and aluminum is fine for most uses.

23.7.3 Circuit Construction

Modular construction is a useful technique for microwave circuits. Often, circuits are tested by connecting their inputs and output to known 50 Ω sources and loads. Modules are typically kept small to prevent the chassis and PC board from acting as a waveguide, which provides a feedback path between the input and output of a circuit, resulting in instability.

PRINTED-CIRCUIT BOARDS AT MICROWAVES

A microwave printed circuit board (PCB) is typically double-sided, with transmission-line circuitry printed on one side and a ground plane on the far side. The ground return path is best provided by plated-thru holes (PTH) connecting the two sides with minimum length. Surface-mount components — integrated circuits, chip capacitors, and chip resistors — are soldered directly to printed pads and transmission lines, and to ground pads with embedded PTH. Many of the microwave integrated circuits come in really tiny packages, with lead pitch as small as 25 mils. See the section on Surface-Mount Technology earlier in this chapter for more information on this type of construction.

When using glass-epoxy PC board at microwave frequencies, the crucial board parameter is the thickness of the dielectric. It can vary quite a bit, in excess of 10%. This is not surprising: digital and lower-frequency analog circuits work just fine if the board is a little thinner or thicker than usual. Some of the board types used in microwave-circuit construction are a generic Teflon PC board, Duroid 5870 and 5880.

TRANSMISSION LINES AND CONNECTORS

From a mechanical accuracy point of view, the most tolerant type of construction is based on waveguide. Tuning is usually accomplished via one or more screws threaded into the waveguide. It becomes unwieldy to use waveguide on the amateur bands below 10 GHz because the dimensions get too large.

Proper connectors are a necessary expense at microwaves. At 10 GHz, the use of the proper connectors is essential for repeatable performance. Do not connect microwave circuits with coax and pigtails. It might work but it probably can't be duplicated. SMA connectors are common because they are small and work well. SMA jacks are sometimes soldered in place, although 2-56 hardware is more common.

At 24 GHz and above, even waveguide becomes small and difficult to work with. At these frequencies most readily available coax connectors work unreliably, so these higher bands are really a challenge. Special SMA connectors are available for use at 24 GHz.

DEVICE SUBSTITUTION

It is important to copy microwave circuits exactly, unless you really know what you are doing. "Improvements," such as better shielding or grounding, can sometimes cause poor performance. It isn't usually attractive to substitute components, particularly with the active devices. It may look possible to substitute different grades of the same wafer, such as the ATF13135 and the ATF13335, but these are really the same transistor with different performance measurements. While two transistors may have exactly the same gain and noise figure at the desired operating frequency, often the impedances needed to maintain stability at other frequencies can be different. Thus, the "substitute" may oscillate, while the proper transistor would work just fine.

You can often substitute MMICs (monolithic microwave integrated circuits) for one another because they are designed to be stable and operate with the same input and output impedances (50 Ω).

The size of components used at microwaves can be critical — in some cases, a chip resistor 80 mils across is not a good substitute for one 60 mils across. Hopefully, the author of a construction project tells you which dimensions are critical, but you can't always count on this — the author may not know. It's not unusual for a person to spend years building just one prototype, so it's not surprising that the author might not have built a dozen different samples to try possible substitutions.

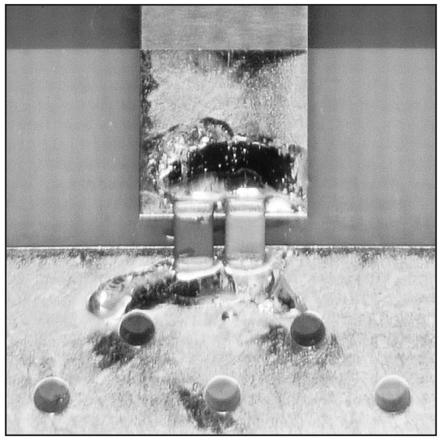


Figure 23.36 — Printed comb-line filter uses two chip capacitors in parallel to reduce loss.

23.7.4 Capacitors for Microwave Construction

Ordinary ceramic chip capacitors cost a few cents, while microwave chip capacitors are more than a dollar each. All of them have parasitic resistance and inductance, but the microwave versions use lower-loss materials that make a difference at higher microwave frequencies. Ordinary ceramic capacitors work fine in non-critical applications like blocking and bypass capacitors, even up to 10 GHz. In critical areas like low-noise or power amplifiers, microwave capacitors are preferred.

For applications requiring high-Q for low loss, like the printed comb-line filter in **Figure 23.36**, two ordinary capacitors in parallel as shown have lower loss than one expensive microwave capacitor. By paralleling, the parasitic resistances are also paralleled, cutting the resistance in half. An additional advantage is that combinations can be chosen to yield non-standard capacitance values. We might apply this trick to really demanding applications, like high-power solid-state amplifiers, by paralleling microwave capacitors to reduce losses.

23.7.5 Tuning and "No-Tune" Designs

Microwave construction does not always require tight tolerances and precision construction. A fair amount of error can often be tolerated if you are willing to tune your circuits, as you do at MF/HF. This usually requires the use of variable components that can be expensive and tricky to adjust.

Proper design and construction techniques using high precision can result in a "no-tune" microwave design. To build one of these no-tune projects, all you need do is buy the parts and install them on the board. The circuit tuning has been precisely controlled by the board and component dimensions, so the project should work.

One tuning technique you can use with a microwave design, if you have the suitable test equipment, is to use bits of copper foil or EMI shielding tape as "stubs" to tune circuits. Solder these small bits of conductor into place at various points in the circuit to make reactances that can actually tune a circuit. After their position has been determined as part of the design, tuning is accomplished by removing or adding small amounts of conductor, or slightly changing the placement of the tuning stub. The size of the foil needed depends on your ability to determine changes in circuit performance, as well as the frequency of operation and the circuit board parameters. A precision setup that lets you see tiny changes allows you to use very small pieces of foil to get the best tuning possible.

23.8 Tools and Their Use

All electronic construction makes use of tools, from mechanical tools for chassis fabrication to the soldering tools used for circuit assembly. A good understanding of tools and their uses will enable you to perform most construction tasks.

While sophisticated and expensive tools often work better or more quickly than simple hand tools, with proper use simple hand tools can turn out a fine piece of equipment. **Table 23.5** lists tools indispensable for construction of electronic equipment. These tools can be used to perform nearly any construction task. Add tools to your collection from time to time, as finances permit.

23.8.1 Sources of Tools

Electronic-supply houses, mail-order/web stores, and most hardware stores carry the tools required to build or service amateur radio equipment. Bargains are available at ham flea markets or local neighborhood sales, but beware! Some flea-market bargains are really shoddy and won't work very well or last very long. Some used tools are offered for sale because the owner is not happy with their performance.

There is no substitute for quality! A high-quality tool, while a bit more expensive, will last a lifetime. Poor quality tools don't last long and often do a poor job even when brand new. You don't need to buy machinist-grade tools, but stay away from cheap tools; they are not the bargains they might appear to be.

CARE OF TOOLS

The proper care of tools is more than a matter of pride. Tools that have not been cared for properly will not last long or work well. Dull or broken tools can be safety hazards. Tools that are in good condition do the work for you; tools that are misused or dull are difficult to use.

Store tools in a dry place. Tools do not fit in with most living room decors, so they are often relegated to the basement or garage. Unfortunately, many basements or garages are not good places to store tools; dampness and dust are not good for tools. If your tools are stored in a damp place, use a dehumidifier. Sometimes you can minimize rust by keeping your tools lightly oiled, but this is a second-best solution. If you oil your tools, they may not rust, but you will end up covered in oil every time you use them. Wax or silicone spray is a better alternative.

Store tools neatly. A messy toolbox in

Table 23.5

Recommended Tools and Materials

Simple Hand Tools

Screwdrivers

Slotted, 3-in, ⅛-in blade
Slotted, 8-in, ⅛-in blade
Slotted, 3-in, 3/16-in blade
Slotted, stubby, ¼-in blade
Slotted, 4-in, ¼-in blade
Slotted, 6-in, 5/16-in blade
Phillips, 2½-in, #0 (pocket clip)
Phillips, 3-in, #1
Phillips, stubby, #2
Phillips, 4-in, #2
Long-shank screwdriver with holding clip on blade
Jeweler's set
Right-angle, slotted and Phillips

Pliers, Sockets and Wrenches

Long-nose pliers, 6- and 4-in
Diagonal cutters, 6- and 4-in
Channel-lock pliers, 6-in
Slip-joint pliers
Locking pliers (Vise Grip or equivalent)
Socket nut-driver set, 1/16- to 1/2-in
Set of socket wrenches for hex nuts
Allen (hex) wrench set
Wrench set
Adjustable wrenches, 6- and 10-in
Tweezers, regular and reverse-action
Retrieval tool/parts holder, flexible claw
Retrieval tool, magnetic

Cutting and Grinding Tools

File set consisting of flat, round, half-round, and triangular. Large and miniature types recommended
Burnishing tool
Wire strippers
Wire crimper
Hemostat, straight
Scissors
Tin shears, 10-in
Hacksaw and blades
Hand nibbling tool (for chassis-hole cutting)
Scratch awl or scriber (for marking metal)
Heavy-duty jackknife
Knife blade set (X-ACTO or equivalent)
Machine-screw taps, #4-40 through #10-32 thread
Socket punches, ½ in, ⅝ in, ¾ in, 1⅛ in, 1¼ in, and 1½ in
Tapered reamer, T-handle, 1/2-in maximum width
Deburring tool

Miscellaneous Hand Tools

Combination square, 12-in, for layout work
Hammer, ball-peen, 12-oz head
Hammer, tack
Bench vise, 4-in jaws or larger
Center punch
Plastic alignment tools
Mirror, inspection
Flashlight, penlight and standard
Magnifying glass
Ruler or tape measure
Dental pick
Calipers
Brush, wire
Brush, soft
Small paintbrush
IC-puller tool

Hand-Powered Tools

Hand drill, 1/4-in chuck or larger
High-speed drill bits, #60 through ⅜-in diameter

Power Tools

Motor-driven emery wheel for grinding
Electric drill, hand-held
Drill press
Miniature electric motor tool (Dremel or equivalent) and accessory drill press

Soldering Tools and Supplies

Soldering station, adjustable temp, assorted tips
Soldering iron, 100 W or higher, ⅝-in tip
Soldering gun, 200 W or higher
Solder, 60/40, rosin core
Desoldering tool
Desoldering wick, ⅛-in and ¼-in width
Liquid flux, pen or bottle
Isopropyl alcohol for flux removal

Safety

Safety glasses
Hearing protector, earphones or earplugs
Fire extinguisher
First-aid kit

Useful Materials

Medium-weight machine oil
Contact cleaner, liquid or spray can
RTV sealant or equivalent
Electrical tape, vinyl plastic
Sandpaper, assorted
Emery cloth
Steel wool, assorted
Cleaning pad, Scotchbrite or equivalent
Cleaners and degreasers
Contact lubricant
Sheet aluminum, solid and perforated, 16- or 18-gauge, for brackets and shielding.
Aluminum angle stock, ½ × ½-in and ¼-in diameter round brass or aluminum rod (for shaft extensions)
Machine screws: Round-head and flat head, with nuts and lockwashers to fit. Most useful sizes: 4-40, 6-32 and 8-32, in lengths from 1/4-in to 1 1/2 in (Nickel-plated steel is satisfactory except in strong RF fields, where brass should be used.)
Bakelite, Lucite, polystyrene, and copper-clad PC-board scraps.
Soldering lugs, panel bearings, rubber grommets, terminal-lug wiring strips, varnished-cambric insulating tubing, heat-shrinkable tubing
Shielded and unshielded wire
Tinned bare wire, #22, #14 and #12
Enameled wire, #20 through #30

which tools are strewn about haphazardly can be more than an inconvenience. You may waste a lot of time looking for the right tool, and sharp edges can be dulled or nicked by tools banging into each other in the bottom of the box. As the old adage says, a place for every tool, and every tool in its place. If you must search the workbench, garage, attic, and car to find the right screwdriver, you'll spend more time looking for tools than building projects.

SHARPENING

Many cutting tools can be sharpened. Send a tool that has been seriously dulled to a professional sharpening service. These services can sharpen saw blades, some files, drill bits, and most cutting blades. Touch up the edge of cutting tools with a whetstone to extend the time between sharpening.

Sharpen drill bits frequently to minimize the amount of material that must be removed each time. Frequent sharpening also makes it easier to maintain the critical surface angles required for best cutting with least wear. Most inexpensive drill-bit sharpeners available for shop use do a poor job, either from the poor quality of the sharpening tool or the inexperience of the operator. Also, drills should be sharpened at different angles for different applications. Commercial sharpening services do a much better job.

INTENDED PURPOSE

Don't use tools for anything other than their intended purpose! If you use a pair of wire cutters to cut sheet metal, pliers as a vise, or a screwdriver as a pry bar, you ruin a good tool and sometimes the work piece as well. Although an experienced constructor can improvise with tools, most take pride in not abusing them. Having a wide variety of good tools at your disposal minimizes the problem of using the wrong tool for the job.

23.8.2 Tool Descriptions and Uses

Specific applications for tools are discussed throughout this chapter. Hand tools are used for so many different applications that they are discussed first, followed by some tips for proper use of power tools.

SCREWDRIVERS AND NUTDRIVERS

For construction or repair, you need to have an assortment of screwdrivers. Each blade size is designed to fit a specific range of screw head sizes. Using the wrong size blade usually damages the blade, the screw head, or both. You may also need stubby sizes to fit into tight spaces. Right-angle screwdrivers are inexpensive and can get into tight spaces that can't otherwise be reached.

Electric screwdrivers are relatively inexpensive and very useful, particularly for repetitive tasks. If you have a lot of screws to fasten, they can save a lot of time and effort. They come with a wide assortment of screwdriver and nutdriver bits. An electric drill can also function as an electric screwdriver, although it may be heavy and over-powered for many applications.

Keep screwdriver blades in good condition. If a blade becomes broken or worn out, replace the screwdriver. A screwdriver only costs a few dollars; do not use one that is not in perfect condition. Save old screwdrivers to use as pry bars and levers, but use only good ones on screws. Filing a worn blade seldom gives good results.

Nutdrivers, the complement to screwdrivers, are often much easier to use than a wrench, particularly for nuts smaller than 3/8 inch. They are also less damaging to the nut than any type of pliers, with a better grip on the nut. Nutdrivers also minimize the chances of damage to front panels when tightening the nuts on control shafts. A set of interchangeable nutdrivers with a shared handle is a very handy addition to the toolbox.

PLIERS AND LOCKING-GRIP PLIERS

Pliers and locking-grip pliers are used to hold or bend things. They are not wrenches! Using pliers to remove a nut or bolt usually damages the nut or the pliers. To remove a nut, use a wrench or nutdriver. There is one exception to this rule of thumb: To remove a nut that is stripped too badly for a wrench, use a pair of pliers, locking-grip pliers, or a diagonal cutter to bite into the nut and start it turning. Reserve an old tool or one dedicated to just this purpose, which can damage the tool.

Pliers are not intended for heavy-duty applications. Use a metal brake to bend heavy metal; use a vise to hold a heavy component. If the pliers' jaws or teeth become worn, replace the tool.

There are many different kinds of fine pliers, usually called "needle-nose" pliers or something similar, that are particularly useful in electronics work. These are intended for light jobs, such as bending or holding wires or small work pieces. Two or three of these tools with different sizes of jaws will suffice for most jobs.

WIRE CUTTERS AND STRIPPERS

Wire cutters are primarily used to cut wires or component leads. The choice of blade style depends on the application. Diagonal blades or "dikes" are most often used to cut wire. Some delicate components can be damaged by cutting their leads with dikes because of the abrupt shock of the cut. Scissors or shears designed to cut wire should be used instead.

Specialized wire cutters are available to trim wires leads on circuit boards. These cutters are often called "flush cutters." Their cutting end is *not* designed to cut thicker wires. Use them *only* to clip smaller gauge wires, such as that on components used in circuits.

Wire strippers are available in manual and automatic styles. The manual strippers have a series of holes designed to remove insulation from a specific gauge of wire. Using the holes that are too big or too small will create nicks in the wire, which usually leads to the wire breaking at the nick. Automatic strippers grab and hold the wire for a consistent strip — some even judge the wire thickness automatically. If you strip a lot of wires, an automatic stripper may be worth the extra expense.

Wire strippers are handy, but with a little practice you can usually strip wires using a diagonal cutter or a knife. This is not the only use for a knife, so keep an assortment handy. Do not use wire cutters or strippers on anything other than wire! If you use a cutter to trim a protruding screw head or cut a hardened-steel spring, you will usually damage the blades.

Table 23.6
Numbered Drill Sizes

No.	*Diameter (Mils)*	*Will Clear Screw*	*Drilled for Tapping from Steel or Brass*
1	228.0	12-24	—
2	221.0	—	—
3	213.0	—	14-24
4	209.0	12-20	—
5	205.0	—	—
6	204.0	—	—
7	201.0	—	—
8	199.0	—	—
9	196.0	—	—
10	193.5	—	—
11	**191.0**	**10-24** **10-32**	—
12	189.0	—	—
13	185.0	—	—
14	182.0	—	—
15	180.0	—	—
16	177.0	—	12-24
17	173.0	—	—
18	169.5	—	—
19	166.0	8-32	12-20
20	161.0	—	—
21	**159.0**	—	**10-32**
22	157.0	—	—
23	154.0	—	—
24	152.0	—	—
25	**149.5**	—	**10-24**
26	147.0	—	—
27	144.0	—	—
28	**140.0**	**6-32**	—
29	**136.0**	—	**8-32**
30	128.5	—	—
31	120.0	—	—
32	116.0	—	—
33	**113.0**	**4-40**	—
34	111.0	—	—
35	110.0	—	—
36	**106.5**	—	**6-32**
37	104.0	—	—
38	101.5	—	—
39	**099.5**	**3-48**	—
40	098.0	—	—
41	096.0	—	—
42	093.5	—	—
43	**089.0**	—	**4-40**
44	**086.0**	**2-56**	—
45	082.0	—	—
46	081.0	—	—
47	078.5	—	3-48
48	076.0	—	—
49	073.0	—	—
50	**070.0**	—	**2-56**
51	067.0	—	—
52	063.5	—	—
53	059.5	—	—
54	055.0	—	—

FILES

Files are used for a wide range of tasks. In addition to enlarging holes and slots, they are used to remove burrs, shape metal, wood, or plastic and clean some surfaces in preparation for soldering. Files are especially prone to damage from rust and moisture. Keep them in a dry place. The cutting edge of the blades can also become clogged with the material you are removing. Use file brushes (also called file cards) to keep files clean. Most files cannot be sharpened easily, so when the teeth become worn the file must be replaced.

DRILL BITS

Drill bits are made from carbon steel, high-speed steel, or carbide. Carbon steel is more common and is usually supplied unless a specific request is made for high-speed bits. Carbon steel drill bits cost less than high-speed or carbide types; they are sufficient for most equipment construction work. Carbide drill bits last much longer under heavy use. One disadvantage of carbide bits is that they are brittle and break easily, especially if you are using a handheld power drill. When drilling abrasive material such as fiberglass, the carbide bits last much longer than the steel bits.

Twist drills are available in a number of sizes listed in **Table 23.6**. Those listed in bold type are the most commonly used in construction of amateur equipment. You may not use all of the drills in a standard set, but it is nice to have a complete set on hand. You should also buy several spares of the more common sizes. Although Table 23.6 lists drills down to #54, the series extends to number #80. While the smaller sizes cannot usually be found in hardware stores or home improvement stores, they are commonly available through industrial tool suppliers and various sources on the Internet.

A "step drill" consists of multiple drill diameters stacked as one bit, looking somewhat like a Christmas tree. These bits are very useful for drilling metal case material used in radio projects, as the fluted edge of the bit in between sizes removes most if not all burrs from the hole you are drilling. Use them carefully and slowly to take advantage of their ability to remove burrs. A step drill also has the advantage of being able to drill a number of different standard size holes without having to change the bit.

SPECIALIZED TOOLS

Most constructors know how to use common tools, such as screwdrivers, wrenches, and hammers. Although specialized tools usually do a job that can be done with other tools, once the specialty tool is used you will wonder how you ever did the job without it! Let's discuss other tools that are not so common.

A hand nibbling tool is shown in Figure 23.37A. Use this tool to remove small "nibbles" of metal. It is easy to use; position the tool where you want to remove metal and squeeze the handle. The tool takes a small bite out of the metal. When you use a nibbler, be careful that you don't remove too much metal, clip the edge of a component mounted to the sheet metal, or grab a wire that is routed near the edge of a chassis. Fixing a broken wire is easy, but something to avoid if possible. It is easy to remove metal but nearly impossible to put it back. Do it right the first time!

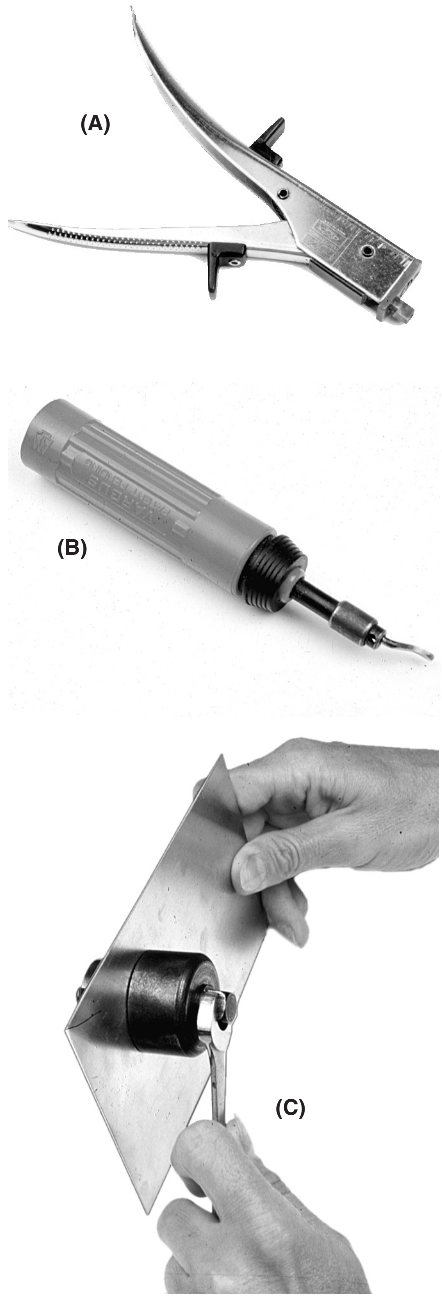


Figure 23.37 — At A, a nibbling tool is used to remove small sections of sheet metal. At B, a deburring tool is used to remove the burrs left after drilling a hole. At C, a socket punch is used to punch a clean, round hole in sheet metal.

Deburring Tool

A deburring tool (see **Figure 23.37B**) is just the thing to remove the sharp edges left on a hole after drilling or punching operations. Position the tool over the hole and rotate it around the edge of the hole to remove burrs or rough edges. As an alternative, select a drill bit that is somewhat larger than the hole, position it over the hole, and spin it lightly to remove the burr. Be sure to deburr both sides of the hole.

Socket or Chassis Punches

Greenlee is the most widely known of the socket-punch manufacturers. Most socket punches are round, but they do come in other shapes. To use one, drill a pilot hole large enough to clear the bolt that runs through the punch. Then, mount the punch as shown in Figure 23.37C, with the cutter on one side of the sheet metal and the socket on the other. Tighten the nut with a wrench until the cutter cuts all the way through the sheet metal. These punches are often sold in sets at a significant discount to the same punches purchased separately. Hand-punches that operate by squeezing will also cut small holes by hand in light-gauge sheet metal, printed-circuit boards, and plastic.

Crimping Tools

The use of crimped connectors is common in the electronics industry. In many commercial and aerospace applications, soldered joints are no longer used. Hams have been reluctant to adapt crimped connections, largely due to mistrust of contacts that are not soldered, the use of cheap crimp connectors on consumer electronics, and the high cost of quality crimping tools or "crimpers." If high quality connectors and tools are used, the crimped connector will be as reliable a connection as a soldered one. The crimped connection is easier to make than a soldered one in most cases.

Crimped coaxial connectors are the most common crimped connector. MIL-spec or equivalent crimp connectors are available for the UHF, BNC, F, and N-series MIL-spec connectors. Power connectors, such as the Anderson PowerPoles and Molex connectors, are probably the second most commonly used crimped connections.

When purchasing a crimper, look for a ratcheting model with dies that are intended for the connectors you will be using. The common pliers-type crimper designed for household electrical terminals will have trouble crimping power connectors and is unsuitable for coaxial connectors. A good ratcheting crimper can be obtained for $50 to $100 with the necessary interchangeable dies. Large ratcheting crimpers suitable for the larger coaxial connectors can cost several hundred dollars. A good crimper and set of dies is an excellent investment for a club or group of like-minded hams.

Compression tools are now the most common method of installing Type F connectors. (See the **Transmission Lines** chapter.) BNC, TNC, and SMA connectors now can be installed using compression connectors and tools, as well. Quad-shielded RG-6 is the most common cable used for this purpose, and a compression tool makes the job very easy. A compression crimp tool works differently as it presses an internal sleeve into the body of the connector to make contact with the shield. Most compression tools are lower in cost than conventional ratcheting tools, and are available at most home improvement stores. You will need to purchase compression connectors designed to work with the tool and cable type you selected. Installation instructions are usually provided on the tool's package and with packages of connectors.

Useful Shop Materials

Small stocks of various materials are used when constructing electronics equipment. Most of these are available from hardware or radio supply stores. A representative list is shown at the end of Table 23.5.

Small parts such as machine screws, nuts, washers, and soldering lugs can be economically purchased in large quantities (it doesn't pay to buy more than a lifetime supply). For items you don't use often, many radio supply stores or hardware stores sell small quantities and assortments. Stainless steel hardware can be kept on hand for outdoor use.

Tuning and Alignment Tools

It's helpful to have an assortment of special tools for adjusting variable capacitors, inductors, and potentiometers. See the section Tuning and Alignment earlier in this chapter.

23.9 Mechanical Fabrication

Most projects end up in some sort of an enclosure, and most hams choose to purchase a ready-made chassis for small projects, but some projects require a custom enclosure. Even a ready-made chassis may require a fabricated sheet-metal shield or bracket, so it's good to learn something about sheet-metal and metal-fabrication techniques.

Most often, you can buy a suitable enclosure. These are sold by most electronics distributors. Select an enclosure that has plenty of room. A removable cover or front panel can make any future troubleshooting or modifications easy. A project enclosure should be strong enough to hold all of the components without bending or sagging; it should also be strong enough to stand up to expected use and abuse.

23.9.1 Cutting and Bending Sheet Metal

Enclosures, mounting brackets, and shields are usually made of sheet metal. Most sheet metal is sold in large sheets, 4 to 8 feet or larger. It must be cut to the size needed.

Most sheet metal is thin enough to cut with metal shears or a hacksaw. A jigsaw or band saw makes the task easier. If you use any kind of saw, select a blade that has teeth fine enough so that at least two teeth are in contact with the metal at all times.

If a metal sheet is too large to cut conveniently with a hacksaw, it can be scored and broken. Make scratches as deep as possible along the line of the cut on both sides of the sheet. Then, clamp it in a vise and work it back and forth until the sheet breaks at the line. Do not bend it too far before the break begins to weaken, or the edge of the sheet might bend. A pair of flat bars, slightly longer than the sheet being bent, make it easier to hold a sheet firmly in a vise. Use "C" clamps to keep the bars from spreading at the ends.

Smooth rough edges with a file or by sanding with a large piece of emery cloth or sandpaper wrapped around a flat block.

23.9.2 Finishing Aluminum

Give aluminum chassis, panels, and parts a sheen finish by treating them in a caustic bath. (See the information on chemical safety at the beginning of this chapter.) Use a plastic container to hold the solution and wear both safety goggles and protective clothing while treating aluminum. Ordinary household lye can be dissolved in water to make a bath solution. Follow the directions on the container. A strong solution will do the job more rapidly.

Stir the solution with a non-metal utensil until the lye crystals are completely dissolved. If the lye solution gets on your skin, wash with plenty of water. If you get any in your eyes, immediately rinse with plenty of clean, room-temperature water and seek medical help. It can also damage your clothing, so wear something old. Prepare sufficient solution to cover the piece completely. When the aluminum is immersed, a very pronounced bubbling takes place. Provide ventilation to disperse the escaping gas. A half hour to two hours in the bath is sufficient, depending on the strength of the solution and the desired surface characteristics.

23.9.3 Chassis Working

With a few essential tools and proper procedure, building radio gear on a metal chassis is a relatively simple matter. Aluminum is better than steel, not only because it is a superior shielding material, but also because it is much easier to work and provides good chassis contact when used with secure fasteners.

Spend sufficient time planning a project to save trouble and energy later. The actual construction is much simpler when all details are worked out beforehand. Here we discuss a large chassis-and-cabinet project, such as a high-power amplifier. The techniques are applicable to small projects as well.

Cover the top of the chassis with a piece of wrapping paper or graph paper. Fold the edges down over the sides of the chassis and fasten them with adhesive tape. Place the front panel against the chassis front and draw a line there to indicate the chassis top edge.

Assemble the parts to be mounted on the chassis top and move them about to find a satisfactory arrangement. Consider that some will be mounted underneath the chassis and ensure that the two groups of components won't interfere with each other.

Place controls with shafts that extend through the cabinet first, and arrange them so that the knobs will form the desired pattern on the panel. Position the shafts perpendicular to the front chassis edge. Locate any partition shields and panel brackets next, then sockets and any other parts. Mark the mounting-hole centers of each part accurately on the paper. Watch out for capacitors with off-center shafts that do not line up with the mounting holes. Do not forget to mark the centers of socket holes and holes for wiring leads. Make the large center hole for a socket *before* the small mounting holes. Then use the socket itself as a template to mark the centers of the mounting holes. With all chassis holes marked, center-punch and drill each hole.

Next, mount on the chassis the capacitors and any other parts with shafts extending to the panel. Fasten the front panel to the chassis temporarily. Use a machinist's square to extend the line (vertical axis) of any control shaft to the chassis front and mark the location on the front panel at the chassis line. If the layout is complex, label each mark with an identifier. Also mark the back of the front panel with the locations of any holes in the chassis front that must go through the front panel. Remove the front panel.

Repurposing Obsolete Equipment Cabinets

One of the best deals at a hamfest or flea market might not be anything having to do with radio — yet! Obsolete industrial and commercial equipment may not be useable in the ham station, but their metal enclosures are top-quality. Purchasing one of these cabinets new would cost hundreds of dollars, but the surplus equipment is often sold very cheaply. Reusing the cabinets is easy, and stripping the electronics often results in a large collection of hardware and electronic components that can be re-used in ham equipment. The overall process is illustrated in a January/February 2017 *QEX* article by Scott Roleson, KC7CJ, included with this book's online material. Keep an eye out for these bargains everywhere, even at garage sales!

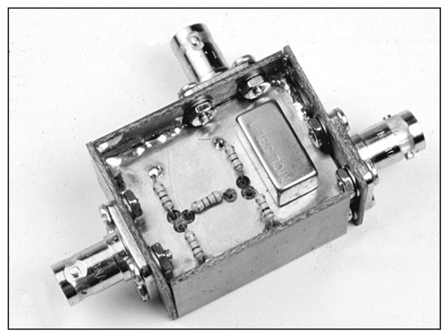


Figure 23.38 — An enclosure made entirely from PC-board stock.

MAKING ENCLOSURES WITH PC BOARD MATERIAL

Much tedious sheet-metal work can be eliminated by fabricating chassis and enclosures from copper-clad printed-circuit board material. While it is manufactured in large sheets for industrial use, some hobby electronics stores and surplus outlets market usable scraps at reasonable prices. PC-board stock cuts easily with a small hacksaw. The non-metallic base material isn't malleable, so it can't be bent. Corners are easily formed by holding two pieces at right angles and soldering the seam. This technique makes excellent RF-tight enclosures. If mechanical rigidity is required of a large copper-clad surface, solder stiffening ribs at right angles to the sheet.

Figure 23.38 shows the use of PC-board stock to make a project enclosure. This enclosure was made by cutting the pieces to size, then soldering them together. Start by laying the bottom piece on a workbench, then putting one of the sides in place at a right angle. Tack-solder the second piece in two or three places, then start at one end and run a bead of solder down the entire seam. Use plenty of solder and plenty of heat. Continue with the rest of the pieces until all but the top cover is in place.

In most cases, it is better to drill all needed holes in advance. It can sometimes be difficult to drill holes after the enclosure is soldered together.

You can use this technique to build enclosures, subassemblies, or shields. This technique is easy with practice; hone your skills on a few scrap pieces of PC-board stock.

23.9.4 Drilling Techniques

Before drilling holes in metal with a hand drill, indent the hole centers with a center punch. This prevents the drill bit from "walking" away from the center when starting the hole. Predrill holes greater than ½ inch in diameter with a smaller bit that is large enough to contain the flat spot at the large bit's tip. When the metal being drilled is thinner than the depth of the drill-bit tip, back up the metal with a wood block to smooth the drilling process.

The chuck on the common hand drill is limited to ⅜ inch bits. Some bits are much larger, with a ⅜ inch shank. If necessary, enlarge holes with a reamer or round file. For very large or odd-shaped holes, drill a series of closely spaced small holes just inside of the desired opening. Cut the metal remaining between the holes with a cold chisel and file or grind the hole to its finished shape. A nibbling tool also works well for such holes.

Use socket-hole punches to make socket holes and other large holes in an aluminum chassis. Drill a guide hole for the punch center bolt, assemble the punch with the bolt through the guide hole, and tighten the bolt to cut the desired hole. Oil the threads of the bolt occasionally.

Cut large circular holes in steel panels or chassis with an adjustable circle cutter ("fly cutter") in a drill press at low speed. Occasionally apply machine oil to the cutting groove to speed the job. Test the cutter's diameter setting by cutting a block of wood or scrap material first.

Remove burrs or rough edges that result from drilling or cutting with a burr-remover, round or half-round file, a sharp knife, or a chisel. Keep an old chisel sharpened and available for this purpose.

23.9.5 Construction Notes

If a control shaft must be extended or insulated, a flexible shaft coupling with adequate insulation should be used. Satisfactory support for the shaft extension, as well as electrical contact for safety, can be provided by means of a metal panel bushing made for the purpose. These can be obtained singly for use with existing shafts, or they can be bought with a captive extension shaft included. In either case the panel bushing gives a solid feel to the control. The use of fiber washers between ceramic insulation and metal brackets, screws, or nuts will prevent the ceramic parts from breaking.

PAINTING

Painting is an art, but, like most arts, successful techniques are based on skills that can be learned. The surfaces to be painted must be clean to ensure that the paint will adhere properly. In most cases, you can wash the item to be painted with soap, water, and a mild scrub brush, then rinse thoroughly. When it is dry, it is ready for painting. Avoid touching it with your bare hands after it has been cleaned. Your skin oils will interfere with paint adhesion. Wear rubber or clean cotton gloves.

Sheet metal can be prepared for painting by abrading the surface with medium-grade sandpaper, making certain the strokes are applied in the same direction (not circular or random). This process will create tiny grooves on the otherwise smooth surface. As a result, paint or lacquer will adhere well. On aluminum, one or two coats of zinc chromate primer applied before the finish paint will ensure good adhesion.

Keep work areas clean and the air free of dust. Any loose dirt or dust particles will probably find their way onto a freshly painted project. Even water-based paints produce some fumes, so properly ventilate work areas.

Select paint suitable to the task. Some paints are best for metal, others for wood, and so on. Some dry quickly, with no fumes; others dry slowly and need to be thoroughly ventilated. You may want to select rust-preventative paint for metal surfaces that might be subjected to high moisture or salts.

Most metal surfaces are painted with some sort of spray, either from a spray gun or from spray cans of paint. Either way, follow the manufacturer's instructions for a high-quality job.

PANEL LAYOUT AND LABELING

There are many ways to layout and label a panel. Some builders don't label any controls or jacks, relying on memory to figure what does what. Others use a marking pen to label controls and inputs. Decals and dry transfers have long been a staple of home brewing. Label makers that print on clear or colored tape are used by many.

With modern computers and available software, it is not hard to lay out professional looking panels. One can use a standard drawing program for the layout. The grids available on these drawing programs are sufficient to make sure that everything is lined up squarely. If the panel label is laid out before the panel is drilled for controls, a copy of the label can be used as a drill template.

Computer-aided design (CAD) programs can also be used to lay out and label panels, although they can have a steep learning curve and may be overkill for many applications.

Surplus meters often find their way into projects. Unfortunately the meter faces usually do not have an appropriate scale for the project at hand. Relabeling meters has long been a mainstay to make home brew gear look professional. With the advent of computers this job has been made very easy. A software package, *MeterBasic*, by Jim Tonne, W4ENE, available with the online content, is very easy to use and results in professional looking meters that indicate exactly what you want them to indicate.

23.10 3D Printing

This section is extracted from the in-depth article "Introduction to 3D Printing for Hams" by John Portune, W6NBC, in the online material for this book. Detailed instructions on how to do 3D printing are not the intent of this section — 3D printing is too broad a topic. The object of this section is to define basic terms and concepts, present the fundamentals of 3D printing, and discuss common materials used for ham applications. Several articles with examples of 3D printed items for amateur radio use are included in the online information, and a list of references is included at the end of this section.

23.10.1 What is 3D Printing?

Fabricating one-piece physical objects has traditionally been a *subtractive* process. That is, the builder starts with a solid volume of material and gradually removes or subtracts material until it is in the final form. An *additive* process, however, builds the object by depositing or *printing* a sequence of layers of material, usually as 2D layers called *slices*, each adhering to the previous layer, until the entire 3D object has been constructed or *printed*.

For the home hobbyist, the most widely used method is *Fused Deposition Modeling (FDM)*, also known as *Fused Filament Fabrication (FFF)*. It is affordable, easy to learn, and offers high print quality. An FDM 3D printer creates an object by depositing fine strings of plastic in layers. The strings are created by heating the plastic until it is soft enough to be forced out of or *extruded* from a nozzle. The softened plastic immediately sticks (*fuses*) to the previously deposited layers below.

23.10.2 Understanding the FDM 3D Printer

THE EXTRUDER

Figure 23.39 shows a typical home 3D printer extruder. Plastic filament (commonly 1.75 mm in diameter) is pushed into the top (cold end) of the extruder by knurled rollers at a controlled rate. At the bottom (hot end), a temperature-controlled heating block softens the filament to make it sticky. Then, under pressure from more incoming filament, the soft plastic extrudes through a tiny hole in the extruder's nozzle to fuse to the 3D object forming on the build plate below.

A typical extruder nozzle diameter is 0.4 mm, slightly less than 1/64 inch. Nozzle diameter and the material from which the nozzle is made can be changed to suit the type of plastic and the detail of the print. Fineness of the extruded string is also affected by the

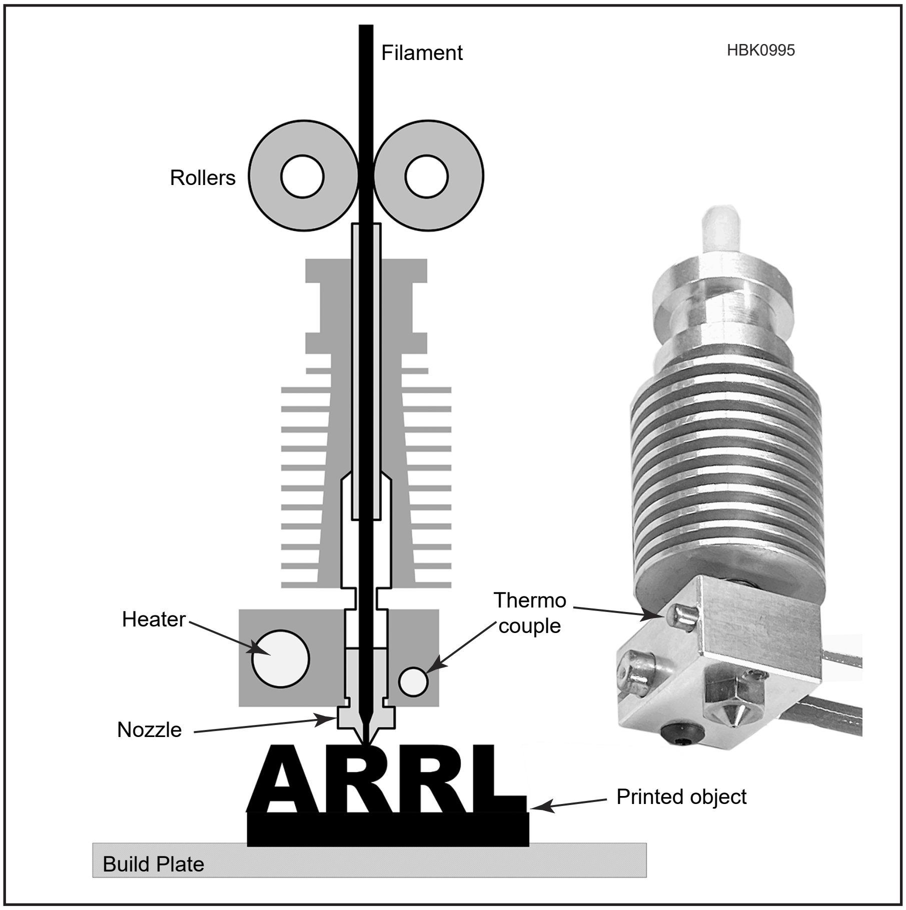


Figure 23.39 — A typical FDM extruder.

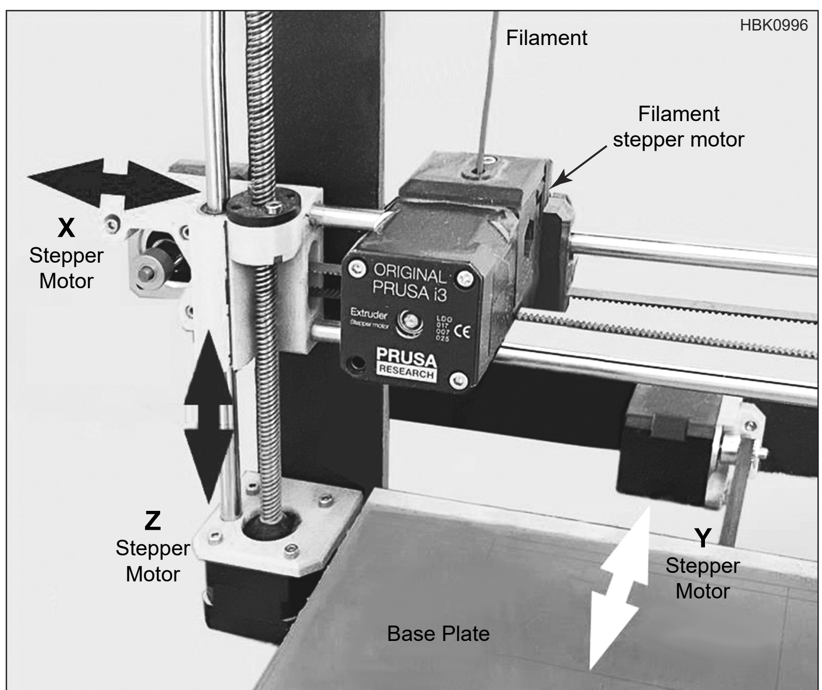


Figure 23.40 — Simplified pictorial of the X, Y, and Z stepper motors and extruder (filament) stepper motor of the popular Prusa i3 MK3S 3D printer.

speed with which the bead is extruded.

THE BUILD PLATE

Most *build plates* have an aluminum base, but the extruded plastic is not deposited directly on the base. Various surfaces are added, such as a flat glass plate, a self-adhesive plastic film, or one or two types of tape. Further, an adhesive is commonly applied to make sure the first layer stays in place. A well-adhered first layer is a critical factor in obtaining a high-quality print and in avoiding *failed* or *mis-prints* in which the object is distorted, extruded plastic fails to fuse properly, and so on.

The build plate on many FDM printers is heated to facilitate adhesion of the first layer and also to reduce warping of the object during the print. It also increases the variety of different plastics that can be used and reduces the number of failed prints.

COMPUTER NUMERIC CONTROL (CNC)

The motions of the 3D printer's extruder and values of other parameters, such as temperature and extrusion rates, are controlled by a microcomputer from a programmed sequence of positions and values in numeric form. This physical/electronic architecture is *Computer Numeric Control* (*CNC*).

To be able to print in three dimensions (X, Y, and Z), the printer must be able to rapidly move the extruder from side to side (X), front to back (Y) and up and down (Z). The extruder typically moves only in the X and Z dimensions; the build plate moves in the Y direction. (See **Figure 23.40**.)

A 3D printer's X, Y, and Z motion is generally produced by *stepper motors*. Stepper motors or just "steppers" are a specialized type of motor designed to move in tiny precise increments. Most 3D printers use three steppers to provide the X, Y, and Z motions. A fourth stepper feeds the plastic filament into the extruder at a computer-controlled rate. Notched or toothed belts, pulleys, and threaded lead screws transfer the controlled rotations of the stepper motors to the build plate, the extruder, and the rollers feeding filament into the extruder.

G-CODE

The sequence of CNC motions and a number of other machine parameter values are encoded as text instructions called *G-code*. The G-code file is loaded into the 3D printer's memory for each print. It is the complete set of instructions to the printer for printing a finished object. G-code for most home 3D printers today is a customized subset of industry-standard G-code for CNC machines and is controlled by the standard RS-274. The complete set of industry standard G-code commands can be found at **en.wikipedia.org/wiki/G-code.**

23.10.3 3D Object Design Software

In common practice, the G-code file is created in two steps by separate software programs. The first is 3D design software, which creates a computer model of an object to be printed. The result is saved as an *STL* file or an *OBJ* file.

The STL file format (*Standard Triangle* or *Tessellation Language*) is the most common format for 3D printing models. STL is the native file format of CAD software created by 3D Systems. It uses the X, Y, and Z coordinates of a series of interconnected triangles (tessellations) to represent the surfaces of the model. All modern CAD software programs will export an STL file. The OBJ format (short for "object") supports multi-color printing and is used for printing more complex objects. You can find out more about STL and OBJ formats at **3dinsider.com/stl-vs-obj**. Also see **all3dp.com/1/obj-file-format-3d-printing-cad/** for more in-depth discussion and a section on resources for obtaining the files. A list of websites that offer free STL files is included in the full article, part of the online information.

The second program is a called a *slicer* because it converts the 3D object model into the layers or slices needed for 3D printing. Unlike the generic 3D object design software, the slicer is specific to each model of printer. The slicer converts the STL file, or other native 3D design output file, into the G-code required by the individual printer. Being printer specific, the slicer's output G-code is not suitable for sharing between different brands or models of printers.

3D DESIGN SOFTWARE — TINKERCAD

Easy-to-use, free hobbyist-level 3D object design software, suitable for most ham projects, is readily available on the internet. As an illustration of a suitable program for ham 3D printing newcomers, we will examine the *Tinkercad* package. It is perhaps the most widely used 3D object design program for beginners. There are many tutorials for it on the internet. It was developed by AutoDesk and is cloud-based. To obtain a copy, create a free account at **www.tinkercad.com**. Finished projects may be saved on the *Tinkercad* cloud or a local computer for later use or modification, either as a project file or an STL file. 3D printed ham project objects are typically combinations of geometric shapes called *primitives* — rectangular solids, cylinders, spheres, cones, pyramids, and so on — that are combined to make complex single objects. One can accurately size, scale, rotate, and align objects in *Tinkercad* in units of millimeters or inches. All of the primitive objects can be subtracted from other objects, for example, to make a hole. The workspace is also easily zoomed and rotated for easy viewing.

SLICER SOFTWARE

The second step in creating the G-code file for the 3D printer is performed by the slicer software. It converts the X, Y, and Z coordinates of the tessellated triangles in an STL file, called a "mesh," into the G-code instructions that the printer needs for each print. Configuration settings for the 3D printer can also be made in the slicer program and included in the G-code file. It is highly advisable at first to stay with the default settings of the slicer. After printing a test object, the presets of the slicer can be changed to improve the final print quality or to correct errors.

Configuration settings for the slicer software itself can also be changed to suit a particular material or object. Many free test ob-

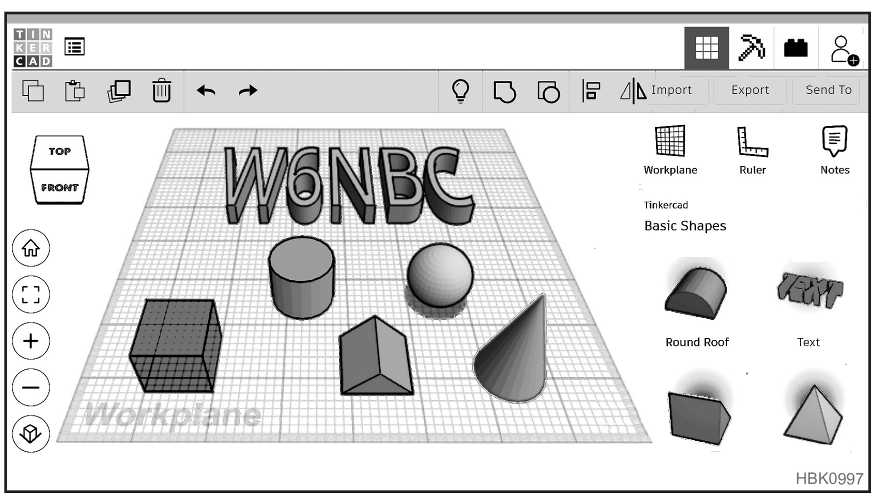


Figure 23.41 — Workspace of *Tinkercad* showing "primitive" geometric shapes and text used to create a 3D printer object and an STL file.

ject STL files are available on the web to help make adjustments. (See the directory of STL file websites in the online article.) A simple test object such as a cube is recommended for initial tests. For more advanced testing and calibration, the best-known test object is a small tugboat called Benchy, available from **www.3dbenchy.com**. Other test objects are available online for specific needs, such as minimizing stringing, which is a common cause of poor prints and printer crashes.

NORMAL SLICER ADJUSTMENTS

There are a number of required inputs to the slicer for every object. The working screen of an example slicer program is shown in **Figure 23.42**. From the top of the selection area at the right of the window:

• PRINT SETTINGS: The user may choose nozzle diameters and print quality modes. This sets the size of the plastic bead to be extruded and the thickness of the layers or slices.

• FILAMENT (Plastic Type): Slicers contain a list of plastic filaments with the correct temperature settings for the extruder, the heated build plate, and a number of other plastic and printer specific settings.

• SUPPORTS: The slicer can be instructed to add *supports* that can be removed after the print is finished. Printers can *bridge* over small open horizontal spaces, typically on the order of 1/4 inch (6 mm).

• IN-FILL DENSITY: Solid shells are printed on the surfaces of the object, but a honeycomb-like gridwork is printed internally. The density of the honeycomb is set for each object from 0% (hollow) to 100% (solid).

• BRIM: For objects that will have only a small surface touching the build plate, set the slicer to generate a *brim* or *skirt*. It is several rows of plastic bead printed around the periphery of, and in light contact with, the first layer of the object.

• COLOR AND FILAMENT CHANGE: The slicer for a single-extruder printer can be set to halt at a particular slice for the filament to be changed. Afterward, the print seamlessly resumes.

At the top of the workspace, on the left side of the window, are controls to add, delete, copy and paste, and to arrange multiple objects on the build plate. At the left edge of the workspace are controls to change the position and rotation of the object, set the object on the build plate, and make limited modification to the object.

23.10.4 Choosing a Printer

The most worthwhile guide is the internet. YouTube is an exceptionally valuable resource, both for learning 3D printing and for evaluating printers, accessories, and materials. Spend time searching on 3D printing in general, on manufacturer's websites, and in 3D printing magazines and blogs. Ham friends and radio club members who already own 3D printers are also an excellent resource. Magazines often publish lists of the top performing printers at that time. Also look at the comments and reviews from readers. (Wikipedia publishes a list of "notable" printer manufacturers at **en.wikipedia.org/wiki/List_of_3D_printer_manufacturers#0–9**.)

If there are a large number of videos on the printer and design software you are considering, and if the video makers find these reliable and easy to use, they are probably acceptable choices. The same is true for 3D design software and for the reputation of printer manufacturers.

Generally, medium-range domestic printers, currently in the $500 to $1000 range, are the best choice for the majority of ham buyers. At the low end of the price range, there are a rapidly growing number of less-expensive 3D printers, some in the low-$100 range. Some are excellent and new models appear frequently. Some companies, including medium-range manufacturers, offer lower-cost and junior-model printers.

It is important to consider how large an object can be printed. In 3D printing this is called the *print volume*. Low-priced printers generally print only small objects. Other important features, often not found on low-priced models, are a heated build plate and automatic build plate leveling. Most experienced 3D printers consider these two features to be necessities.

23.10.5 Plastic Filament for 3D Printing

Essential to the hobbyist entering 3D printing is a familiarity with the commonly available plastic filaments. The best choice for each project will depend on the nature of the project. **Table 23.7** is a quick guide, sorted by ease of printability. The two filaments that stand out for printability are PLA and PETG. This accounts for their popularity, and they are good choices for beginners.

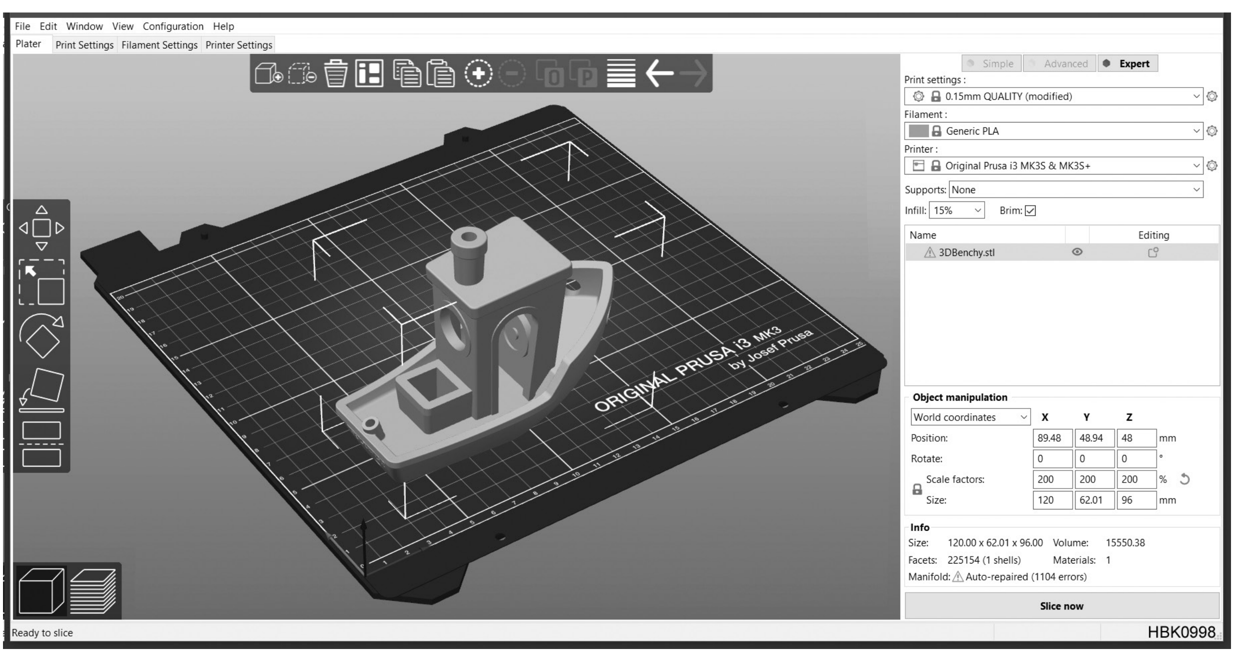


Figure 23.42 — Workspace of the slicer program for a Prusa i3 MK3S printer with a single "Benchy" test object loaded.

Table 23.7
Printing Properties of 3D Filament Plastics

	Printability	*$ / kg*	*Strength*	*Rigidity*	*Durability*	*Max. Temp.*
PLA	9	10-40	65	7.5	4	52C/126F
PETG	9	20-60	53	5	8	73C/163F
ABS	8	10-40	40	5	8	98C/208F
Nylon	8	25-65	62	5	10	95C/203F
Carbon fiber filled	8	30-80	46	10	3	52C/126F
Polypropylene	8	60-120	32	4	9	100C/212F
Wood filled	8	25-50	46	8	3	52C/126F
ASA	7	35-40	55	5	10	95C/203F
Metal filled	7	50-120	25	10	4	52C/126F
Flexible	6	30-70	35	1	9	65C/149F
HIPS	6	25-35	32	10	7	100C/212F
Polycarbonate	6	40-75	32	6	10	120C/248F
PVA	5	40-110	78	3	7	75C/167F

Websites such as **www.simplify3d.com/support/materials-guide/properties-table** and **all3dp.com/1/3d-printer-filament-types-3d-printing-3d-filament** also provide a lot of information about the properties of different plastics for 3D printing.

RESISTANCE TO ULTRAVIOLET LIGHT

Of particular concern to hams is the plastic's resistance to the ultraviolet light (UV) in sunlight. Many ham 3D printed objects will be used outdoors and experience long-term exposure to sunlight. Without exception, all sources agree that the best plastic for UV resistance is ASA (Acrylonitrile Styrene Acrylate). Acrylics are virtually impervious to UV. ASA is somewhat more difficult to print with and is more expensive, but for ham projects involving long-term exposure to sunlight, it is the best choice.

Table 23.8 gives a general picture of the UV properties of the common 3D printing compiled by Robert Zavrel, W7SX. Another abbreviated summary for other common 3D plastics is available at **3dprinterly.com/best-filament-for-outdoor-use-is-pla-suitable**. PLA, ABS and PETG are discussed in more detail in the online article.

RF POWER HANDLING CAPACITY OF PLASTICS

Another important factor in choosing a 3D printing plastic for ham projects is how well a plastic will work in a high RF environment. The property is called *dissipation factor* (*DF*) or *loss tangent* (*tan* δ). It is a measure of how much the materials dissipate RF as heat. The higher the DF, the lower an RF field strength the plastic will tolerate before melting. Materials with a high DF are referred to as "lossy." It is important to consider both the physical and the electrical properties in selecting a 3D printing filament for a ham radio project.

Table 23.9 is organized from lowest to highest dissipation factor. One or two plastics not commonly used for 3D printing are included to illustrate the general use of dissipation factor in selecting plastics for electronic applications.

23.10.6 The Build Plate

BUILD PLATE SURFACES

Most printers have a base aluminum build plate. Printing is not done directly on the base build plate. An additional surface is added to the base. Common surface materials include borosilicate glass plates, polyetherimide (PEI) sheets, and Kapton tape which are available online. Blue painter's tape, a common hardware store item, is also used.

A removable flexible steel sheet is another common added print surface. A self-adhesive PEI plastic sheet is then typically applied to the steel. A removable flexible sheet makes the removal of printed object easy: a small flex and the printed parts pop off. Tightly adhered objects can be difficult to remove from a fixed build plate, increasing the risk of damage to the build plate.

BUILD PLATE ADHESIVES

In most cases it will also be necessary to apply an adhesive to the build plate so that the first layer of plastic sticks firmly. Problems in the first layer are the cause of a high percentage of failed prints. Poor adhesion can also allow objects to warp, especially with some plastics, such as ABS. A table of common adhesives and the plastics they are used with is included in the online article.

The two most-used adhesives are glue stick (PVA) and hair spray (PVP). Both are water soluble. Before applying any adhesive, clean the surface with isopropyl alcohol (most common) and/or hot water. Occasionally also clean the surface with acetone. Due to ABS plastic's tendency to shrink as it cools, a special adhesive is highly recommended, variously called ABS slurry or ABS juice. (See

Table 23.8
UV Resistance of 3D Printing Plastics

Material	*UV Resistance*
ABS	Fair
Acrylic	Good
Polycarbonate	Fair
Polyethylene	Poor* – Fair
Polypropylene	Poor* – Fair
Polystyrene	Poor – Fair
PTFE	Very Good
PVC (plasticized)	Fair

*Basic UV resistance can be improved with additives.
From "Using Plastics for Dielectrics," Robert Zavrel, W7SX, *QEX*, January/February 2021

Table 23.9
Dissipation Factor of Common Plastics

Plastic	*Dissipation Factor (tan δ)*	
	1 kHz	*1 MHz*
PTFE (polytetrafluoroethylene) Teflon	<0.0001	<0.0001
LDPE (low density polyethylene)	0.0003	0.0003
PP (polypropylene)	0.0003	0.0003
PLA (polylactic acid)	0.0003	0.0004
HDPE (high density polyethylene)	0.0005	0.0005
PS (polystyrene) Styron	0.0005	0.0005
PVC (polyvinylchloride)	0.013	0.006
PC (polycarbonate) Lexan	0.0015	0.01
PI (polyimide)	0.0025	0.01
PET (polyethyleneterephthalate) Mylar	0.005	0.016
PMMA (polymethylmethacrylate) Plexiglas, Acrylic [ASA]	0.04	0.03
Nylon6 (polycaprolactam)	0.016	0.036

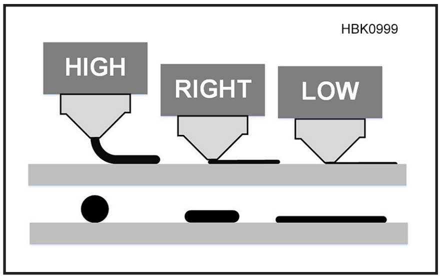


Figure 23.43 — "Squishing" of the extruding plastic bead: (left) No squish; (center) correct amount of squish; (right) too much squish (not-to-scale).

the full article for more on this.)

EXTRUDER HEIGHT ADJUSTMENT

The instruction manual for every printer includes the steps to adjust the working height of the extruder during a print. However, before it can be set, the mechanical geometry of the printer must have been carefully aligned. This is called the *X, Y, Z calibration*. Follow the manufacturer's calibration instructions meticulously. This cannot be overemphasized.

The next adjustment that must be correct before extruder height can be set is *bed leveling*. It is only a manual adjustment on lower-cost printers; it is automatic on others. Many experienced 3D hobbyists consider automatic bed leveling essential, and it is highly recommended.

To visualize why extruder height adjustment is critical, see **Figure 23.43**. To cause the soft plastic to positively adhere to the build plate and to subsequently fuse to a previously extruded layer of plastic, the tip of the extruder's nozzle must flatten the bead, called *squish*. The gap between nozzle tip and the bed or object must be roughly half the height of the diameter of the extruding plastic bead as illustrated in the figure. Recognizing a properly squished bead is an essential 3D printing skill and requires close attention. Become very familiar with first layer calibration of extruder height.

23.10.7 A Typical Print Sequence

The following basic steps are followed for every print. This is a reference for the newcomer and also highlights some tips for all experienced 3D printers.

1) Load the object's G-code file.
2) Let the printer go through its start-up sequence.
3) Clean the build plate surface.
4) Check for any particles or scraps under the added surface, if any.
5) Clean the extruder tip.
6) Select the G-code file to print.
7) Check the *purge strip*.
8) Check the skirt or brim, if any.
9) Confirm the printer proceeds to the second layer.
10) Remove the printed part.
11) Clean up the printed part.

23.10.8 3D Printing References

"3D Printing," Wikipedia, **en.wikipedia.org/wiki/3D_printing**.

"Introduction to 3D Printing," Ham Radio Workbench (podcast, list of additional online resources) **www.hamradioworkbench.com/podcast/introduction-to-3d-printing**, 2016.

"Introduction to 3D Printing," Instructables (**www.instructables.com/3D-Printing-1**), 2012.

Pingree, J. WB2TVB, "Get Started with 3D Printing," *QEX*, Jul/Aug 2019, pp. 16–19.

Wade, P., W1GHZ, "3D-Printed Antennas," Microwavelengths, *QST*, Jan 2019, pp. 67–68.