Prototyping Techniques

v 1.2, 2/19/03

D. Grover

Once an electronic circuit has been designed on paper (or simulated in software), the next obvious step is to connect components together and see if the physical circuit works as one wishes. In some cases, one might explore changes to component values, or make structural changes to the circuit based on what one actually sees. It's also the case that often one wants to use the circuit (or demonstrate it) over a period of time. This document discusses different methods for creating physical realizations of electronic circuits for various needs.

A few of the uses for prototype circuits are:

- Demonstrate/verify function of a circuit design
- Fine-tune design
- Create one-off or limited number of prototypes

Some of the characteristics and limitations of prototypes include:

- Correct function: The prototype should function as well as the production device needs to, including with respect to noise, temperature, etc. Not all methods of prototyping produce results that are comparable to a well-designed printed circuit board, particularly in the area of high-speed and low-noise.
- Size (similar to production version): Some prototyping methods are not well-suited to compact prototyping.
- Robustness: The prototype should stand up to normal handling and use; connections should be solid and not affected by movement.
- Neatness: A neat prototype allows quicker diagnosis of problems, and more clearly illustrates the design.
- Debugging accommodations: Provision should be made for the attachment of test probes (oscilloscope probes, VOM probes, logic analyzers, in-circuit debugging, etc.), including connectors to allow inserting equipment to measure current and to inject signals in place of components. It is very useful to solder 1/2” diameter loops of bare solid wire to allow easy connection of ground clips. Logically separate modules should be connected using connectors to allow isolation, replacement, and easier debugging.
- Cost: Appropriate to the purpose and budget.
- Labor: The amount of labor varies dramatically between prototyping methods; in some cases this labor can be applied to producing multiple devices, or even to producing final production designs.
- Multiple uses, reusable: If a great deal of effort is expended in creating a prototype, in some cases it will be advantageous to create it in modules, pieces of which might be usable for other projects, or to add additional function or flexibility to allow reuse for another project or variation of the project.
- Replicable: In many cases it will be very useful to be able to easily create a duplicate of a prototype, allowing multiple people or groups to work on the same design at the same time, providing a backup unit, or to allow quicker debugging.
- Documentable: Some methods automatically produce documentation of the design during the creation of the prototype, and others require additional (but necessary) work.

- Stable over time: Ideally the prototyping method will produce a device that is relatively stable over time. Oxidation of contacts, for example, is a process that could lead to less reliability over time for some methods.

What now follows is a brief introduction to several prototyping methods. While not exhaustive, these are the most common methods for relatively simple circuits that might be encountered by a typical student or hobbyist.

1. Plastic breadboard

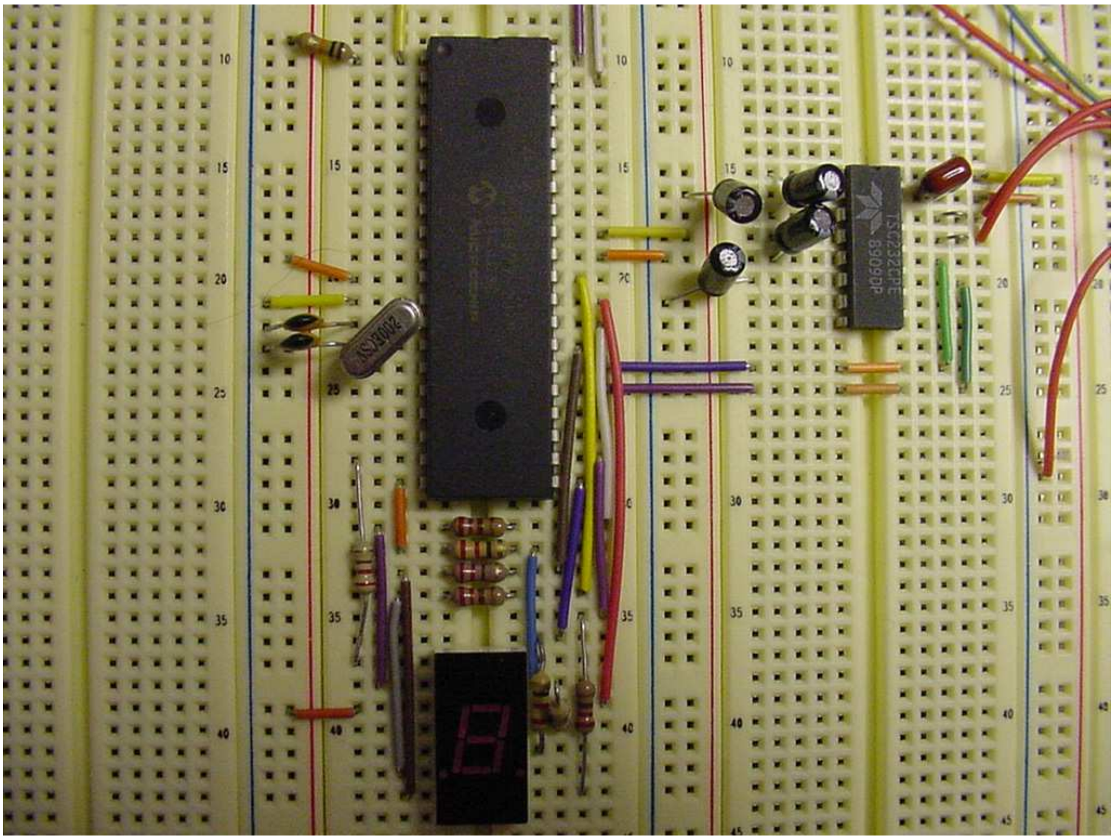

Figure 1: Plastic breadboard

Description: White plastic boards with parallel rows of connectors under a matrix of holes centered on 0.1”. Components press into the board, and wires are plugged into adjacent holes to connect components.

General comments: Plastic breadboards allow quick construction of low-complexity, non-critical circuits. The boards can be reused, which leads to low cost over time, but use also leads to degradation of the contacts. Capacitance between rows limits usefulness for high-frequency work (between 2-5 pF between adjacent strips)[1], while the type of plastic can vary in properties such as resistance, which argue against use in prototyping sensitive designs. The distributed capacitance can also mask design problems that can occur when some designs are moved to a printed circuit board which does not have the same capacitance at each pin. Because nothing is soldered in, components and interconnect can come out easily, with no clear indication where they came from. The planar nature of neat wiring limits complex designs (especially

1   Bob Pease, Troubleshooting Analog Circuits, 1991, page 153.

buses), and the alternative (wires connected free-form in the space above the board) is often messy, hard to understand, difficult to work around without dislodging connections, and less robust to handle. Components on other than 0.1” spacing are difficult to accommodate.

2. Point-to-point wiring

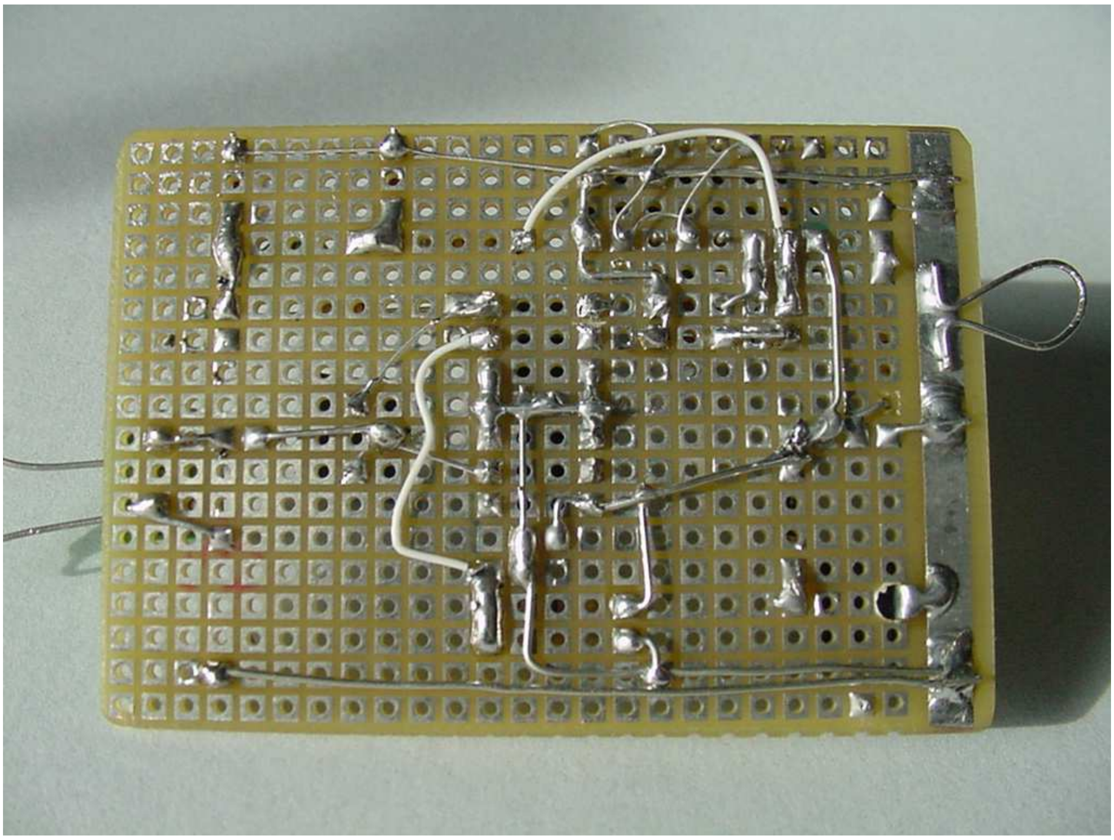

Figure 2: Point-to-point soldered prototype (see Figure 4 for PCB version)

Description: Thin phenolic or fiberglass material with a matrix of holes on 0.1” spacing, typically with tin-plated copper pads around each hole, is used to mount components. Wire is soldered to the bottom of components to connect components together.

General Comments: Can be robust (all connections are soldered). Wire should be as fine as possible consistent with current-carrying requirements, and should be solid. Wire-wrap type wire is a good choice. Typical wire used with plastic breadboards is usually too unwieldy, and has insulation that melts too easily. Good planning of component placement can allow solder bridges (normally to be avoided) to be used for many connections. Bare wire is often acceptable for grounds and connections where shorts will not occur.

3. Dead-bug

Description: A variation of point-to-point soldering is used when there is a need to prototype very sensitive circuits, ones in which capacitance between wires needs to be at an absolute minimum, and a good, low-impedance ground plan is a must. Called "dead bug" prototyping because of the way ICs are mounted on their backs with the "legs" in the air, wires are connected from pin to pin through air, and so reducing capacitance. The ICs can be glued to a copper clad board (e.g., an un-etched circuit board) allowing ground connections to be short, reducing inductance. However, such circuits are rather messy to work with and are best reserved for the small sections of very sensitive analog circuits, rather than general circuits. (See Pease for details.)

4. Wire-wrapping

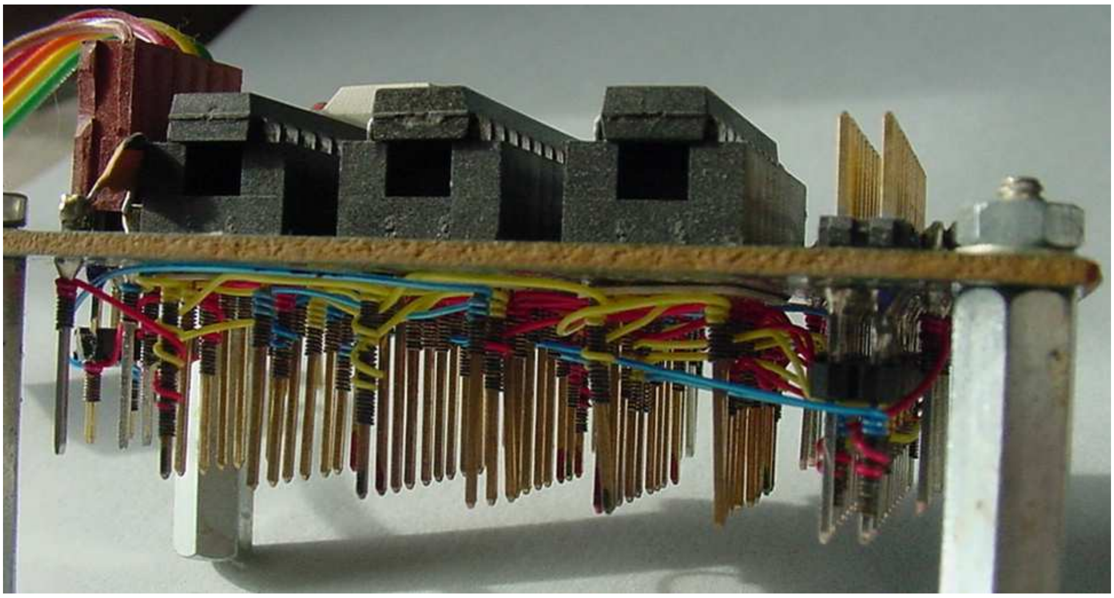

Figure 3: Wire-wrap prototype. Note 0.1" spaced 0.025" square post headers at right.

Description: In wire-wrapping, all components are socketed or otherwise attached to a circuit board with a matrix of holes on 0.1" centers. Beneath each component pin is a long square pin between 0.5" and 1" long and 0.025" square. Connections between pins are made by wrapping the stripped ends of very tiny diameter solid wire around the pins. The sharp corners on the pins dig into the wire many times as the wire is tightly wrapped using a special tool, and form an air-tight connection.

General Comments: Pre-stripped wire and motorized wrapping guns make connections relatively quick, but the special wire-wrap sockets are a bit expensive. One can also use inexpensive manual tools and cut and strip wire to length manually. Wire-wrapped boards are relatively robust, though there is a substantial thickness penalty (up to an inch), and the pins can short if bent over. Unwrapping wires is possible (sometimes using the same tool as for wrapping, other times a special tools is used), but because multiple wires are often wrapped on the same pin, changing one wire sometimes leads to having to unwrap and remove multiple wires. (It is generally not advisable to unwrap a wire and rewrap it, due to the deep nicks and resulting fragility of the unwrapped wire. In a pinch, it can be done.)

5. Printed circuit boards

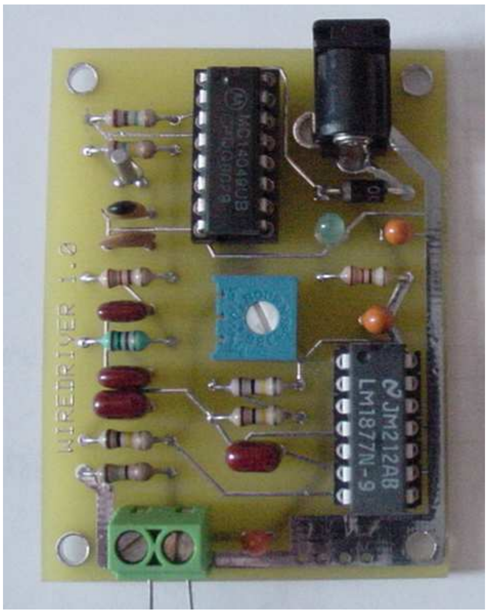

Figure 4: PCB (double-sided). (See Figure 2 for point-to-point soldered version.)

Description: Printed circuit boards (PCBs) are used for production-level quantities of electronic circuits. While there are variations, the most common type is a sandwich of copper layers on either side of a non-conductive material such as fiberglass-reinforced epoxy or a phenolic material. A photomechanical process is typically used to create the boards, usually involving having an acid etch copper from selected areas in order to leave traces connecting pins. This is the technology of the familiar circuit boards found in PCs and most all other (consumer) electronic devices.

General Comments: For most student projects, a two-layer (or double-sided) circuit board--that is, with traces on both sides of the board--is sufficient. Single-sided boards can sometimes be easier to make using certain do-it-yourself methods, but are more difficult to design since with just a single layer for traces, wire-jumpers must be used when traces need to cross-over each other.

An important element of double-sided board (or any boards with 2 or more layers with traces on them) is the use of “plate-through” holes. With traces on both sides of the board, it is necessary that component pins soldered into the board make electrical contact with traces that connect to the hole on both sides of the board. This is a problem on the top side of the board since components are typically soldered only on the bottom of the board. Indeed, it might be difficult or impossible to solder the pins on the top side (or component, vs. solder, side of the board). For this reason almost all modern manufactured 2-layer (and up) PCBs have all component holes plated, thus making a good electrical connection between top and bottom layers. Soldering a component pin on the bottom of the board therefore also means you can count on a good electrical connection to the top side. As an added benefit, plated-through holes are also physically more robust against overheating--the pads on single-sided boards can easily be peeled off with too much physical force or excessive heat (e.g., from repeated or prolonged soldering).

PCBs generally have copper traces, but may be plated with tin, tin/lead, or silver alloys to prevent oxidation and make it easier to solder. A solder mask may also be applied, which is the (often green) coating that covers both sides of the circuit except where a connection will be soldered. White silk-screened legends are also common for production circuit boards, and include component labels, outlines, and values, along with other information about the circuit board (such as company, version, date, etc.). The setup charge to create the silkscreen is generally prohibitive for small quantities of boards.

Design/manufacture: PCBs can be made with oil-based markers (e.g., Sharpie) and an etchant (available at Radio Shack, for instance), but such “home-made” PCBs will lack plate-through holes, and require tiny holes to be drilled in the rather abrasive epoxy board. (Carbide drills are preferred, since regular “high-speed steel” drills will dull quickly. However, carbide drills are even easier to break and so must be used in a drill press or similar.) Engineers typically design a circuit board using specialized software, and send the resulting, standardized set of files (“Gerber” files to describe the circuit board traces, and a drill file to specify hole sizes and locations) to a circuit board house for manufacture. There are a number of quick-turn board houses that can make a double-sided circuit board of small to medium size in a day, and ship out the resulting boards overnight for a total turn-around time of a few days, for between $60 and $100 for 2 or 3 boards.

An alternative method to manufacture very small numbers of boards is to use a dedicated CNC milling machine. Using the same Gerber files, a small but accurate milling machine can remove copper just around the traces. The resulting board is then drilled, and ideally the copper is then plated, but rarely are the holes through-plated. The process, while automated, takes a while, since each trace must be isolated from the surrounding copper one trace at a time. Thus, a circuit board might take hours to mill.

The downside to PCBs, aside from cost, is the need to acquire and master the software necessary to generate the PCB files. Most PCB layout software is rather expensive (sometimes >$10,000/seat), and almost all requires a great deal of work before a reasonable design can be created. However, recently a few quick-turn board houses have taken the step of making available for free simplified PCB layout software. (For example, see http://www.expresspcb.com/.) By limiting many aspects of the program, the user can quickly master the software for many simple designs. In addition, the program is constrained to generate designs that match the manufacturing capability of the board house (eliminating a costly and time consuming step of iteration if the design exceeds the ability of the board house). Further, the software generates an output in a proprietary format, which while it does restrict the manufacturing options, does so with minimal investment on the part of the user.
One especially important upside to PCB layout software is the fact that often one can use schematic capture software to generate a schematic whose connectivity information can be imported into the PCB layout program. This is very useful in that it can otherwise be very difficult to verify that all the connections have been properly made.

Hints for PCB design

- Lay out ground connections first, using thicker traces (to reduce resistance)--at least 0.020” wide (above 0.050” and it gets harder to route), and even wider for higher current.
- Next, lay out power traces, also using relatively thicker traces.
- Unless there is a compelling reason, use traces at least 0.015” wide for most signal connections—even if the board house can do smaller. Wider traces are easier to rework (e.g., inserting a surface mount resistor inline, cutting or jumpering traces, etc.).

- Bypass capacitors should be located physically near their respective ICs and have short connections to both power connections on the IC.
- See comments elsewhere on grounding.
- Be sure to measure the diameter of components such as diodes which might have a larger wire diameter. The hole of any pads should be at least a little larger than the largest diameter necessary (the board manufacturer will have information on the tolerance they have on plated-through holes). Remember to measure square pins diagonally when calculating hole size.
- The pad for holes must be larger than the hole itself. The minimum annular ring is often specified by the board manufacturer, but should be at least 0.020" larger in diameter than the finished hole diameter.
- Group components based on their connection with other components, though the board will look neater if you orient parts the same direction.
- Components with a definite orientation, such as LEDs, diodes, polarized capacitors, transistors, ICs, etc., should be relatively consistent with others of the same type whenever possible. Assembly and debug is much easier when all the diodes, say, point the same direction.
- Pin 1 of ICs should be in the same direction whenever possible.
- Mark pin 1 of connectors using text on the copper layer and/or on the silkscreen layer.
- Include mounting holes (#6 screw: 0.140". #4 screw: 0.113" diameter hole) for screws, feet, etc.
- Include text on the top copper layer (or silkscreen layer) with description, version number, date, designer, etc.
- Label on top copper (or silkscreen layer, if used) all connectors (e.g., "J1", "J2", etc.).
- See http://www.expresspcb.com/ExpressPCBHtm/Tips.htm and http://www.murrietta.com/mc-ls.htm , http://www.mitsi.com/PCB/Hints/tips.htm,

Construction Hints for PCBs

- Non-polarized components, such as resistors, should all nonetheless have the same orientation, so that color codes are easily read.
- Capacitors and other components (such as diodes, crystals, inductors, etc.) should be mounted (when there is an option) so that any markings can be read after assembly.
- Use IC sockets for all ICs unless there is a compelling reason not to.
- "Machined pin" IC sockets are expensive, but much nicer to work with and worth considering for prototyping.
- Use the pins from machined-pin IC sockets or equivalent to create custom sockets for components so you don't have to unsolder & solder in to change a component value (for example, in a feedback resistor in a amplifier).
- Soldering components in order from shortest to highest makes it easier to hold components flat against the circuit board during soldering:
    - Surface mount devices (R's & C's first, then ICs)
    - Resistors (1/4W), diodes, small axial capacitors & inductors
    - IC sockets
    - Pots (i.e., single-turn 3/8" square)
    - Small capacitors (ceramic, epoxy-dipped plastics, tantalum)
    - LEDs
    - 0.025" square post headers
    - Larger items in ascending order of height

- For multiple boards or boards with many components, a special assembly frame makes assembly much more efficient.
- After all soldering is done, but before ICs are inserted, remove all soldering flux. Most all electronic solder in wire form contains flux; organic fluxes can be washed off in warm water (do so as soon as possible—within hours after soldering, not days). Other fluxes require solvents. Flex left on a circuit board can alter the behavior of the circuit, and will over time look bad. (If using water, dry the board as soon as possible. Most all electronic components are fine with water and many solvents.)

6. Evaluation boards

Description: Evaluation boards allow quick demonstration and prototyping involving semiconductor devices such as microcontrollers and other complex devices. Often these are offered by the manufacturer, often heavily subsidized, and can be inexpensive as they may also include supporting software. When appropriate, these can be used as the basis for designs, building from a known-good core and well-characterized environment. Often there are unpopulated holes for adding simple circuits, and almost always extensive connectors for adding ribbon cables to external prototyping boards (e.g., wire-wrap, custom PCBs, etc.).

The following are some general comments about prototyping components.

Wire
Wire and cable have a variety of characteristics, including:
--wire gauge: the effective diameter of the wire sets the maximum safe current it can carry, as well as the resistance per unit length. Tables are widely available showing the safe current limits for various gauge wire.
--solid or stranded: solid wire is not appropriate for situations where wire can flex, since it can fatigue and break. Most interconnect between boards and or sensors should be stranded. Jumpers entirely on one board should be solid wire. Solid wire must be used on plastic protoboards.
--multi-conductor: Ribbon cable offers multiple conductors parallel to each other, and can be easily and reliably terminated using insulation displacement connectors (IDC). These are the familiar connectors used on the insides of most PCs, for example. Conductors may be shielded, twisted, parallel & flat, or just parallel within a plastic jacket. Shielding and twisted pairs can reduce external interference.

Connectors
Direct soldered connections are appropriate when there is minimal flexing of the wire expected (e.g., the wire goes to a location on the same PCB) or appropriate strain relief can be provided, and where there is no need to disconnect the two components. In many cases, however, it is advantageous to use connectors. In general, avoid connectors easily mistaken for other uses (e.g., never use AC power-style connectors for anything other than AC; use caution when using telephone-style connectors since in some cases people could connect to an actual phone line with 80+ volts on it). Connectors should be able to carry the maximum current necessary, and be polarized when appropriate. A few common connectors and uses are:

- Insulation-displacement connectors: Used to terminate ribbon cable, good for 10-50 connectors. Not shielded. Can parallel conductors for more current. Uses special tool (or great caution and vise) to install. Mates with common dual-row 0.1” spaced 0.025” square post pins.
- 0.1” Center connectors: Mates with single-row version of 0.1” spaced 0.025” square post pins. Good for 2-10 conductors, up to a couple of amps. Pins are crimped (and you might also want to solder them, carefully, for added robustness) and then inserted into a housing. Example: Molex--GC/Waldom 0.100” High Pressure Terminal Housings (DigiKey part number WM2604-ND) and crimp terminals (DigiKey part number WM2612-ND). Mates with 0.100” center headers such as DigiKey part number WM4003-ND.
- For currents about a couple of amps, Molex makes power connectors such as WM1324-ND, good for 12A.
- D-Sub connectors: The familiar 9-pin “DB9” used for serial connections on the PC comes in 9, 15, 25, 37, and 50 pin versions. Contacts are generally specified at no more than a couple of amps. Cables are widely available, including shielded. Not very conducive to prototyping methods requiring components on 0.1” centers.
- 1/8” (aka 3.5mm) and 2.5mm stereo jack: Providing 3 conductors, these also come in shielded versions and are good for signals (e.g. microphones, headphones, RS-232) and light power (See Cui and Switchcraft's offers in DigiKey under audio, power, and phone jacks).
- 2.1mm and 2.mm coax power jacks: Most DC wall transformers are terminated in the familiar coax power plug. The DigiKey part number CP-002A-ND is a popular style.
- RF connectors: For just one or two RF signals, BNC style connectors are available as are cables in several impedances. In cases where size is an issue, SMA type connectors are much smaller, though quite expensive. It is probably worth looking at the availability of cables or antennas to determine what style makes the most sense. There are some DB-style connectors that mix shielded and non-shielded connectors on the same connector, though these are also expensive.
- Terminal blocks: Screw-clamp terminal blocks generally make sense only when it is not possible to prepare the end of any attaching conductors past stripping insulation. Chose spacing conducive to your prototyping method (e.g., 0.1” or 0.2” spacing for wire-wrap).

Standoffs & mounting hardware

Plastic (typically nylon) or aluminum standoffs are necessary for wire-wrap boards, and useful for point-to-point soldered boards. Non-skid rubber bumpers are nice to have on PCBs since they'll keep the board from moving as easily on work surfaces. Beware metal hardware—make sure the screws or standoffs will not short out traces near the mounting holes.

Heatsinks

Provision should be made to mount heat sinks on any components necessary. Datasheets will typically provide thermal design parameters for power semiconductors, and sources such as Horowitz and Hill outline design procedures for sizing and applying heatsinks.

Cases
Cases improve robustness by protecting against physical damage, but can also be important in shielding from electrical noise. Install connectors such that removal of the circuit board is straightforward--it is a perverse fact that the harder it is to remove a component, the more likely it is that one will wish to remove it. Plastic cases (such as PacTec or Serpac) are easily machined--use sharp drill bits and mills at lower speeds--but offer no shielding. Plastic cases are often offered with removable front and rear panels (easy to mount connectors), and can be had with compartments for both 9V and 2 AA batteries. Do not overlook used equipment as a source of small, well designed enclosures. Used low-speed modems, for example, can often be had for cheap that provide a small enclosure well-suited to many projects. Cases must allow for the dissipation of any heat generated, whether by circulation of air, or conducting heat to the case.

Power supplies
Wall transformers can be had in sizes up to 10-20W or more. Check for AC vs. DC output, regulated or not, multiple voltage output (common among surplus units, multiple outputs can be handy if your circuit can use them), and connector type. Regulation may not always meet your requirements, so on-board voltage regulation is always a good idea.

Fusing
It is prudent to always make sure there is a overcurrent protection of some type. External power supplies as well as internal batteries (especially NiCads) can supply large amounts of current in case of a short or component failure. “Polyswitch” (a trademarked name) resettable overcurrent protectors are one option to consider--they are cheaper than circuit breakers, and automatically reset.

Debugging connections
Prototypes often benefit from having multiple and easily accessible points for connecting ground clips. On large boards, place these close to likely test points. See Pease and also Johnson for more ideas.

Power, reset
Include a power switch when possible. A diode (e.g., 1N4001) can be included in series with incoming DC power to prevent damage in case the wrong power supply (AC or DC with an opposite polarity) is plugged in. Include a reset button for microprocessor-based devices—this is much better than cycling power to reset a circuit.

Grounding
The large noise margins of digital electronics can lure one into not paying much attention to the analog reality of all electrical signals, but pitfalls await if attention is not given to proper grounding and power supply issues. Any current passing through a ground trace will produce a voltage (since the trace has some resistance)--this is just ohm's law. Digital circuits can produce spikes of current that translate into noise on

traces to ground, so analog circuits should have their own connection to a ground "mecca" (often either where external power comes onto the board, or perhaps near the voltage regulator). Copper fill can be used to create a low-impedance ground path for sensitive sections of circuits, though it constrains routing on that layer of copper. Application notes from Analog Devices (www.analog.com) on mixed signal processing addresses some of these issues, and Johnson (see references) goes into good detail on power supply bypassing and ground requirements.

Leave space

Leave space to add additional filtering, more memory, additional bypass capacitors, etc. The prototype is no place to skimp on board size if its primary use is in debugging/development. Even if the final product will use a stripped-down version of a processor, use a faster part with plenty of memory for prototyping.

Neatness

Any time spent on making the prototype as clean and neat as possible will generally be paid back by reducing the debugging time. Make solder joints look nice, so that it is easy to scan them for bad joints. Use cable ties to organize wires and cables. Avoid using electrical tape--it does not work well for mechanical tasks and leaves an unpleasant adhesive residue. Heat shrink is the preferred method for providing insulation (and some amount of strain relief) around wire connections.

Working schematic

Strictly speaking, a working schematic is part of the design process rather than prototyping, but since it seems to be overlooked all too often, here is another reminder. A working schematic is the up-to-date record of the circuit, capturing clearly and completely the circuit. It should be easy to explain, document, or reproduce the circuit using just the schematic. Debugging, especially by someone not involved in the construction of the circuit, is much more efficient with this document. The working schematic is just that—a work in progress. It is not necessary that it be drawn in a CAD program—handdrawn on engineering paper is fine. The schematic collects notes during construction and debugging anyways. An excellent excerpt from Horowitz and Hill on drawing schematics is on line at: http://xcircuit.ece.jhu.edu/goodschem/goodschem.html

A few, selected references

- Horowitz and Hill, The Art of Electronics, Second Edition, Cambridge University Press, 1989.
- Bob Pease, Troubleshooting Analog Circuits, Butterworth-Heinemann, 1991.
- Howard Johnson & Martin Graham, High-Speed Digital Design: A Handbook of Black Magic, Prentice Hall, 1993
- ExpressPCB.com