

AutoCAD for Electronics Part 3

Tips and techniques for high-speed schematic drawing.

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Last month we looked at custom menus and their macros, perhaps the easiest way to make AutoCAD fast and convenient. As we mentioned, AutoCAD straight from the box tries to be everything to everybody; it's probably the most flexible program around, with every possible parameter under your control. However, this also means that you'll be typing your brains out selecting default after default, and often you'll probably find yourself lost down a long winding path because the operation of some commands depends on how you select points, whether you use a window, etc.

A quick recap: many of AutoCAD's variables are determined by a master drawing called ACAD.DWG. Load this as an existing drawing, change the variables to suit yourself and then save it. Use the Reference

Manual to check out ACAD.MNU, the menu that comes with it, and see how it's laid out; write a screen menu (the one that appears in the right-hand sidebar) to call up your most-often used commands the way you like them. Investigate the pulldown menus (Release 9 or 10 only) and you can create hundreds of commands just the way you like them. If you have a digitizing tablet, write your own tablet menu. Using the three types of menus, you can almost retire the keyboard, and AutoCAD will zip along as fast as you can go (usually).

The June issue contains detailed explanations of how to write menus, with the single most useful function being the macro. Each command title on your menu can actually call up a sequence of further commands, letting you do a complex bit of draw-

ing with no more than a few mouse clicks.

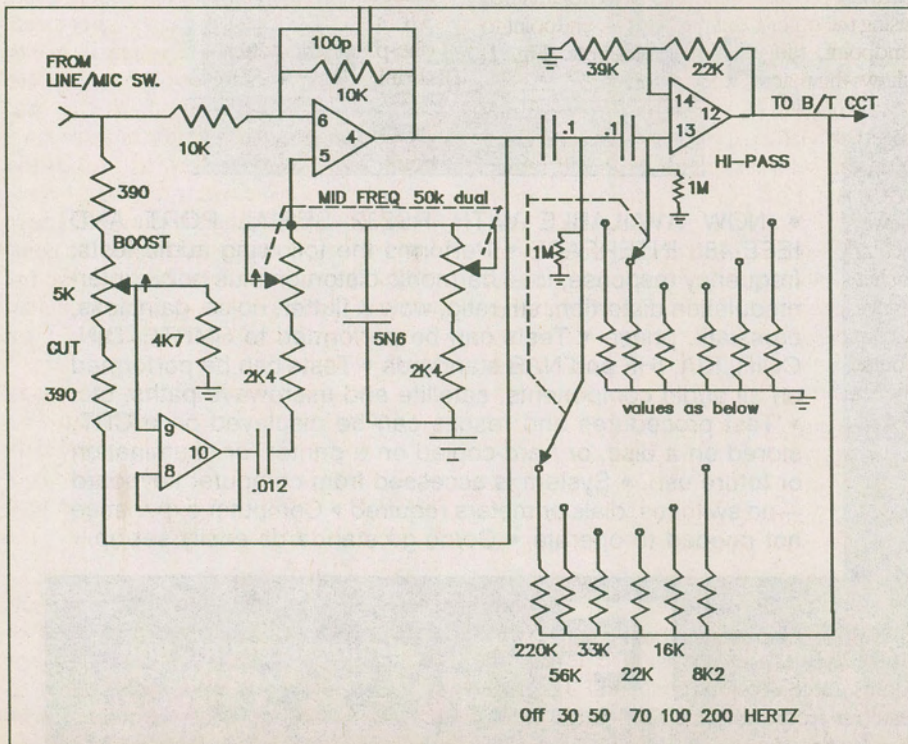
The PGP File

I assume that you've tinkered with your custom menus and you're ready to go. But first, I also assume that you've ended up a bit frustrated because editing your menu means exiting AutoCAD, loading your word processor, reloading AutoCAD to try it out and so on. You can speed this up considerably with AutoCAD's ACAD.PGP file, a utility that lets you call up other programs right in the middle of AutoCAD. When you exit your new program, you're back in the AutoCAD drawing editor. Much faster. Here's my PGP file:

```
DEL,DEL,27000,File to delete: ,0
DIR,DIR,27000,File specification: ,0
EDIT,WS,200000,,0
SH,,27000,*DOS Command: ,0
SHELL,,127000,*DOS Command: ,0
TYPE,TYPE,27000,File to list: ,0
```

You can get the syntax from the reference manual, but in general it works like this, using the EDIT line as an example: the first word (EDIT) is what you would type at the AutoCAD command prompt to start your word processor. The next word (WS) is the command that AutoCAD will pass to DOS (to start WordStar), and the assumption is that the word processor is in your ACAD directory or path. The number 200000 is the amount of RAM reserved for your processor. The two following commas mean that no prompt is issued (unlike DEL, for instance, which will print out "File to delete:"). The last number is a return code, with zero meaning that the screen stays in the text mode rather than returning to the graphics screen (a "4" returns to graphics).

Setting up your own PGP file means that you can scoot in and out of DOS, or your word processor, or whatever, all without the need to reload your AutoCAD drawing.



Starting Off

Let's start by drawing a schematic (*whew! At last!*) and examining the bottlenecks that crop up along the way.

One time-consuming operation is hooking one line onto the end of another, or attaching a block at an exact point. The monitor may show a perfect connection, but under some zoom conditions you can see that it's not really connected at all. Another is the problem of getting lines exactly horizontal or vertical; when printed out they often lean, though they look good on the monitor.

And of course, the frustration of trying to fix your mistakes — you often want to scrap it and start over because you're just making the muddle worse.

And while all this is going on, AutoCAD is taking *so-o-o* long to regenerate...

Step 1: Layers

Before starting to draw, you should have a minimum number of layers. AutoCAD is actually fairly fast at regeneration — if you're drawing lines. It's text, hatching and filling in solids that gives it that slow-motion look, and if these are on their own layers you can shut them off when they aren't needed (with Freeze Layer).

Call up ACAD.DWG and create Text and Hatch layers; while you're at it, type SOLID OFF so that solids will be drawn as wireframe rather than filled in. All new drawings will then have these parameters.

Step 2: Osnap

Osnap, or Object Snap, will eliminate all that jiggling and wiggling with the mouse, trying to get things to hook up. There are two ways to do it: one is to put AutoCAD into permanent object snap by typing OSNAP and then the modes you prefer, such as Endpoint, Midpoint, etc. This works well under some conditions and gets in the way in others. Another method, and the one that I like because it's easily called from menus, is one-time Osnap. This is invoked by typing or selecting Endpoint, Midpoint, etc., whenever AutoCAD requires a point.

Here's a macro I use all the time that will demonstrate what I mean. Most schematics have a lot of components hooked up to a supply or ground rail, and it can take a lot of time to accurately connect all those vertical lines.

```
[End,Perp] ^ CLINE;ENDP;\PERP;\;
```

The square brackets denote the com-

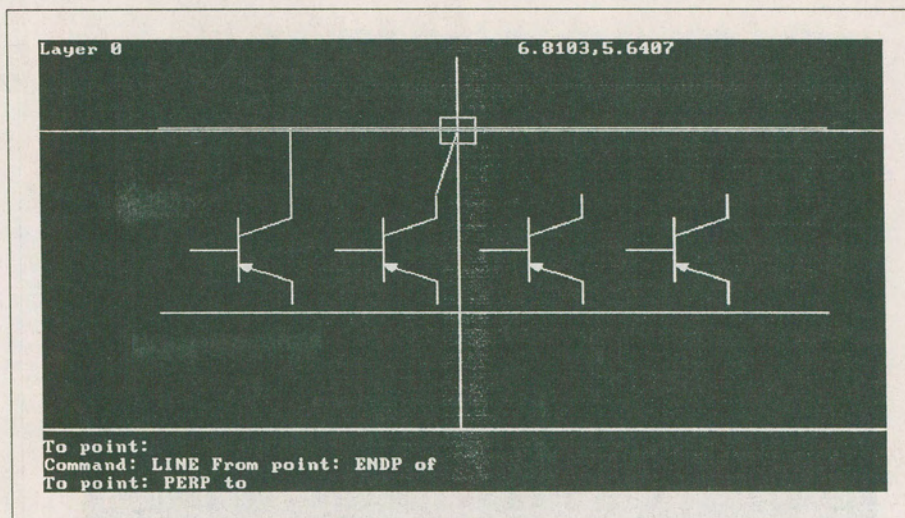


Fig. 1. Illustrating the ENDP,PERP macro described in the text. You only need to get the cursor near the transistor collector and the supply rail — AutoCAD will then draw a perfectly perpendicular line. No zooming or mouse-jiggling needed.

mand name. The Ctrl-C cancels any command in progress, the semicolons represent Returns, and the left slash waits for you to pick a point or object. To use this, say from a transistor collector to a supply rail, select the command and AutoCAD will ask "Endpoint of?". Select the transistor collector and AutoCAD will ask "Perpendicular to?". Click on the supply rail and there you are, a perfectly vertical, perfectly terminated line — no editing required, and you don't even have to hit the points exactly; Osnap captures within 10 pixels (the default Aperture). You can make lots of other macros using the other Osnap modes — endpoint to endpoint, tangents to circles, etc. Fig. 1. shows the macro in operation.

Another method of fitting things together neatly is to use the Snap and Grid. You can set the size and the on/off setting with ACAD.DWG. There are times, however, when Snap isn't that useful, particularly when loading blocks. You can attach a symbol for an op amp to a line, for instance, only to find out that none of the other terminals match anything. The cure, of course, is to design your symbol library with the Snap mode in mind; more on that later.

Zooms and Regen

As noted, Freeze any layers you don't need. You may have a tendency to use Zoom Window to zoom in and Zoom All to see the whole drawing. This will result in a Regen almost every time.

Instead, use Zoom Window to move in closer, and Zoom Previous to get back; AutoCAD stores up to five levels of zooms in the Previous command. This will result in Redraws, which are much faster than Regens.

Another good one is View Save and Restore. This lets you bounce around the drawing at Redraw speeds.

Blocks

One of the cardinal rules of CAD is *never draw anything twice*. To avoid doing this, most people have a library of drawing files containing components that they use all the time. There are two ways to make one of these library drawings. One is to start a new drawing and create the component in the size that you'll be using most; for instance, I made the transistor symbol about 3/8" square because I make fairly small, one-page schematics — if you make giants, shrink the symbol accordingly (note: use the SCALE command to do this, not ZOOM; zooming only affects your viewpoint, not the actual drawing size). If the size turns out to be wrong, the Insert command lets you change it during loading. Use the BASE command to pick the point on the drawing that later becomes the Insertion Point. Save your drawing (you may want to use a subdirectory to keep all your blocks organized).

The second way is to create a new component in the middle of another drawing. AutoCAD allows you to select the component and define it as a Block (which is stored only in the drawing itself — to write it to your block library, use WBLOCK).

Block Confusion

Blocks are an enormous timesaver, but sometimes they'll drive you nuts. They're like that.

First, when you work with them in a drawing, AutoCAD treats them as a single object and won't let you Trim or Erase block

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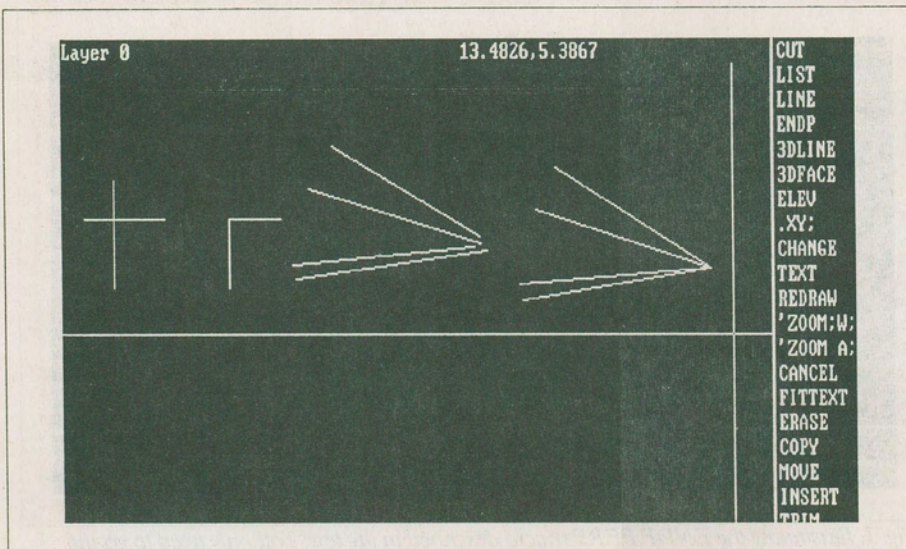


Fig. 2. The lines at the left have failed to intersect properly because they're too long. The fastest fix is to use CHAMFER or FILLET with zero settings — the lines will instantly be trimmed to a perfect corner. The converging lines, second right, fail to meet when you zoom in. Select CHANGE, type in a new point, and AutoCAD redraws each line until they meet exactly.

parts. You can return them to a group of separate objects with the EXPLODE command and then edit away. Or, during loading, you can precede the block's filename with an asterisk (*). This loads the block as a group of separate objects.

Second, blocks are with the drawing forever, even if you erase them. Getting rid of dead blocks is bizarre: you have to exit, reload the drawing and then type PURGE. You'll be asked if you want to purge various layers and blocks. If you type anything else before Purge, anything, AutoCAD says "Purge can't be used right now."

Third, blocks in your library can interact. Suppose you have a diode block, and you use it to make another block called LED. If you then load the diode into a drawing and redefine it, the LED block will reflect the same changes when it's loaded in. Try creating a new block rather than redefining.

Drawings of any complexity can be loaded as blocks and scaled to any size. For instance, I loaded all my electronic component blocks into one drawing called Symbols. I then printed it out and stuck it on the wall for reference.

Last month we looked at a pulldown menu for Release 9 or 10 that lets you call blocks up on the screen icon-style, eliminating any typing. You can get 16 blocks on the menu comfortably, and you can have as many icon menus as you like, making block insertion practically painless.

Block Troubleshooting

Wrong Size: call up the block drawing and use SCALE, assuming the change should

be permanent. For changes in a drawing (ones that don't affect the block file), use SCALE within the current drawing.

Wrong Insertion Point: load the block drawing and use BASE.

Facing Wrong Way: to turn your block mirror-image, add the following version of the Mirror command to your menu:

MIRROR;\Y;

Pick a vertical line around which the block should rotate, and that's it.

Remember that blocks are stored in your drawing so that drawing files can be transportable; otherwise you'd have to copy your block library onto every disk. Suppose you load a symbol for multiple insertions into your drawing, but find that it isn't quite suitable. Use Explode and various edit commands to change it, but don't use the Block command to redefine it unless you're really sure about this. If you generate different blocks with the same name (one in your drawing and one in the library), you're heading for future confusion for sure. Rename it, or use Multiple Copy for any further insertions within the drawing.

Editing

I think learning how to edit mistakes was the most painful thing to do in AutoCAD. It seemed to take ages before I could do it with any speed, but now I've accumulated a whole box full of tips and tricks (somebody once said that the difference between a veteran and a beginner is that the veteran can fix his mistakes faster). AutoCAD has a wealth of editing commands for fixing, adding and changing. Here's a rundown on

some commands, some problems, and some fixes.

Poor Connections: You've probably zoomed into a section of your drawing only to find lines that don't meet or symbols hanging in space instead of being attached. You can prevent this from happening by using the Osnap methods previously described, but for fixing, try the Move command with Osnap: MOVE,Endp,Endp will reconnect a line properly from one point to another or reconnect a symbol where it should be (you can also use all the Osnap features: Perp, Mid, Cent, and so on).

Tilts: lines that appear horizontal or vertical may have a tilt to them when magnified or printed. The Change command will let you replace one of the endpoints (or other point) to straighten things out. It will also work with a lot of lines at once, as shown in Fig. 2. Change has so many features, I'll give it its own section, next.

Change: If you select an object or objects and click the mouse, Change will replace points. If you press Return, a little menu appears, letting you change Color, Elevation, Layer, Linetype and Thickness. It beats erasing and replacing.

Change has special features for certain types of objects. If you select a circle and type a Return, you can change its radius. If you select text, yet another submenu appears, letting you change all the characteristics of the text. Sure beats retyping just to change the text height. See Section 5.3 in the Reference Manual for even more Change functions.

Wrong Lengths: Sometimes lines will be too long or too short because of editing changes or because they were drawn imprecisely. The TRIM and EXTEND will cut and lengthen objects, respectively. Note that you can't edit a Block unless you use the Explode command on it first. The BREAK command is like Trim, but works on a single line. It's a funky one, Break is, because its operation depends on where you pick points. For this reason, I prefer Trim if I can use it.

Here's a macro that lets you cut a line to any length. First, it lets you draw a line through the point where you want the existing line cut off. Pick the part you want trimmed, and the macro will do this and then erase the temporary line it drew.

[CUT]LINE;\Y;TRIM;L;ERASE;L;

Poor Intersections: Sometimes lines that seem to be making a perfect angle are actually crossing each other when you zoom in. Rather than using Trim or Break to fix this, try CHAMFER or FILLET with their values set to zero. They'll instantly snip off

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the offending overhang, leaving you with perfect corners. I use Fillet for rounding corners, so I leave Chamfer set to zero at all times for line editing (either one works — pick the one that suits you).

Stretch: if you ever tried Stretching anything, you may have been puzzled by its apparently random action. It's not really a problem, just ambiguous wording. The Stretch command tells you that you must use a window for object selection, but fails to mention that it must be a Crossing Window (which selects all objects that are in it or cross it anywhere). Further, if the objects are enclosed within the window, all you get is the equivalent of Move. Relocate your window so that the objects cross it, and they'll turn into rubber bands for you. Very handy if you've made a complex drawing too narrow — it's much faster than Breaking the objects, Moving, and then splicing in new lines.

Multiple Entries

You can avoid drawing things twice by using the Copy function's *Multiple* feature. This lets you select an object and then mouse around the screen leaving as many copies as you want anywhere you want. This is much faster than a series of Inserts.

There is also the Array function, which can multiply any object into a vertical, horizontal or circular lineup containing any number of identical objects. This one is certainly a powerful tool whenever repetition is need, such as a row of resistors or other components. It takes a while to get the hang of picking the distance between objects so that the array ends where you want it.

The MINSERT (multiple Insert) command seems less than useful; multiple Copies and Arrays are more flexible.

Text

The text handling of CAD programs falls behind their desktop publishing counterparts; labeling everything can take up half your drawing time. However, there are a few shortcuts. Here's a command for your menu that eliminates typing Returns; it accepts whatever defaults you've set with previous Text commands:

```
[FASTTXT] ^ CTEXT;;
```

It works very quickly, and if you want to change Text size, just use the regular Text command to reset defaults. Here's another one that will fit your text between any two points:

```
[FITTEXT] ^ CTEXT;F;;
```

The problem with this one is that the lettering distorts to suit the space available, and your text starts to look a bit messy if you overuse it.

Because of the limitations of most monitors, text will often look quite different when a test print or plot is made. The Change command will let you reset the text size to compensate; just select all the text and Change will step through each occurrence (and you can edit the text while you're at it).

The Text commands are, well, adequate. However, if your drawing is made up mostly of readymade blocks, as schematics would be, the **Attribute** function is where you can really pick up speed.

Attributes

This function lets you predetermine the size and location of the text attached to blocks. It even prompts you with your choice of messages. Just load the block and type in its labels — no mousing, no fiddling with sizes. However, it's not all that well explained in the manual; third-party books do a better job of explaining the rather intricate syntax. Here's a quick introduction to the basic workings.

Suppose you've drawn an NPN transistor for your block library (but haven't defined it as a block just yet). You'd like to have two labels on every transistor in the schematic, the Q number and the manufacturer's type number.

Type ATTDEF. You'll be asked for the following:

Tag is the general name of the attribute, used for reference. You could type *NPN* or similar.

Prompt is the message that will appear when you insert the block. Type *Q number* or similar.

Value is the default label that will appear if you don't type anything in. Type a Return unless you want to enter a default.

When you're finished, type a Return to restart ATTDEF, and add another attribute, this time typing in the same tag name and *Enter part number* or similar.

When you're done, define the transistor and attributes as a block. Try it out with the Insert command. It should draw your NPN and ask you for the part number and Q number.

Attributes are not separate text, but part of the block; you can't Move or Change them as you would other objects. Instead, use ATTEDIT to change the labels or positions (or DDATTE for a dialog box in Versions 9 and 10). ATTDISP will toggle the attributes on or off to speed regeneration.

Next month we'll investigate attributes and layers in more depth, plus more editing tricks, plus reviews of some excellent books on AutoCAD. ■