

AutoCAD for Electronics Part 5

More tips for rapid drawing and editing,
plus tricks for file management.

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Last month, we looked at some uses of the basic AutoCAD commands; this month, more on quick arrays. We'll also check out file management, not a major issue, but one that saves lots of headaches when you're trying to solve problems later on down the road.

Mental Blocks

Last month I was lamenting that AutoCAD wouldn't let you modify any blocks in your drawing unless you rename them. I should have known. It's been pointed out to me that I overlooked the equals sign, AutoCAD's handy symbol for swapping files.

Suppose you've drawn a block and put insertions of it all over your drawing, and then discovered that the thing is drawn wrong. Here's the catch: if it's a simple drawing that doesn't call in any other blocks, you can Explode it, edit it, and redefine it. However, if it calls other blocks (even if it's made up of multiple copies of itself), AutoCAD will say "Block references itself. Invalid".

Here's a way out. Let's say that you've loaded a resistor block Res, and grafted several of them into an attenuator Atten. You want to edit and redefine the block Atten.

First, use the Wblock command to write the original Atten to your disk. Save your drawing and load Atten.Dwg. Edit to your heart's content and save it.

Reload your original drawing and type Insert. When it asks for a filename, type **Atten=C:Atten**. Now the disk version will automatically redefine the drawing's Atten block and replace all occurrences of it. It seems like a lot of jiggy-pokery just to change a block, but it beats re-editing each occurrence.

Array of Hope

Previously, I mentioned that the Array command was wonderful for inserting rows or circles of objects, but that it was limited to straight lines, and that it was also difficult to space the objects just right. Here are two other types of arrays that pretty much fill the bill for multiple insertions.

Divide is a command that can mark a line into any number of equal parts if you give it a number, or it can insert a block instead of markers if you type a B and then a block name. By the by, the first time you try to divide a line into segments you may get no results. This is because the command uses the Point function, which is normally set to zero. Use **Setvar Pmode 2** for a cross, 3 for a rotated cross, etc. **Measure** works in exactly the same way, except that it marks off a line in some requested measurement rather than a number of equal segments. For instance, if you Measured a 25mm line into 8mm segments, you'd have 1mm left over at the end. If you Divided the same line into 8 segments, it would come out evenly, with each segment being 3.125mm long.

Suppose you wanted 10 LEDs and associated resistors to fit neatly along the bottom of an existing schematic. First, draw a ground line (or supply rail) the desired length under the existing part of the drawing. Next, draw or insert an LED and a resistor. Mark the two as a block.

Now use the Measure command to insert ten blocks along the line — but to do this, you'll need to ask for *eleven* segments. Note that there's a bit of overhang at the beginning and end; this can be removed with the Trim command.

If you know the length of the line and it's an easily divided number, the Divide command will work as well. Note that Divide and Measure work equally well with Lines, Arcs, Circles and Polylines.

Files and Directories

When you first started AutoCAD, you may have just loaded it onto your hard disk using a directory called \ACAD or similar. This works for a while, but when you consider that AutoCAD consists of more than 64 files (without sample drawings or drivers), it won't take long for your directory to become unmanageable. Soon one DWG filename will look like another (unless you care to number your files and keep an index).

One cure for this block madness is to store all your blocks in subdirectories; if you do nothing but electronic drawings, you can probably get by with one or two subdirectories for schematic symbols and so on. If you do complex architectural drawings, you might want a whole series of subdirectories.

However, when you go to Insert your drawing, you'll find that you'll have to type your brains out: "Block name:

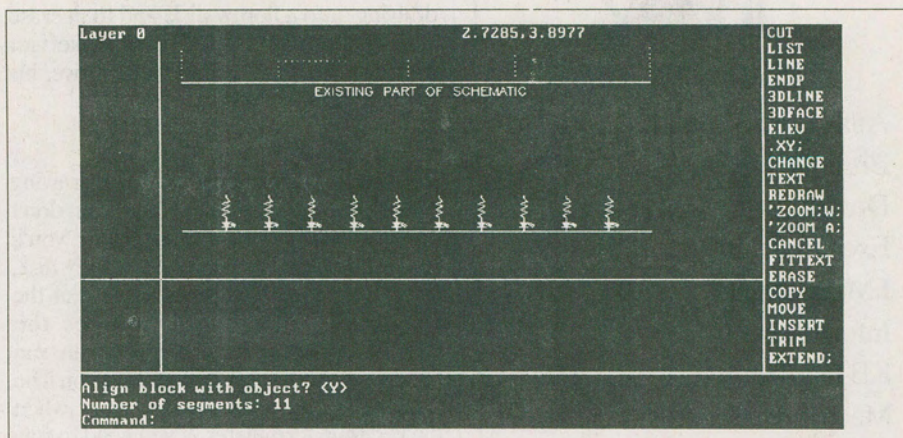


Fig. 1. The Divide command is used instead of Array to quickly insert a line of blocks into a drawing. Some trimming of the line is needed.

C:\ACAD\PLUMBING\FIXTURES\Toilet" might be a typical filename. To get around this, you can write your favorite block names into a pulldown icon menu as described in the second part of this series. Another way is to borrow a programmer's DOS trick and create some fake disk drives. Here's how it works. MS-DOS lets you define the number of disk drives from A to Z by inserting the line LASTDRIVE=n into your Config.Sys file, whether the actual mechanical drives actually exist or not (check your manual to see whether or not you have this feature if you're using DOS versions prior to 3.0).

Next, your DOS utilities disk should

have a file called SUBST.EXE on it. Put this line in your Autoexec file: SUBST n: C:\ACAD\dimame, where "n" is the letter of a drive that doesn't actually exist and dimame is the name of your drawing or symbol subdirectory.

Now when AutoCAD asks for a block name, you just have to type n:block-name. The SUBST command will make DOS think that your subdirectory is actually a disk drive.

Needless to say, the same technique can be used for Drawing files, although you may prefer my method for these: create and edit the file on the hard drive for best performance, copy the finished

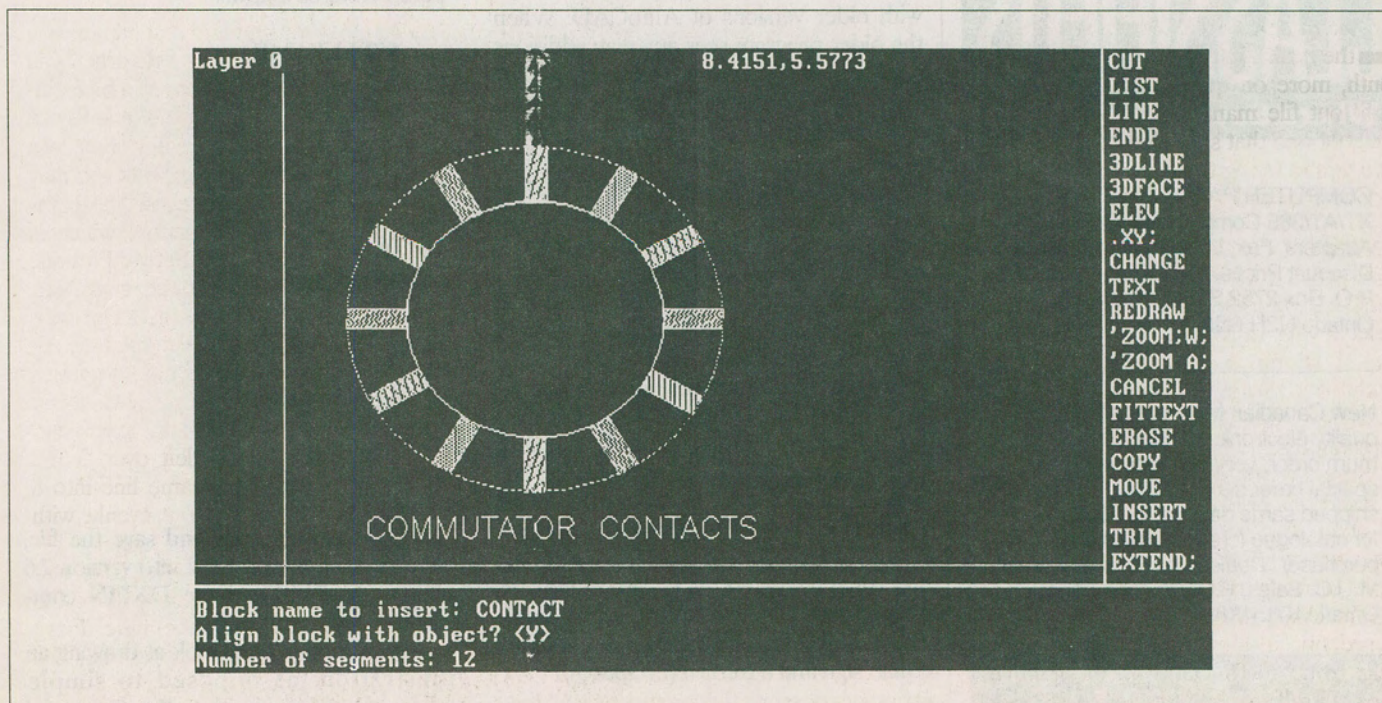


Fig. 2. The above Divide command can also be used for circles, arcs, etc.

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drawing onto a floppy disk, and then erase it from the hard drive. If you make vast drawings and have a huge hard drive, his obviously doesn't apply.

Files and DOS

It's inevitable that you'll load the drawing editor and suddenly find that you don't know where your block file is, or you'll remember that you left it on a floppy disk, or something similar. No need to exit the program — AutoCAD provides the SHELL function for this. You can run DOS commands or full programs; you'll be returned to the Command prompt when the program terminates. If you need to stay in DOS, type a Return when the words "DOS Command:" appear. When you're done, type Exit to return to the command prompt (the DOS prompt will appear doubled to let you know you're in the shell, like this: C:>).

If you get not-enough-memory errors, check your Acad.Pgp file by calling it into an ASCII word processor, such as WordStar in the non-document mode. Autodesk's default value for Shell is 127K; I found that increasing it to 256K runs any program I need. You'll also notice the command SH with a default value of 27K; this can be used for smaller programs or internal DOS commands.

Files In and Out

One of the problems in sending a DWG file out into the world is that it won't work with older versions of AutoCAD. When the older program sees any new additions it will dig in its heels and refuse to budge. Why Autodesk didn't see fit to let the older versions ignore new additions, I don't know.

The universally popular DXF file format suffers from the same problem; an older version of AutoCAD will turn up its nose if it sees new features in the DXF. Incidentally, loading the DXF format is badly explained in the Reference Manual, and you may have had a few "file discarded" messages while vainly attempting to import a DXF file. The trick is this: for unknown reasons, Autodesk insists that you load DXF only into a new file, but if you select a new file from the main menu, it won't work. It turns out that since AutoCAD configures each new drawing according to the ACAD.DWG template, it's not truly a new drawing... Odd, non? The cure is to follow your filename with an equals sign and a Return (no spaces):

Enter NAME of Drawing: BLEEN=

This loads a new drawing without reference to the template file; the very first command should be DXFIN. This should work every time if you don't have compatibility problems between versions.

One away around the software problem is the IGES file (International Graphics Exchange Standard). This is a very peculiar format which swells your file to five times its size or more (it converts the DWG format to ASCII) and at the same time, leaves out certain functions. I've had it ignore hatching, fills, and solids. On the other hand, it does mean making file transfers without worrying about device dependence.

Like the DXF, IGES drawings must be loaded into a new drawing using the *filename =* trick. If you make fairly large drawings, the bloated IGES format may overload any floppy disk that you use for transport. The drawing STPAULS, for instance, which is about 44K bytes, gains weight to 325K in the IGES mode.

Downward DXF

If you need to draw something in Release 9 and send a drawing file to someone with version 2.6, here's a fix that's cumbersome, but it works. First, use the Style command to set the text style to Standard. Then make a DXF file of your drawing. Load the DXF file into an ASCII text editor (WordStar in the N mode or the equivalent) and find lines 382 to 397. You should see four parameters, as follows:

```
$SPLFRAME
70
0
9
$SPLINESEGS
70
8
9
$ATTDIA
70
0
9
$ATTREQ
70
1
9
```

Delete these lines and save the file, which can now be loaded into version 2.6 using a new file and the DXFIN command.

Next month, we'll look at drawing an illustration, as opposed to simple schematics: dealing with Ortho, Snap, and Dimensions. ■