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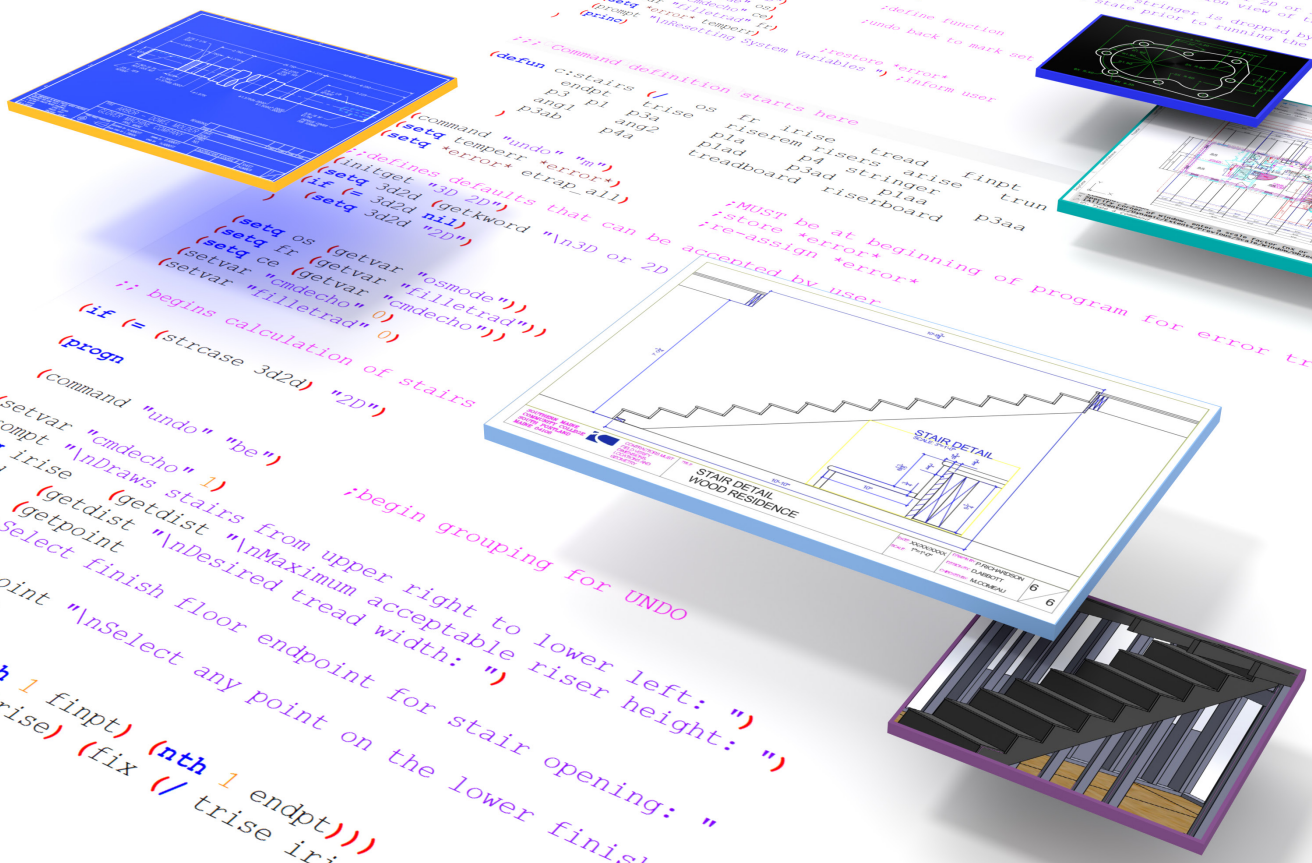
The AutoCAD® Professional's Handbook

Standards, Customization, Workflows and Automation



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CHAPTER 1

AutoCAD Productivity

Considering the complexity of AutoCAD, it's not surprising that many people who use it miss something important along the way. Many users had little training on the software before being expected to start producing useful work, and these users in turn have trained others based on what they figured out on their own. Even the most experienced AutoCAD users have likely forgotten some useful things they once knew.

This chapter is the result of the many questions I've been asked by AutoCAD users over the years while teaching, training, consulting, and responding to e-mails and phone calls. Here I'll offer advice on using AutoCAD, with an emphasis on features and techniques that are often overlooked or forgotten by users. I'll include general design advice that applies to the use of any CAD system, providing my recommendations for universal standard practice in using AutoCAD, reviewing techniques and commands that apply to all releases of AutoCAD.

This chapter isn't meant to be a comprehensive review of AutoCAD—many excellent books already provide that—nor is it a replacement for learning how to use the software. What I've selected here are items that people often overlook.

- Design Standards
- AutoCAD Best Practices
- Review of Selected Features

Design Standards

CAD software is used in so many fields of design that it would be impossible to develop extensive standards that apply to all of them. I've trained people who use AutoCAD to design quilts, hearing aids, doll clothes, houses, barns, commercial buildings, M16s, submarine hatches, and the myriad components of machinery. But there are some foundational rules that represent a consensus among serious users of CAD. You'll find exceptions to these rules, of course, but think of them the way you think of the rules for Dimensioning drawings: You follow them **if possible.** The fact that a rule has rare exceptions doesn't reduce its value as a guide. You certainly follow the rules requiring you to drive on the proper side of the road all the time—except when a dog darts out in front of you, or the road is washed away by a flash flood, or you're passing someone. So, here are some of my rules for using AutoCAD.

Draw Everything Actual Size

Unless you have a very good reason not to, draw everything at its actual size. Details can be drawn full size if you use Layouts properly. They may not look right to you in the Model Space tab, but you can display them in Paper Space Viewports and give them any scale you want. At one time, you might have created details separately and used the SCALE command to increase their size and then set DIMLFAC to compensate for Dimensions. There has been no need for that since Paper Space was introduced. You have a complex enough job as it is, keeping track of so many details. Why not simplify your life by drawing everything the size it's supposed to be? You're not at a drafting table. Worry about how something will plot when you set up a Layout.

I've done a lot of training for different industries and have looked for situations where it was impossible to draw full size; I haven't found an instance yet that couldn't be addressed using Paper Space. At one shop I worked for, two groups of designers who used AutoCAD had a difference of opinion about full size versus scale. One group of designers thought they couldn't draw full size because they were designing long pieces with almost no detail along their lengths but a lot of detail at the ends. They drew only the ends, then broke the piece with a conventional break and plotted it for the fabricators. The dissenting designers wanted to draw parts at their actual lengths so they could use them in assembly drawings without re-creating them.

The solution I offered was to draw the pieces full length, with proper end detail, and then create a Layout with two Viewports to represent each end of the object. As long as the two Viewports were at the same scale and aligned, they could be separated for a break symbol to be added in Paper Space. Even the Dimension of the overall length was correct, because it was in Model Space, where it belongs.

They could drag the value right or left so it could be seen in one of the Viewports. (See Figure 1.1.) The entire part is shown in Model Space above, Paper Space below with the conventional break created with two floating Viewports.

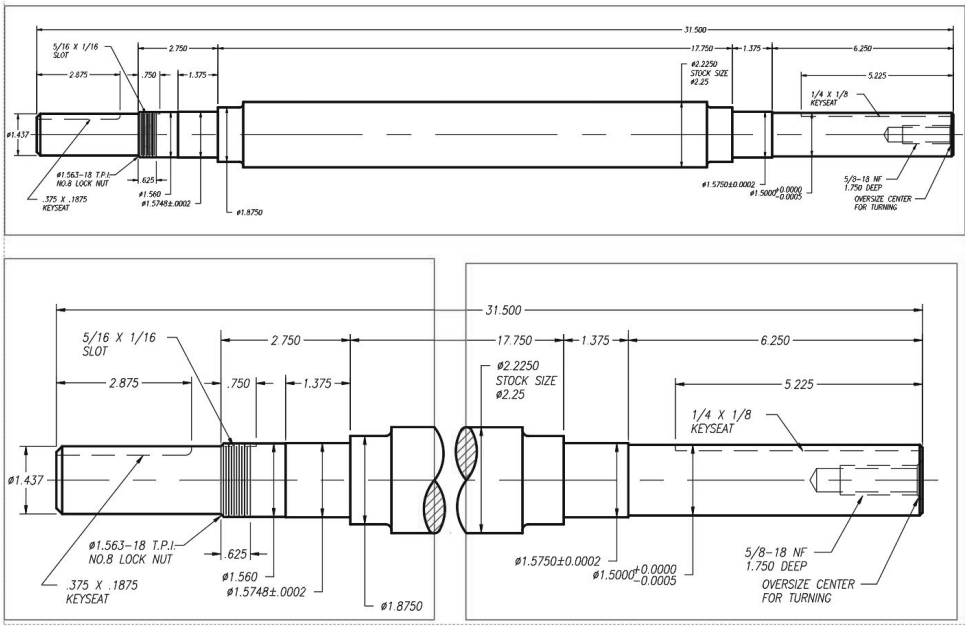


Figure 1.1 **Long parts with a conventional break**

Draw Existing Features "as Built"

This tip probably seems obvious, but I've been asked more than once what I recommend when designing for renovations or additions to existing structures. The question is usually in this form: "I have the original drawings of the building. Should I use them to create an AutoCAD drawing of the existing structure, or should I create the AutoCAD geometry 'as built'?"

Clearly, creating geometry "as built" rather than "as designed" permits you to solve problems in the software instead of in the field, because very few actual structures had no changes from the original plans. You can use the original plans to create a base drawing and then check key locations and Dimensions for changes. This is one of the great benefits of using such a precise design system. You can reduce what some builders call on-site engineering by drawing everything as accurately as you possibly can. That way, the results are much more likely to match the plans. Who knows, someday "as built" and "as designed" may become the same thing.

Draw Mechanical Parts at MMC

My advice to draw mechanical parts at Maximum Material Condition (MMC) may be less obvious than my "as built" advice, because mechanical parts always have a

specified tolerance—at least, they're supposed to. The question is, where in that range should you create your accurate-to-15-decimal-places geometry when using a CAD system? (You know it's impossible to make anything an exact size. If you think you can, you're not using a precise enough measuring tool.)

There are several possibilities. Some people draw objects in the middle of the size tolerance range. Others use what they consider the design size or nominal size—the base size given before the plus/minus sign that is sometimes used for tolerances. After all, isn't that the ideal size? Well, it may be the ideal size, but neither of these approaches is good practice, whether you're creating a 2D drawing or a 3D model. The fact is, a mechanical part has no ideal size. If a part is designed properly and given a functional tolerance, as required, it will work fine as long as its Dimensions fall within that tolerance. No size in that tolerance will make the part perform better than any other size. As a machinist, you may aim for the middle of a range, but not because that makes a better part. You do that to reduce the chances of making the part too small or too large and having to throw it out.

Draw all parts and create all solid models at MMC unless it's one of those rare features that are controlled at Least Material Condition (LMC) in a tolerance frame. Features with outside Dimensions, like a shaft or a pin, should be drawn at their largest acceptable size. Features that have inside Dimensions, like a hole or a slot, should be drawn at their smallest possible size. In both cases, the result is a part with the maximum amount of material in it. That's usually when parts are least likely to go together; this approach allows you to draw parts in the situation where they're most likely to fail and to check for interferences more readily. It's also easier to be consistent this way, because the rules for Geometric Dimensioning and Tolerancing assume MMC for many of the specific situations where the rules apply. I recommend this approach for both 2D and 3D modeling when you're using any other CAD package.

Use Logical Increments

When designing something, you can use any size increment you wish. If you choose increments that are easy to work with, or that result in less waste during fabrication, you save both time and materials. During the early design stages, you can set a SNAP to that increment in AutoCAD, with a GRID set to twice the increment. Doing so can speed up the initial Layout.

For architectural design, I recommend using the largest increment possible, such as whole inches, one foot, two feet, or four feet. Doing so makes it much easier to use standard sized sheets of material during construction. For mechanical parts, use increments of 2mm or 0.1 inches if possible, and avoid increments based on fractions that have been converted to decimal values.

METRIC IS EVERYWHERE

I've been predicting since 1976 that the U.S. was about to go fully metric. I keep making that prediction, and I'm getting less and less wrong. That may not be as good as being right, but it's very likely that you'll have to deal with the metric/inch conflict at some point.

I've assisted both mechanical designers and architectural designers in converting existing designs from inches to metric, and if this is done as a hard conversion, it often results in Dimensions that aren't logical to people who are used to working in the metric system. Sometimes that can't be helped — it may not be possible for a precision mechanical part to be redesigned from values in decimal inches into whole numbers of millimeters. It may be possible, however, for an architectural design to be soft converted to millimeters or meters without any ill effects. Does anyone use metric units for buildings? Well, yes. The whole rest of the world does, and some companies in the U.S. are beginning to find that out—for example, cabinet makers whose cabinets won't fit into the space available except in U.S. houses, and plywood makers who have no market for 4' X 8' sheets anywhere but here.

A log-home builder in Maine once sold a home to a company in Japan. The builder boxed it up, put it on a container ship, and then got the fax: What the heck are these numbers, and how soon can we get drawings in millimeters? When they contacted me, I suggested that they convert their shop drawings to millimeters by changing DIMLFAC to 25.4 and then setting DIMRND to 2. All Dimensions were then in whole, even millimeters. Nothing was off by more than one millimeter, so the design wasn't compromised, but the drawing Dimensions were clear to the Japanese crew who had to read them. Imagine how confusing it must be to see 7'-8'-3/4" when you're used to values like 2400mm.

AutoCAD Best Practices

I've been training people to use AutoCAD for technical graphics and design since the late 1980s. At that time, I frequently got resistance from drafters and designers who felt I placed too much emphasis on absolute accuracy. They would point out that once a drawing is plotted, the absurd 15 decimal places of precision that AutoCAD uses for calculations become meaningless. Once you plot a drawing, even significant errors are difficult to detect, as long as you carefully replace any key Dimensions with those you typed in. But a CAD file is more than a way to get lines and Dimensions on to a sheet.

I believe all industries that rely on computers for design and documentation should share common standards when using CAD software. In this section, I offer advice on accuracy and other aspects of AutoCAD that I believe should be standard practice across all disciplines. If you understand AutoCAD, it's much faster to produce an accurate drawing than an inaccurate drawing. You or your company have invested a considerable amount of money in the hardware and software necessary to use CAD, so why not produce drawings that are as accurate and useful as possible?

Use the Help System

The AutoCAD Help system is as good as that available in any software I've used. If I had to identify one AutoCAD feature as the single most underused, this is it. I've gotten many phone calls from people who have an AutoCAD question that I answer by simply going to the AutoCAD Help system. It's pretty good, although its coverage of AutoLISP functions was once much better.

Use Blocks as Often as Possible

Blocks can dramatically reduce file size, allow you to quickly update large amounts of work, and make your drawings more consistent. Any time you create a symbol, standard detail, Title Block, or other collection of objects that you may ever want to use again, consider creating a Block definition. Set up a Block library collection grouped into drawings in some logical way and then create tool palettes to make them easy to find.

Once you've used a Block, never explode it unless you have a very good reason. This is especially true for Dimensions, which are anonymous Blocks. Once you explode it, the Block entity no longer exists. You lose the ability to update all insertions, and you increase the file size, sometimes dramatically. Dimensions will no longer update values when you modify geometry, and you can't update Dimension appearance with changes to dimstyles. Don't explode Hatch patterns, either, for the same reason.

Never Override Dimension Values

When you're adding Dimensions to drawings, it's tempting to type in the correct value when a Dimension is wrong. Unless you absolutely don't have the time to do it, redraw the geometry so it's correct, and then add an associative Dimension. Otherwise, you and everyone who ever uses your drawing in the future will assume it's correct—with potentially dire results. You know you'll forget that you did it.

If you do override a Dimension because you just can't help it, then make sure you note the change so you can go back and re-create the geometry later when you have time.

When I'm Dimensioning architectural plans, I set the precision to an increment of 1/256" in the Primary Units tab of the Dimension Style dialog box. That's ridiculously small for a Dimension, but by using such a small increment of precision, I know immediately if there's an error in the geometry as I'm adding Dimensions. If there are no errors, the proper Dimension value is displayed. Do this for the precision of angles as well and set "suppress trailing zeros". When increment is set to zero, errors are rounded to the nearest whole value. I've seen problems with this in every CAD program I've used, particularly with angles for drawings used with CNC.

HELP SYSTEM

I'm not exaggerating when I say I answer many questions from confused users by bringing up the AutoCAD Help system—often while I'm on the phone with them. On time I got a call from a frustrated user who was using AutoCAD 2006 to edit text in a vertical title block. The version of AutoCAD they had upgraded from displayed the text horizontally; but AutoCAD 2006 displayed the text in place, which meant he had to tilt his head to the side to read it. He was about to rotate the title block for editing when he decided to call me.

I knew the MTEXTED variable would allow him to change the MTEXT editor to the previous version so he could use horizontal text, but I didn't remember the correct setting for doing so. As he was asking the question, I used the Help system to look up the MTEXTED variable, and then I told him to set it to OldEditor. He sighed and asked, "How can you know everything about AutoCAD?" When I admitted that in this case my secret was the Help system, I think I heard him slap his forehead.

No one can know everything about AutoCAD, but if you want to be the indispensable AutoCAD go-to user in your office, become familiar with Help. Whether you reveal your secret source is up to you!

Use Layers with Logical and Consistent Names

Using Layers with logical names allows you to separate different kinds of geometry and different functions. Don't be one of those people who produce nightmare drawings by placing all or most entities on a single Layer. Layers give you control over a drawing that's essential to efficient management. If your Layer names are logical, it's easy to manage multiple Layers.

That logic should be embedded in an office standard and reflected in the use of standard drawing templates, and a .DWS standards file. There have been many attempts to standardize Layer naming across disciplines. See Chapter 4, "Applying Graphics Standards," for more information, but any standard is better than none.

Set All Properties to ByLayer

Unless you have a good reason, avoid the urge to use multiple colors, Linetypes, or lineweights on a single Layer. If you get a drawing from someone who has done this, use FILTER or QSELECT to select and move objects to different Layers, and then change their properties to ByLayer.

Good reasons may include the desire to create a symbol or detail that contains multiple Linetypes and colors. Although this can be done by using objects on different Layers, many users prefer to have all elements of a Block definition reside on a single Layer so only that Layer controls the appearance of the Block. In this case, you can apply a specific color or Linetype to an element before including it in a Block definition.

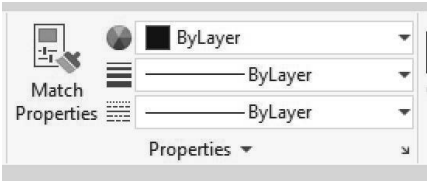


Figure 1.2 **Don't set properties here**

LAYER MANIPULATION

Consider the following example from a residential floor plan. Layer names all begin with a field that designates the floor of the residence: FL1, FL1-DIM; FL2, FL2-DIM; FND, FND-DIM. Each floor has a group of associated Layers for hidden lines, center lines, Dimensions, appliances, electrical, and so on.

Because the Layer names are uniform, you can use the following syntax either at the command prompt or within a menu to make changes to groups of Layers:

- Your code should look like this. At the command line each semi-colon would be replaced with an ENTER.

```
-Layer;T;*;S;FL1;F;~FL1*;;
```

The * is a wildcard meaning *all*. The ~ (tilde) is a wildcard meaning *all except*. The result would:

- Thaw all Layers.
- Set Layer FL1 as the current Layer.
- Freeze all Layers that don't start with the characters *FL1*.

If you find it necessary to apply a color or linetype directly to an object rather than to ByLayer, I recommend doing so with the Properties palette *after* the object is created. If you select the color in the Properties toolbar, you may forget to set it back to ByLayer and continue drawing. You should close the Properties toolbar to avoid using it. (See Figure 1.2.)

Use the Drafting Tools

Learn to use direct-distance entry, object snaps, object tracking, temporary tracking, and polar tracking. They're great tools that can dramatically speed up your work once you understand how they work together. If you are an old-timer, check out object snaps, like M2P, Temporary Track Point, Parallel, and Extension. However, don't overdo the running object snaps. Open the Drafting Settings dialog box and uncheck those you don't want all the time. See the chapter on getting started with AutoLISP for a way to automate a return to your favorite defaults if you do change them.

If you're drawing lines at unusual angles, and you want to be able to continue with lines that are perpendicular *from* the last line segment you drew, set PER as a running object snap and hover over the end you just selected. Make sure that object tracking is turned on.

Plot from Layouts in Paper Space

Read Chapter 6, "Plotting," for the full story; but for now, follow these steps in sequence when you create a new drawing file:

1. Create your geometry full size in Model Space, but don't add Dimensions, text, or Hatches.
2. Set up a Layout with all views at the proper annotative plot scale.
3. Add annotative Dimensions, text, Hatches, and schematic symbols to your drawing from a Layout with a properly scaled Viewport or in the Model Space tab after setting a correct annotative scale.

> The command CHSPACE can move objects between Paper Space and Model Space, but it will not move annotative entities correctly, so make sure that all your Dimensions are annotative and placed in an active Viewport or in the Model Space tab when you place them.

Draw Perfectly—Somebody Will Assume You Did

I've seen some horror stories about AutoCAD drawings that were done using the Etch-A-Sketch® method and then reused later by someone who assumed they were done accurately. Most of these stories involve hapless users stuck with their predecessors' drawings, but in some cases, users run into trouble by relying on their own inaccurate drawings. In one case, which involved a lawsuit and a lot of money, a bid was made on a structural design based on the assurance that accurate AutoCAD drawings of the existing structure would be provided by the client. Unfortunately, those drawings were facilities drawings— done to show egress and general locations—and were unusable for the intended purpose.

How does this happen? Because most AutoCAD drawings look accurate even if they aren't, we forget that we created something quickly by sketching and then saved the file.

> It's quicker to use the accuracy tools built into AutoCAD than to draw by eye, so why not use them?

Set Text Style Height to 0

If you set the height to a fixed value when creating a new Text style, the text can be used only at that height no matter how you set the text height in your Dimension style. Setting a height of 0 gives you control over text height every time you enter the text. If you can't read a Dimension because the text is too small, but you can see the arrowheads, you've specified a fixed height in your Text style.

Don't Use the Name Standard

The name Standard is used as a default name for the Text style, Dimension style, table style, mleader style, and Mline style. It's misleading, because it's a standard only in the sense that it always shows up in an AutoCAD environment. Never does the name represent a real standard in any discipline. To avoid a nasty surprise down the road, build a template file that banishes Standard as a name for anything.

I suggest naming Text styles for the fonts they use. If you set their height to 0, you won't need all those Romans48 type names; Romans will work fine. For Dimensions, give the styles names that represent their disciplines, or the name of the client. For tables, use names that represent their use. Or use the names of your favorite pets—just don't call anything Standard.

Be Cautious When Using REFEDIT

REFEDIT is used to edit external reference drawings or Block definitions; it replaces the originals if you tell it to. If you click Save Reference Edits, you'd better mean it. You can protect a drawing that will be used as an external reference (known as an XREF) from being edited with REFEDIT by setting the variable XEDIT to 0 before saving the drawing file that you intend to use as an XREF. Before AutoCAD 2006, double-clicking a BLOCK insertion opened REFEDIT. Now it opens the Block Editor. Double-clicking an XREF still opens REFEDIT. Users who don't understand what REFEDIT does will sometimes close the resulting toolbar and keep working. Unfortunately, they will still be reference-editing without knowing it and eventually will get the not in the working set error. If you get this error, type **REFCLOSE** at the command line.

Beware When Moving or Renaming Files

Don't change filenames or locations for hyperlinks, External References (drawings, PDF files, images), menu files, icon BMP files, or other support files unless you know how to redefine the path used to locate them. Otherwise, you'll get blank rectangles for images and a line of text for XREFs, your menus won't load, many commands won't work, or you'll see clouds or question marks on your custom toolbars. Using the Relative Path option can help for images and XREFs, but not always.

THE PROBLEM WITH *STANDARD*

Every AutoCAD drawing uses *Standard* as the default name for styles. At some point, you may insert your drawing into another host drawing. If you never bothered to rename the style you use, you'll have a conflict in the host drawing between its Dimension style and yours. Only one of the two styles named Standard can win this fight. Will it be your drawing, or the drawing into which it's inserted? Hint: Your drawing will lose this fight, and all your Dimensions will look awful if you explode the resulting block.

Control Imperial vs. Metric Settings

Starting from scratch with either an imperial (**acad.dwt**) or metric (**acadiso.dwt**) template controls the drawing's units, default Dimension Styles, insertion units, default plotting settings, and files used for Linetypes and Hatch patterns. **ACAD.lin** and **ACAD.pat** are used for imperial, and **ACADISO.lin** and **ACADISO.pat** are used for metric. You can change this using the MEASUREMENT variable. The imperial setting is 0; the metric setting is 1, but that won't affect anything that has already been created in the drawing.

Changing MEASUREMENT doesn't change the limits or the default Dimension style after the fact. When you plot a drawing starting with the metric template (or when you've changed the setting for Measurement to 1), and you select a paper size measured in inches, the plot scale is automatically set to 1:25.4. If you get odd results when plotting, check this value.

Learn to Use QSELECT

QSELECT is really useful when you're trying to fix a problem drawing. I once received a DXF file from the engineer of our city hall. He had generated hundreds of points using GPS equipment to map the location of sewers, drains, manhole covers, and so on. The problem he had was that all the points and all the text were placed on one Layer. The text height was so large that the text overlapped and was unreadable. (See Figure 1.3.)
To fix the problem, I did the following:

- I used QSELECT to select all the points and put them on their own Layer, which I immediately locked to protect the valuable locations generated by the GPS software.
- I used QSELECT to select all the text (height 294) and change its height to 5 so it would be manageable.

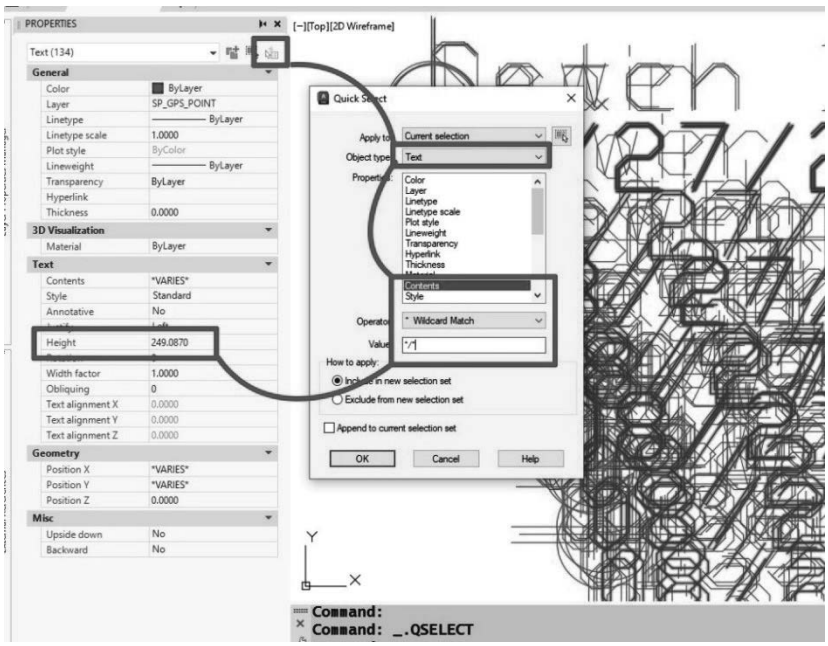


Figure 1.3 **Using QSELECT**

- I used QSELECT to select all text containing a forward slash (all the dates) and put them on their own Layer. This required using the ***** wildcard match operator (nice to know some DOS). By placing ***/*** in the Value window, you get all text containing a forward slash anywhere in the string.

Create Tool Palettes to Enforce Standards

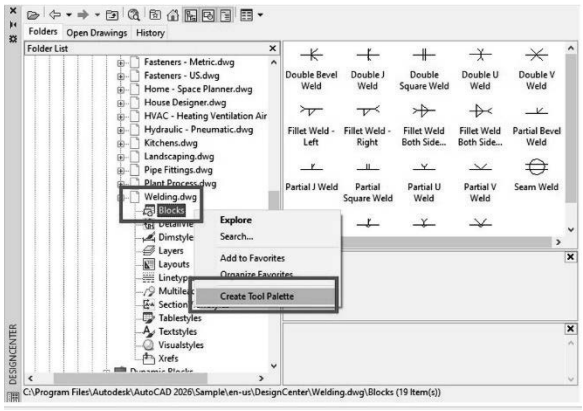


Figure 1.4 **Creating a tool palette from a symbol drawing**

Using AutoCAD DesignCenter (ADC), you can create a tool palette that contains all the Blocks from a symbol-library drawing with a single selection. Locate the drawing in the browser window of ADC, right-click it, and select Create Tool Palette, as shown in Figure 1.4.

Once you've created a tool palette, you can use it to enforce standards by setting the properties of any object on the palette, including the Layer it's on (all tools), the scale (Blocks and Hatch patterns), and rotation angle (Blocks and Hatch patterns).

To add a Hatch pattern, use ADC to locate the file **ACAD.pat** or **ACADISO.pat**, and drag and drop a pattern to the palette.

Feature Review (All Releases)

One of the difficulties with an application as complex as AutoCAD is that everything seems to change with each release. It can get a little discouraging. Why learn the nuances of anything when that knowledge may be worthless in 12 months? And if you do dig into a release and learn to use it productively, can you keep doing those things after the next release?

The fact is, many things about AutoCAD haven't changed over the years, including fundamentals like the underlying Cartesian coordinate system, the basic command structure, the way menus and toolbars work, the methods for creating and modifying most objects, and how files are saved. This section reviews the functionality that has been fairly constant in AutoCAD across many releases and is likely to stay that way. There can be a big difference in drawing efficiency between one user and another that has nothing to do with new features.

Earlier in this chapter, I discussed some rules and standards for using a CAD system. Here I'll point out general AutoCAD features, big and small, that a lot of users have missed. They aren't secrets; they just seem that way if you don't know about them. And because they aren't new features, they don't show up in the New Features Workshop as spiffy and new. I think of these elements as spiffy and old. I don't care what release of AutoCAD you use; there's something here for you.

Drawing Efficiency

At the heart of a CAD system is the ability to create accurate geometry. Speed is always secondary to accuracy, but it's possible for one user to be much more efficient than another in creating geometry while still maintaining accuracy. That efficiency isn't merely the result of being fast with a mouse and keyboard, it's the result of planning strategies for approaching each new object to be drawn. And, of course, you must be able to get information from the drawing to check the accuracy.

The following suggestions may improve your speed and accuracy in drawing and improve your ability to get information from a drawing quickly.

COMMAND-LINE VERSIONS OF COMMANDS

One technique for becoming more efficient with AutoCAD is to type aliases and commands at the keyboard. If you can't type, you may be out of luck; but when I watch keyboard jockeys use AutoCAD, I see countless places where they save a few seconds here, a couple of seconds there, and pretty soon it adds up to real time. But what happens if the command brings up a dialog box? You have to wait for the dialog to display, grab your mouse, make some picks, click OK, and get back to work.

It may be faster if you don't have to deal with a dialog box, and many AutoCAD commands have both a dialog-based version and a command-line version. When they do, you can issue the command-line version by placing a minus sign in front of the command name or its alias: for example, -AR or -ARRAY. This behavior is in AutoCAD to protect legacy applications written by users of past releases, but you may find it more efficient than toolbars, tool palettes, or pull-down menus.

Knowing how to bring up a command-line version of a command is a godsend when you write AutoLISP programs. You can check the sequence for creating Layers, for example, by typing **-LA** or **-LAYER** to avoid the dialog box and see the prompts.

Speaking of the LAYER command, let me use it as one example of efficiency. When I need a new Layer—say, one named "DIM"— I'm about to draw, I almost always type something like this: **-la** ↵ **m** ↵ **DIM** ↵. (The ↵ symbol represents the ENTER key) I just timed it: It took me six seconds. DIM is now the current Layer, and I'm drawing away. I did the same thing using the Layer Properties Manager dialog box, and it took me 16 seconds. You may see that as saving only 10 seconds. I see that as a 267 percent improvement in my efficiency.

This technique doesn't suppress all dialog-based commands. If you want the command-line version of the SAVE command, for example, you need to first set the variable FILEDIA to 0. Then, typing **SAVE** (no minus sign needed) won't display a dialog box, but will give you save options at the command prompt.

Array Commands

There are now three versions of an array command, the most recent of which is an associative array added in Release 2012. It includes an option to array along a path, in addition to the rectangular and polar option, and the array that is created using any of the three types becomes a single object that can be edited after being created. It has only a command-line version, and many users find it more awkward to use than the dialog box version of ARRAY that is now named ARRAYCLASSIC. It is possible to use the ARRAY command to create a non-associative array, or to explode an associative array that has been created if you don't need or want the editing features. In that case, you might want to use ARRAYCLASSIC since the dialog box is more intuitive than the command line version. The third option for arraying objects is the original command-line version, which can be launched by typing a negative sign as a prefix to ARRAY. That can be helpful in writing AutoLISP programs.

The addition of the Path type of array adds a function that has been available in both the DIVIDE and MEASURE commands, but only when inserting Blocks by name. Any entity can be arrayed along a path now by either method using the Method option. The editing function is thorough and powerful, but there are still some advantages to using ARRAYCLASSIC, which can be very helpful.

Rectangular Array

When you're creating a rectangular array, it's easier than you think to confuse columns and rows when you're asked to specify their distance. Columns are vertical, and the icon illustrates that in the dialog box. Keep in mind that the distance between rows and columns is the distance from a point on one item to the same point on the next item.

Figure 1.5 **Precision array at an angle**

You can create a rectangular array at any angle by selecting the "Angle of Array" option in the Array dialog box, even if you don't know the angle. Suppose you want to array the shape on the left in Figure 1.5 to form the shape on the right.

In the example, both the angle and the size of the object are randomly assigned. In other words, you don't know the exact size or angle, and you can't estimate because you want the result to have no gaps or overlaps of the objects as they're arrayed. You can approach this situation using two methods, one of which works with either the command-line version or the dialog-box version of ARRAY.

Dialog Box Array

To create the shape in Figure 1.5, follow these steps:

1. Run the ARRAY command, pick the Select Objects button, and select the Spline object.
2. Enter **1** for the number of rows and **4** for the number of columns.
3. Pick the "Angle of array" button shown in Figure 1.6; The dialog box will close temporarily to allow you to select two points in response to the prompt **Specify angle of array**.
4. Using the Endpoint object snap, select each end of the Spline. Doing so returns an angle and places the value in the window at the precision set by AUPREC.

5. Pick either the Column Offset or Unit Distance button, which results in the prompt **Specify distance between columns**.
6. Using the Endpoint object snap, select each end of the Spline once again. Doing so returns a distance this time and places that value in the window at the precision set by LUPREC. Click the Preview button; if you like the results, you're done.

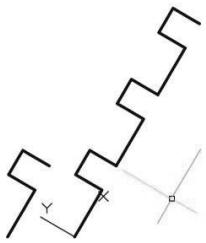
Command-line Version of Array

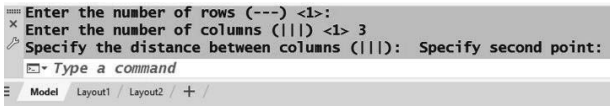
You may decide to automate a process like this using AutoLISP. In that case, you need to follow these steps, using the command-line version of the ARRAY command:

1. Run the UCS command, and type **O** ↵ to set a new UCS origin. O isn't displayed as an option. It's a hidden option of UCS. Select one endpoint of the object.
2. Run the UCS command again (pressing the space bar or ENTER key will repeat the last command), then type **Z** ↵ to rotate a UCS around the Z axis.
3. Select the other endpoint of the object to indicate the angle of rotation.

Any time you're prompted for a distance or an angle in AutoCAD, you can select points on the screen instead of typing them in. Doing so is both fast and accurate. When you start creating your own commands in AutoLISP, you can use specific functions to make your commands behave the same way.

4. Run the ARRAY command at the command prompt (**-AR or -ARRAY**) and select the object you want to array at an angle.
5. Press the Enter key at the Enter the type of array [Rectangular/Polar] <R> prompt.
6. Specify one row and four columns, and you'll be prompted to give the distance only between columns.
7. Select the two endpoints of the object again to give the exact distance.





8. Use the UCS command to return to the World Coordinate System (WCS).

Because WCS is the default, you can type **UCS** and press the Enter key twice—once to execute the command, and once to accept the default. That will return you to the World Coordinate System without changing the geometry of your drawing.

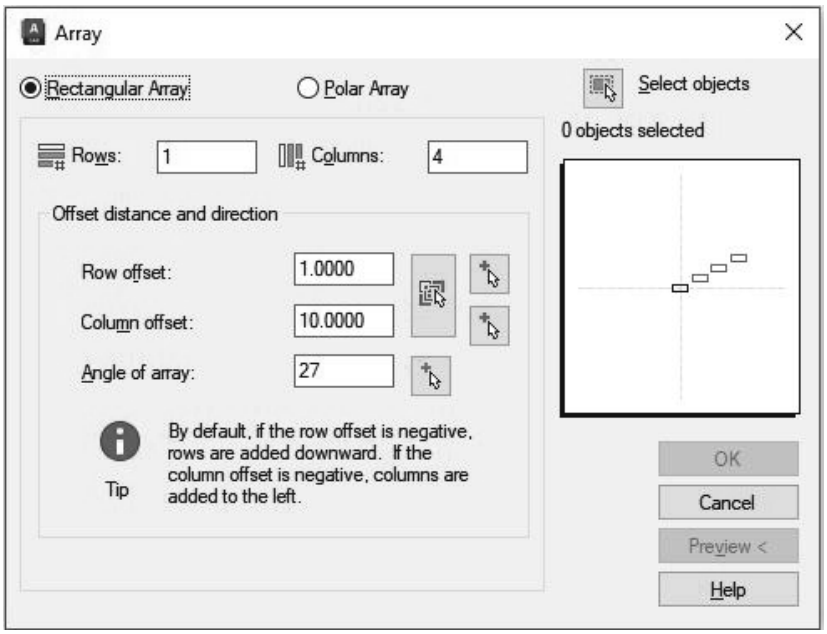


Figure 1.6 **Array dialog box**

DISPLAY PRECISION

Don't be fooled by the integer of 27 shown for the angle in Figure 1.6. The actual angle may be different. Why? Because the default precision for angle display is 0, which means all angles are rounded to a whole number for display purposes only. If I change the angular precision using either the variable AUPREC or the Units dialog box, you'll see the angle in this case is actually 27.37591770°. This often fools people when linear distances are displayed, as well. (Linear precision can be set with the LUPREC variable or the Units dialog box.)

This doesn't affect the actual precision AutoCAD uses for calculations. All calculations are done to 15 decimal places no matter how the results are displayed, but it is easy to be misled.

This is true for every CAD program I have used. Angles always have a default precision of zero, which is probably OK for many disciplines, but mechanical design that is used to control CNC machines requires a greater precision for angles, so thinking the angle displayed is correct can cause serious errors in manufacturing. Always increase the angular precision to at least 2 decimal places when designing mechanical parts so you can get an accurate value. You might even want to increase the precision in the Dimension style for angles and check off "suppress trailing zeros" so adding angular Dimensions will give you an accurate result if there is an error.

Polar Arrays

Geometry with a repeated angular pattern is often found in mechanical applications and sometimes in architectural and civil applications. To create polar arrays efficiently, avoid repeating the same set of editing operations for each feature arrayed. Start by identifying a repeatable pattern on the object. Draw the whole pattern once, and then array the result, rather than arraying each of the components of the pattern separately. Be careful. It's easy to select an extra entity when you use the Polar option of ARRAY. The result looks fine, but you may have entities on top of each other. The image in Figure 1.7 illustrates an efficient technique for using polar arrays by identifying and creating the repeatable pattern before arraying anything. The repeatable pattern is shown on the left, and the result of doing a single array of that pattern is on the right. The alternative involves multiple construction lines and nine times more editing for each feature.

PEDIT

After using ARRAY to create geometry from lines and arcs, you can determine whether the result is perfect using the JOIN or the Join option of the PEDIT command. The entities can only be joined if they connect perfectly at all intersections. Your geometry is perfect if JOIN creates a single entity or if the first option of PEDIT, Close, changes to Open after you've joined all your lines. If the geometry isn't perfect, you may still be able to join the segments into a single object using the MPEDIT command and setting a fuzz factor.

```
Convert Lines, Arcs and Splines to polyline
Enter an option [Close/Open/Join/Width/Fit/
Join Type = Extend
Enter fuzz distance or [Jointype] <0'-0">:
```

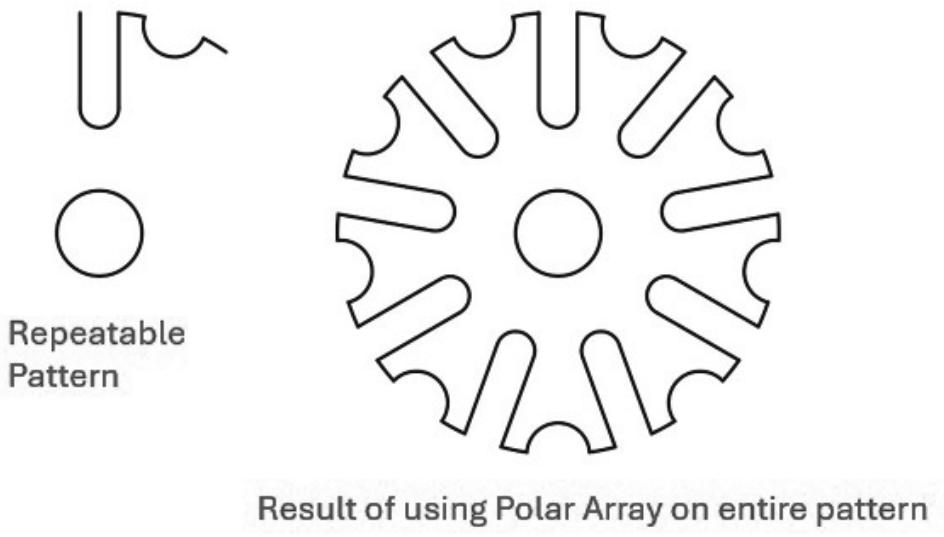


Figure 1.7 **Arraying the repeatable pattern**

REDUCE THE USE OF CONSTRUCTION LINES

Try to use as few construction lines as possible when creating geometry. The more lines you use to locate points or edges, the more chances that you may confuse one or more for actual lines or leave lines behind that are on top of each other. I've seen this result in subtle, but often important, errors.

The often-related problem of lines on top of lines may seem like no big deal, because everything plots fine; but if you have a short line segment on top of a longer one, it's possible to snap to the wrong endpoint or midpoint without noticing it and create inaccurate geometry. The OVERKILL Express Tool can help fix this problem.

MPEDIT allows you to set a fuzz factor for combining lines, arcs, or PLINES into a single object. To use it, run MPEDIT, and then select all the objects you want to join. You're prompted for a fuzz factor. Enter a number that is greater than your likely error, and MPEDIT cleans up the mess. (I hope you're working on someone else's mess, because you should be able to avoid sloppy drawing if you use the accuracy tools available in AutoCAD.)

There are also some alternatives to using a single closed object for finding areas, even if your geometry has overlapping lines. Using the BOUNDARY command, you can select a point and have a closed PLINE or a REGION created automatically. This works great for interior spaces, but you can use this command even in situations that are less obvious. For example, when you have overlapping construction lines and want to create a clean set of double lines (a floor plan, for example), use the technique shown in Figure 1.9: Enclose the entire group of lines with a closed shape like a rectangle or circle, and select two points: one inside the rectangle but outside the lines, and one inside the lines.

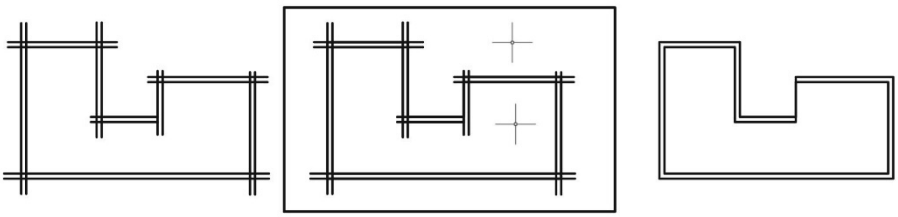


Figure 1.8 **Using BOUNDARY for quick shapes**

Alternative to PEDIT

You can create regions with the BOUNDARY command or with the REGION command. Either way, you can also use regions to create complex objects quickly by using the SUBTRACT, INTERSECT, and UNION commands. The shape shown in Figure 1.9 was created in under one minute (actually, 38 seconds).

To create regions from existing closed PLINES or CIRCLES, use the REGION command and select the objects. You can create a new object using SUBTRACT, UNION, or INTERSECT, and that object is also a region. Once it is created, if you need to work with individual lines or arcs, use the EXPLODE command to break the region into entities.

> If you have trouble understanding what a region is, think of it as a flat 3D object with a thickness of 0. The 3D Boolean editing commands will work with any region; the REGION command can be useful for other applications as well, including creating floating Viewports in a Layout.

Drawings with Interior Angles

For many drawings, you know the length of each line and the angles between lines. However, you probably don't know the absolute angle of each line in the X-Y plane (angle from 0° or East in the Cartesian Coordinate system). There is a strategy for doing drawings of this kind; it involves recognizing that supplemental angles form a straight line (180°). It also involves using the often overlooked Relative option of AutoCAD's polar tracking feature.

To set polar tracking to Relative, right-click the POLAR button at the bottom of the screen and select Settings. Select the **Relative to Last Segment** radio button in the Polar Tracking tab of the Drafting Settings dialog box. Select the **Track Using All Polar Angle Settings** option as well, as shown in Figure 1.10.

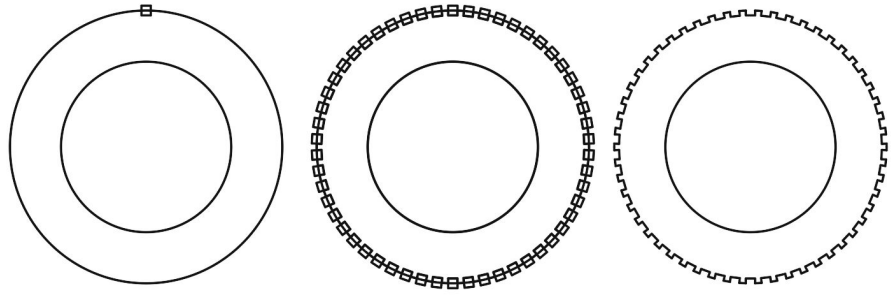


Figure 1.9 **New shapes from regions**

Note that in this case, Increment Angle is set to 15 degrees. Any angle can be used, but small angles aren't easy to work with. If the angles you're using are based on larger increments, use them. It's much easier to use an angle increment of 5, 15, or 45; but *any* increment can be typed into the window, including values so small they are completely unusable as angles. That's one of the things I like about AutoCAD. It doesn't restrict what you can do by overprotecting you from the results of commands.

In Figure 1.11, supplemental angles are determined from the interior angles and used to calculate the resulting relative angle of the next line segment. As you draw, use the readout from polar tracking to determine the relative angle for the next segment. Sometimes it will be reported to you as 60 instead of 120, for example, because it's increasing in the clockwise direction rather than the default counterclockwise direction.

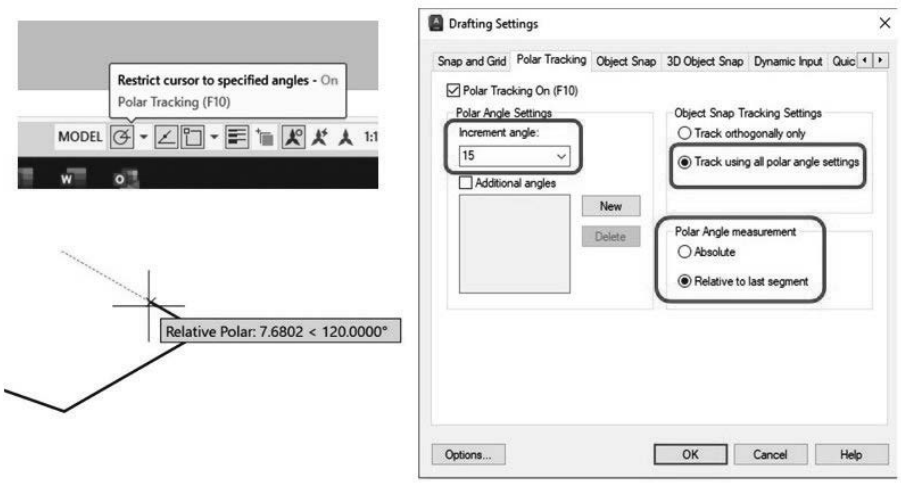


Figure 1.10 **Polar tracking settings**

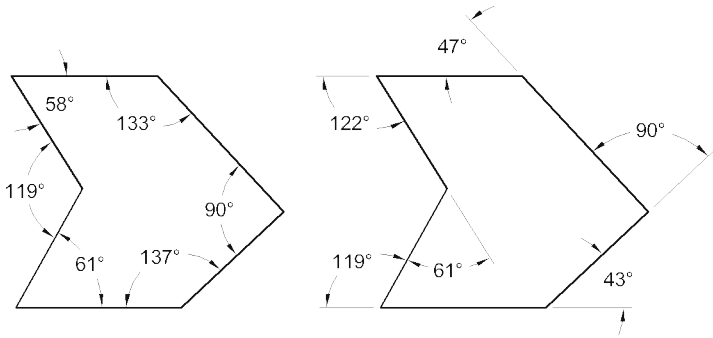


Figure 1.11 **Supplemental angles with relative tracking**

You can change the default direction of angles from counterclockwise to clockwise when it's easier to draw clockwise. But if you change the angular direction from counterclockwise to clockwise to simplify drawing a single shape, make sure you change it back to avoid confusion later. The Cartesian system assumes that angles increase counterclockwise.

Use From, Auto Tracking, or the @ Symbol to Begin Drawing

If you know that you need to start drawing an object a specific distance from an existing object, use the From Osnap with the @ symbol. For example, if you want to start a rectangle 10 units over and 45 units up from an existing endpoint, do the following:

1. Start the RECTANG command.
2. Use the From Osnap.
3. Select the starting point.
4. Type the coordinates for the first corner of the rectangle as **@10,45**.

If you have POLAR, Osnap, and OTRACK on, and the Extension Osnap is set as a running Osnap, you can pause over any running Osnap (endpoint in Figure 1.12), acquire a temporary tracking vector, and type the distance from the acquired point to quickly start relative to another object.

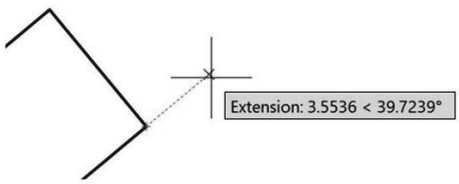


Figure 1.12 **Hover over an Osnap to acquire a point**

Even better, play with the Temporary Track Point Osnap until you understand how to use it, because it allows you to chase points all over the screen by typing TT prior to acquiring a tracking point. The new tracking point is temporary until you select it. I use TT a lot, and if you haven't figured it out, it's worth trying.

The @ symbol can always be used by itself to select the last point you entered.

Using CIRCLE and FILLET to Create Tangent Arcs

Another common shape involves tangent arcs. The quickest way to create an inside (concave) arc between two circles is to use the FILLET command with the proper radius set. However, an outside (convex) arc can't be drawn between two circles or arcs using FILLET. You must use the CIRCLE command with the TTR option and trim out the unwanted portion of the circle. The location of your cursor on the circles when you select tangent points determines whether a concave or a convex arc result. (See the Deferred Tangent Tooltip in Figure 1.13.)

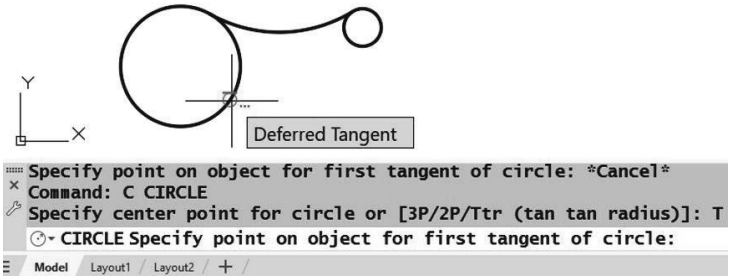


Figure 1.13 **Circle TTR for tangent arcs**

FILLET

To connect two parallel lines with a tangent arc, select both lines when prompted and AutoCAD will calculate the size of an arc that it will use to join the ends nearer your selection. For non-parallel lines, hold the Shift key while selecting two lines and they will be connected with a zero radius to create a sharp corner.

DTEXT (Single Line Text)

Most justification options are logical. Every line of text has four vertical locations for justification: Top, Middle, Baseline, Bottom, in that order. The Bottom is a line running through the lowest point on a lowercase letter with a descender (j, g, p, y). These locations are shown in Figure 1.14. The Justify options of DTEXT are `[Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]`. The two-letter combinations match the locations: TL is Top Left, ML is Middle Left, and so on.

Extracting Information from Drawings

You can use geometry created in AutoCAD to get information accurate to 15 decimal places. However, the information you get out of AutoCAD is only as good as the accuracy of the geometry you create. Following are some tips on using inquiry commands.

AREA

It's easier to work with CIRCLEs, PLINEs (POLYLINEs), or REGION entities when determining areas, particularly if you're trying to add or subtract areas to get a final total. This section will give you some advice about using other entities to create PLINES or REGIONs so you can easily find their areas.

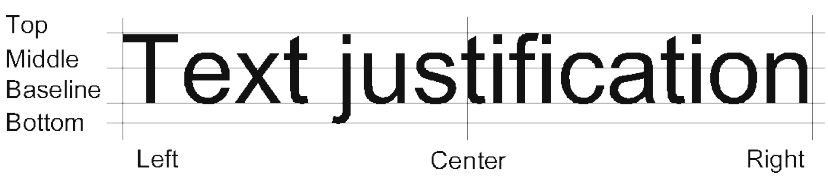


Figure 1.14 **Understanding DTEXT justification options**

JOIN and PEDIT

You can use either the JOIN or the PEDIT command to create a closed PLINE from multiple LINE and ARC segments or to create a single SPLINE from multiple SPLINEs. To use the JOIN command, execute the command and select all the objects you want to join with one of the many selection options. It will only join connected entities, which will become single entities. JOIN is more direct than PEDIT, which doesn't work with SPLINEs, but PEDIT has some additional options that can be useful for editing. To use the Join option of PEDIT, issue the command, pick one of the entitles or select the Multiple option, answer Yes when asked if you want the entities turned into Polylines, then select the JOIN option, and pick the other segments with a window. Remember:

- All segments must be touching but not overlapping.
- You can't have segments on top of each other.
- The resulting PLINE should be closed.
- Use the Fuzz option of MPEDIT if your geometry isn't exact.

The variable PEDITACCEPT affects the prompts you get with PEDIT. If it's set to 1, the user isn't prompted with the question `Object selected is not a polyline. Do you want to turn it into` one? <Y>. Instead, the object is automatically turned into a PLINE. 0 is the default and is saved in the system registry. If you don't want to have to answer that, set it to 1.

Entity Selection

When you use the AREA command, it's easier to select objects than it is to pick points. To find the area of a large object minus the areas of several smaller objects, you can use the Add and Subtract options of the AREA command. Pay attention to the command prompts. Type **AREA** ↵ **A** ↵ followed by **O** ↵, select the largest entities, and then right-click when you finish; doing so adds the areas of all the objects you picked. Type **S** ↵ to select the Subtract option, followed by **O** ↵ for Object, and select each object whose areas you want to subtract.

The alias for the AREA command is AA, which doesn't follow the normal pattern used for other commands. Why not? A (ARC) and AR (ARRAY) are already taken. This can cause a problem, though, because it is easy to type ARR thinking it's the alias for AREA.

Quick Area Calculation for Complex Features

It's not unusual to want to find the area of an object with many closed features removed. It's easy to miss one or more objects in the process. The quickest way to get a total area when you have many areas to subtract is to create a Hatch pattern by selecting an internal point with the HATCH command with Island Selection checked, which is the default.

In Figure 1.15, it took 15 seconds to get the total area of the object with all internal shapes removed. Here are the steps:

1. Execute the HATCH command.
2. Select a point inside the overall shape, but outside all the internal shapes.
3. Use the Object option of the AREA command and select the resulting Hatch. Using the Last option to select the last entity created is quick. You can also find the area as a property of the Hatch in the Properties palette.

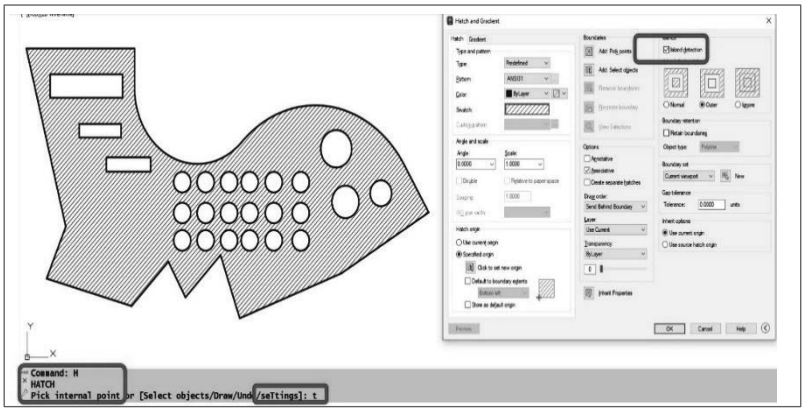
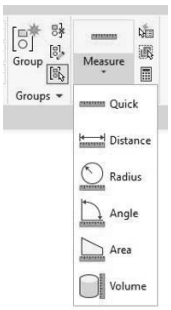


Figure 1.15 **Using a Hatch pattern to calculate a complex area**



MEASURE

The MEASURE tool (MEASUREGEOM at the command line) combines several commands that can be used to extract information from an AutoCAD file. It appears on the Ribbon. The Quick option allows you to hover over any entities and get a quick display of various values, including lengths, diameters, radiuses, and angles. I don't find it very useful because it displays so many values, many of which are meaningless in specific disciplines, but it is worth trying out. I usually prefer the direct application of the other tools, either from the ribbon or command line.

DIST

The DIST command gives you an accurate distance if you use object snaps to pick the points. It's also the quickest way to get an angle in the X-Y plane. Pick the points in the proper order, or the angle will be off by 180°. AutoCAD is based on the Cartesian Coordinate System, so angles are measured counterclockwise by default. To get an angle between two lines, use DIMANGULAR (DAN), which is more flexible than the Angle option of the MEASURE tool because you can move the cursor to see the angle or its supplement. The lines don't have to exist; you can select three points with DIMANGULAR and get the angle between the segments defined by the points.

PROPERTIES and LIST

Both these commands give you information that varies by entity selected, including the arc length of an arc. PROPERTIES can be used to change the properties of multiple objects as well as individual objects. For example, the height of many text entities can be changed without changing the locations.

One piece of information that you can get from the LIST command that you can't get from the Properties palette is an object's handle. This may seem meaningless to you now, but the handle can be useful because it's the unique identifier of an object and never changes.

TIME

TIME gives you the total time during which a drawing has been open for editing. You can also use it as a running timer by selecting the Reset option before beginning something new. Use this timer as you try to develop more drawing speed. Pick one object and draw it several times using different strategies. The TIME command can tell you if you're getting any faster. I've also used the TIME function to see if one computer in an office really is slower than another, or if it's just the perception of a frustrated user. See Chapter 7, "AutoCAD Scripts" for a benchmark system to test a computer's performance with no user input.

The Four SAVE Options

You can save drawing files using any of the following commands: SAVE, QSAVE, SAVEAS, and WBLOCK. Each of these commands creates a DWG file in different ways, but each has its own advantages.

SAVEAS When you select Save As from the pull-down menu, or type SAVEAS, AutoCAD saves to a specified path with a specified name and makes the resulting path and filename the default. The next time you use a Save command, including QSAVE (Ctrl-S), this name and location will be used, so it is the way to change the location and/or name of a file.

SAVE The SAVE command, if it's issued by typing at the command prompt, allows you to save a drawing to a different location or name without redefining the default drawing. The Save option on the pull-down calls the QSAVE command. I wish all Windows software had this kind of save option because it makes it easy to create a backup file while continuing to let you work with the current path and name.

QSAVE The QSAVE (quick save) command automatically saves to the default location and name. Ctrl-S issues the QSAVE command it can be issued from the pull-down menu.

WBLOCK WBLOCK is a save command. It lets you save part of an existing drawing, using one of three methods. See Figure 1.16. Many users don't realize WBLOCK is a save command. I think PSAVE ("partial save") would have been a better command name. See Chapter 5, "Symbols, Tables, and Fields," for more information on the WBLOCK command.

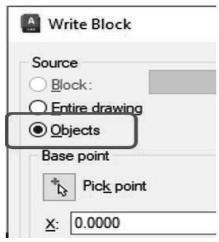


Figure 1.16 WBLOCK options for saving a DWG file

GROUP The GROUP command should be used more often, in my opinion. It allows you to name a selection set so that you can use it again.

Once a group is created, you can type **G** ↵ at any selection prompt followed by the name of the group you created (make the names short and memorable). However, the group must be selectable. You can change whether a group is selectable using the GROUP dialog box.

PICKSTYLE The settings of the AutoCAD variable PICKSTYLE determine how other members of a group are treated, including associated Hatch patterns, whenever a single member of the group is selected. PICKSTYLE has four settings that you can toggle using the Ctrl-H keys.

- PICKSTYLE = 0 allows selection of individual members of the group.
- PICKSTYLE = 1 allows the selection of the whole group but not boundaries.
- PICKSTYLE = 2 groups the selection of Hatches with boundary objects.
- PICKSTYLE = 3 allows the selection of whole groups and group selection of associative Hatches with their boundaries.

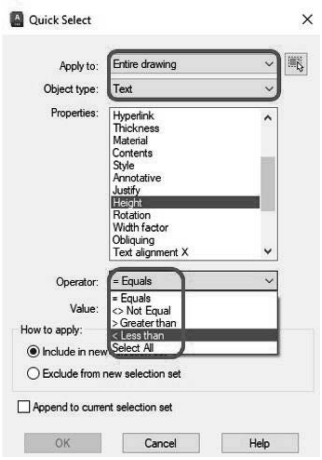
If a Hatch boundary is erased when you erase the Hatch, check your PICKSTYLE setting.

Edit Commands

One of the most underused editing functions in AutoCAD is the Properties palette. Some of the editing commands can be used in ways they weren't necessarily designed for, and there are command options that a surprising number of people overlook: the Through option of the OFFSET command and the Reference option of the SCALE and ROTATE commands, for example.

PROPERTIES

This command displays the Properties palette of selected entities. If more than one object is selected, the properties common to all are displayed. Properties of similar objects can be changed simultaneously, and objects can be filtered in the window that lists selected entities or using the Quick Select button.



Using the Quick Select button, you can filter objects in a selection set based on multiple criteria by applying each criterion once and then selecting the current selection set for the next application. This is a great way to clean up problem drawings that you get from someone else. You can change the Layer of all text, for example, or the style used.

Let's walk through an example. Let's say you have a drawing that has a lot of text. Every text entity must stay in its current location, but the height of all text less than .25 in height must change to .12. Open the Properties palette and click the Quick Select icon in the upper right corner. Apply to Entire Drawing, select Text in the Object Type window, and Height in the Properties: window. Select <Less Than, in the Operator: window, enter .12 as the value and the text heights will change to .12. You can change any of the properties of any selection set you create.

CHAMFER and FILLET

You can use either of these commands to clean up sharp corners or extend two unconnected lines into sharp corners by setting the value to 0. You also have the following option: Hold down the Shift key while selecting the second line to create a sharp corner no matter what the current setting is for fillet radius or chamfer.

If you're having trouble snapping to what you think is an intersection between two lines, use FILLET with a radius of 0, select the two lines, and then try again. If that works, you don't have to take the time to zoom into the intersection to see if there's a gap.

You can quickly create a slot by selecting two parallel lines when prompted by the FILLET command.

DIVIDE and MEASURE

These commands behave similarly. The difference is that DIVIDE results in an entity divided into a specified number of segments, all the same length. MEASURE divides the entity into segments of a given length, with one shorter segment (usually) at the end. Entities aren't really segmented; instead, a point is placed at each division.

It's also possible with either DIVIDE or MEASURE to have a Block inserted at each division instead of a point, which is a useful and often overlooked feature. You can quickly approximate a complex Linetype, for example, by placing any Block along a line, Pline, or Spline object. This works well for placing arrows on an egress map. (See Figure 1.17).

You can accomplish the same thing using the Path option of the ARRAY command that replaced the renamed CLASSICARRAY command, which I prefer to use.

The appearance of all the points in a drawing (except those on the Defpoints Layer) is controlled by the variable PDMODE. You can set this variable using the Point Style dialog.

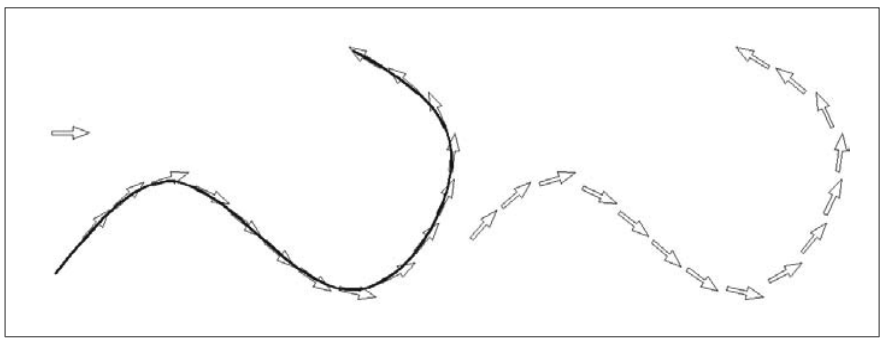


Figure 1.17 **Using Measure to place blocks along an entity**

EXPLODE

You can use EXPLODE with the following entities: Blocks, Hatches, Mlines, Plines, solid objects, array objects, and Blocks of text created with the MTEXT command. Blocks can be exploded unless the property Allow Exploding was changed from its default setting of Yes, or it was placed using the MINSERT command. So, avoid MINSERT if you want the option of exploding later. You use one of the three ARRAY options instead.

The TXTEXP Express Tool can explode individual pieces of text into vectors. The results aren't always pretty, but I've used this technique successfully to create cutting-tool paths for CNC machines used to mill letters in either metal or wood. The results differ between SHX fonts and TTF fonts. Shape fonts, which are vector-based, are exploded into line segments which can be joined into single entities. TrueType fonts are exploded into closed polylines. Often a single letter requires multiple closed polylines that must be edited into a single outline.

You can form 3D block letters by using TXTEXP to explode a TrueType font and then using EXTRUDE to create 3D solids out of all the segments used for each letter. Use UNION to create a single solid.

OFFSET

This command has always had two options: Distance and Through. The Through option allows you to offset an object through a selected point, even if the object isn't long enough to pass through the point. There are other options as well.

You can offset an object onto the current Layer. You can also elect to erase the source object after using offset. That sounds like it would just become the MOVE command, which it would, except that the multiple option lets you continue picking through points or offsetting the same distance multiple times.

OOPS

People use UNDO sometimes when they really should use the OOPS command. OOPS can be used at any time to restore all the entities erased as a single selection set the last time the ERASE command was used. It doesn't have to be issued immediately after the entities are erased. Once in a while – more often than I should – I erase objects that are in the way of a delicate editing operation, and then I use OOPS to get them back as soon as I'm done editing. I don't recommend this approach, but it works for me.

OOPS restores objects even if their Layer is currently frozen or turned off.

ROTATE

The remarkably useful and often overlooked option of ROTATE is Reference. You can use the Reference option by entering any angle at the keyboard or by picking two points on the screen. You then type the angle to which you want the selected points to align.

Instead of typing the new angle, you can also make one more selection. It becomes the second point on a new angle, using the first point selected as a reference for a base point.

SCALE

SCALE also has a Reference option like ROTATE. I use it frequently to resize an image using a feature on the image that I know the size of — this can be a photograph of landscape, nautical charts, a mechanical part, or anything else that's not to scale when inserted. In the example in Figure 1.19, a National Oceanic and Atmospheric Administration (NOAA) nautical chart has been referenced as an image. Scaling it using the Reference options involves the following steps:

1. Issue the SCALE command.
2. Select the edge of the image.
3. Select a base point.
4. Type **R** to select Reference.
5. Select a start point. In this example, the latitude line showing 43 minutes is the first point.
6. Select a second point. In this example, the latitude line showing 44 minutes is the second point.
7. Type **1** for one nautical mile, the base unit of this drawing. Because the distance from 43´ to 44´ is one minute of latitude, which represents one nautical mile on the earth, the image scales correctly. This is not a common use for AutoCAD, but I use it to create accurate charts for teaching navigation and for laying out track lines for use on a boat.

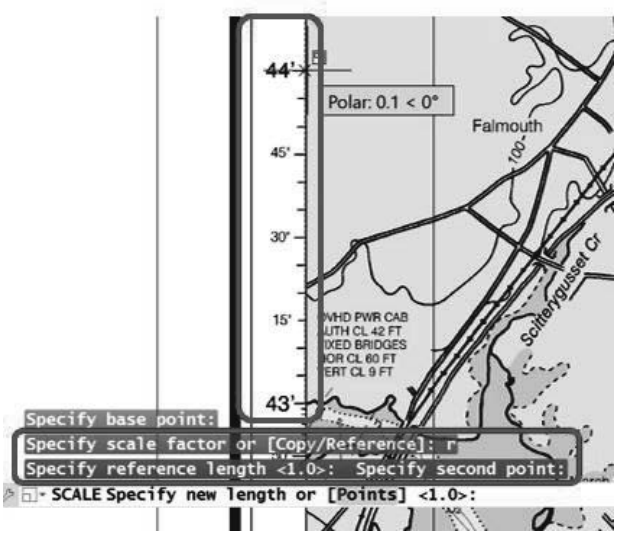


Figure 1.18 **Using the Reference option of the Scale command**

UNDO

The UNDO command displays the following options at the command prompt:
`Auto/Control/BEgin/End/Mark/Back<Number>:`

Auto Auto requires an additional specification of On or Off. When Auto is on, any operation taken from the menu, no matter how complicated, is treated as a single command, reversible by a single U command.

Back Back returns the drawing back to the state it was in when the last mark was entered. If you don't enter a mark, the drawing goes back to the beginning of the editing session. AutoCAD automatically places marks when some commands are used, but normally you must place them yourself. You can place multiple marks and UNDO to each.

Control Control limits the UNDO operation or disables it completely. It offers the following options:

- All enables the full UNDO feature.
- None disables U and UNDO commands.
- One limits U and UNDO to a single operation.
- Combine groups zoom and pan together into a single operation.
- Layer combines all operations taken in the Layer dialog box at one time into a single operation.

BEgin and End BEgin and End cause a group of operations to be treated as a single operation for the purposes of U and UNDO. A group, once ended, is always treated as a single, indivisible operation.

This is useful when you're writing an AutoLISP program. Place an UNDO BEgin at the beginning of the program and an UNDO End at the end. If the user doesn't like what the program did, it only takes one U to undo it. Otherwise, a separate UNDO is required for each step executed by the AutoLISP program, which can require repeating UNDO dozens of times.

Mark Mark makes a special mark in the undo information. The Back option of UNDO returns to that mark. If you're about to go off on a design tangent, you may want to set a mark with UNDO so you can return to that point quickly if the tangent doesn't work out.

Number Number is the default option. You can specify the number of preceding operations to be undone by typing the value. UNDO 1 is the equivalent of the U command. When you use a different value, 20 for example, 20 operations are undone at once. Typing REDO restores all 20, which is an advantage. Using this option allows you to search back through the history for the current editing session for a specific point in the design process by typing in large values. Since REDO only restores one UNDO operation, this can allow you to quickly find a step. Save the drawing before using this technique. REDO must be used immediately after using UNDO. If you do anything else, including panning or zooming, you've lost the ability to REDO.

You can see a history by selecting the down arrow next to either button on the Quick Access Toolbar.

Figure 1.19 **Undo and Redo history**

LENGTHEN

This interesting command can be used to change the length of a line, polyline, or an arc. It's the way to quickly create an arc with a specific arc length. To do that, create the arc, and use the Total option of LENGTHEN to change the total length to whatever you want. Select closer to the end you want to lengthen. LENGTHEN's options include the following:

DElta DElta allows you to add a specific amount to the length of an existing line or arc, or to add a specific angle to the existing included angle of an arc.

Percent Percent allows a specific percentage increase in the length of a line or Pline or the included angle of an arc.

Total Total lets you set a new overall length for a line, a Pline, or an arc.

DYnamic DYnamic allows you to change the length of a line (but not a Pline) or the included angle of an arc by picking a new point and watching it change.

<Select object> <Select object> gives the existing length or angle of a line or arc.

Dimensions

Dimension Styles are discussed at length in Chapter 4 because they're critical to applying technical graphics standards correctly. Here I want to point out some aspects of the process of Dimensioning that people often get wrong, starting with a difference that almost nobody seems to understand—aligned versus rotated Dimensions.

Aligned vs. Rotated

In my experience, most users don't understand rotated Dimensions. All linear Dimensions are either aligned or rotated, but most are rotated. Vertical Dimensions are rotated to 90, horizontal Dimensions to 0. Because those angles are built in, users often overlook the possibility of rotating Dimensions to other angles and frequently attempt to use an aligned Dimension where a rotated one is needed. Figure 1.20 shows the problem with trying to use DIMALIGN.

To place a rotated linear Dimension between two points at any angle, use DIMLINEAR and select the Rotated option. You can specify a rotation angle directly or by picking any two points that are at the desired angle of rotation.

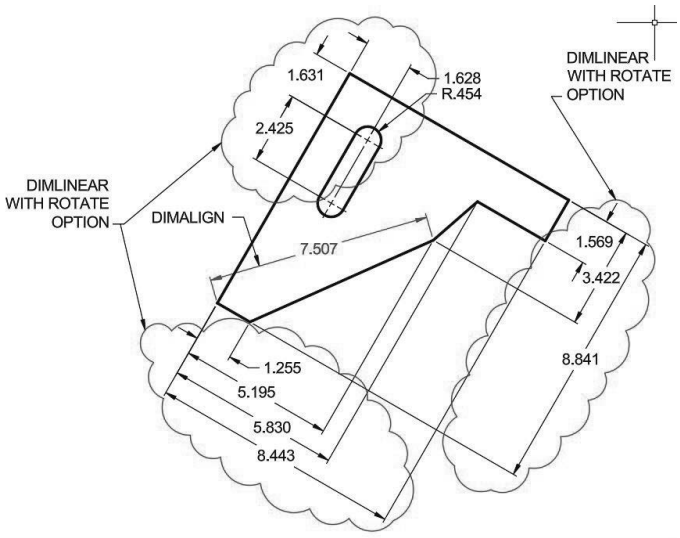


Figure 1.20 Rotated Dimensions versus aligned

ORDINATE

Ordinate Dimensions are used to set Dimensions as coordinates in the X and Y axes. Ordinate Dimensioning is used most often in mechanical drawing, but when I draw a structure that I'll be building myself, I always Dimension it with ordinate Dimensions rather than the traditional continuous method. That way, I can hook a tape to one end of a sole or top plate and mark all locations of openings from a fixed point rather than chaining them together, from opening to interior wall to opening. It is quicker and less likely to accumulate tolerance errors when you lay out walls.

Although there's a DIMORD command, the quickest way to place ordinate Dimensions is with QDIM. Select the points you wish to Dimension, and then select a new datumPoint for 0,0. Now you can choose the Ordinate option, select your objects, and place all ordinate Dimensions with one pick. If you have objects with different base points, use the datumPoint option again.

> To facilitate the use of ordinate Dimensions, you can place points using the POINT command at every location you wish to Dimension. Using either the FILTER or QSELECT command, you can now select only points. You're less likely to inadvertently select too many objects.

If you don't use QDIM, you must first establish a new base point by setting a UCS with its origin at the 0,0 location on the part you're Dimensioning. Use the UCS command, select the Origin option, and then select the 0,0 point on the part. If UCSICON is ON you can select it and move it to the new origin with the grip. AutoCAD automatically places the correct distance in the X or Y direction, depending on which direction you move from the origin point you pick.

If DIMASSOC is 1, all Dimension values are associated with defpoints, not the objects themselves. This association is maintained for the UCS that was current when the Dimension was placed. Including the defpoints in a selection set for the STRETCH command will update the values for the ordinate Dimensions, even if a new UCS has been set since the Dimension was created.

Dimension Variables

AutoCAD 2026 includes 86 Dimension variables. The easiest way to change them is to use the Dimension Style Manager dialog box. You don't have to know all of them, but there are two you should understand.

DIMASSOC

This variable was added to AutoCAD 2002 to control associative Dimensions. It replaces DIMASO, but not completely, because earlier drawings can be opened in current releases. When an earlier drawing is opened, the setting for DIMASO is used to determine the value of DIMASSOC, and that value will be 1, which can cause a problem if you place Dimensions in Paper Space, which I do not recommend.

DIMASSOC has three settings:

0—Dimensions are exploded when placed, which is a very bad idea. Because this was common practice in the 1980s, old, archived drawings may have this setting.

1—Dimensions have the traditional defpoint association, but they're not associated with objects and aren't accurate when a Dimension is placed in Paper Space.

2—Dimensions move with objects and reflect true size when placed in Paper Space.

DIMSCALE

This variable controls all size Dimension variables, such as DIMTXT, DIMASZ, DIMEXE, DIMEXO, DIMTVP, DIMTSZ, and DIMGAP. In the past, for Model Space plotting, DIMSCALE was set to the reciprocal of the plot scale to be used. If plotting will reduce a drawing 48 times to fit on a sheet of paper, you must first increase the size of the Dimensions 48 times, so they'll be readable. You may find this technique was used in older archived drawings.

I highly recommend placing all Dimensions in Model Space and plotting from Paper Space. Leave DIMSCALE set to its default value of 1, make your Dimensions annotative, and use defined annotative scales in Paper Space Viewports. Select Annotative in the Dimension Style Manager dialog box on the Fit tab for each parent and child variation of Dimension Styles. See Chapter 6 for detailed information on plotting.

Some offices still use the once standard practice of setting DIMSCALE to a value that reflects the reciprocal of the plot scale. This is a legacy of the old days of AutoCAD when such practice was required. If you choose to replace this practice to use annotative entities, you should do so in the context of a complete overhaul of office standards. Chapter 7 on plotting, and Chapter 6 on annotative objects provide suggestions for doing that.

Settings Commands

This section discusses some of the system variables (known as "setvars") in AutoCAD, but certainly not all of them. What follows are specific recommendations for the settings that people sometimes overlook.

System variables control the appearance or behavior of AutoCAD. You normally set these by making a selection in an appropriate dialog box, but they can also be set from the command line. Old-time users probably do this a lot by typing the variable name quickly. If I want to change my linear unit precision to check something, I type **LUPREC** and then type **8**. When I have the information I want, I undo that action or type **LUPREC** again.

Not everyone can touch-type; however, I appreciate being able to. (This may be a good time to thank my mother for insisting that I take high school typing in 1967, an era when the manual typewriters used in classrooms had blank keys so you wouldn't hunt and peck.)

The OPTIONS command (see Chapter 2, "Managing Your System") allows you to set all drawing aids, file search paths, the appearance of the screen, and the variables affecting saving and opening drawings, and also lets you save these settings in a user profile. The profiles can be exported for use on other computers.

There has been pressure over the years to eliminate the AutoCAD command line because it's seen by some people as outdated. I've always found this suggestion alarming, because a keyboard jockey can be much faster and more efficient than a toolbar user with comparable skills. For us keyboard users, Autodesk's decision to make the command line one more optional aspect of the interface was a welcome response to this pressure. I recommend docking it at the top or bottom of your screen and turning off dynamic input, but the interface is something you can customize to your liking without affecting the drawing itself.

APERTURE

The APERTURE variable allows you to control how close you must be to an object, measured in pixels, when using an object snap. It can also be set using the Drafting tab of the OPTIONS dialog box. If you're having trouble isolating an object in a cluttered area, try setting this value to a smaller size—I prefer a setting of 3. If you're having trouble picking an object because you have to get too close to it, try a larger size.

PICKBOX is similar, but it controls the size of the box that shows up when you're asked to select an object for editing. You can also control it visually by using the Drafting tab of the OPTIONS dialog box. I like a setting of 3 for this one, as well.

LTSCALE (Line Type Scale)

In the past, it was common to set this variable to the reciprocal of the intended plot scale. If you use Layouts properly, however, you shouldn't use LTSCALE to control the appearance of Linetypes, because it's global and affects all lines in a drawing. Instead, set LTSCALE, CELTSCALE, MSLTSCALE and PSLTSCALE all to 1. If necessary, you can change the Linetype scale for individual entities with the Properties dialog box. With PSLTSCALE set to 1, your Linetypes scale automatically in Paper Space Viewports, so they plot the same regardless of the annotative scale used. If MSLTSCALE is set to 1, the appearance in the Model Space tab will also reflect the Model Space annotative scale.

Some people don't like drawing in the Model Space tab unless their linetypes display correctly. Traditionally, if you put entities representing each linetype on a different Layer and use different colors for those Layers, you could easily identify the type of line it represents without seeing the segments. I like seeing them, so I do set an annotative scale after setting up a Layout, and make sure that MSLTSCALE is set to 1. You can display only one annotative scale in Model Space, unlike Paper Space where each Viewport can have its own scale.

If some of your entities have Linetypes that aren't scaled to your liking, instead of changing that with LTSCALE, use PROPERTIES to make changes to each individual object in a Viewport in the Layout.

Linetype problems are often the result of confusion between metric and imperial units. If you need an LTSCALE of 25 or so to see linetypes properly in a Layout, you've probably started your drawing in imperial units but are drawing objects measured in millimeters. If you need a small setting, say .04, then it's likely you have the opposite problem: You started a drawing in metric units but are drawing in inches. See section on "Control Imperial vs. Metric Units."

Layers

There are many useful tools for dealing with Layers. Here are a few of them:

- Locking allows you to lock a Layer so that it's visible but not changeable. You can still use Osnaps on the visible entities. Most users would benefit from using Layer locking more often to avoid inadvertent changes to their drawings.

People sometimes confuse Layer locking with Viewport locking in a Layout. They do different things and aren't related. Locking a Layer doesn't affect the display of objects in any way.

- Using Layer filters and freezing Layers by Viewport are useful advanced techniques in the Layer Properties Manager dialog box. You can control Layer properties at the command line by typing **-LAYER** or **VPLAYER**.
- You can change Layer names in the Layer Properties Manager dialog box, but the RENAME command is faster when you're changing names of Layers from a bound XREF, because you can use wildcards. Layer 0 is the only Layer you can't rename.
- Plot/No Plot settings can be made for Layers, so there's no longer a need to use the Defpoints Layer for that purpose. Don't make Defpoints the active Layer, because any objects you add to it won't plot.

I've seen two cases of odd behavior with objects placed on the Defpoints Layer: One user was unable to select a Viewport even though it was visible. At one time it was common to place entities on the Defpoints Layer so they wouldn't plot, but that is unnecessary now that you can make any Layer a non-plot Layer. Freezing Layer 0 created his problem. When he thawed Layer 0, he was able to select the Viewport.

Another user wanted a different display style for points on the Defpoints Layer, but points can't be displayed as anything other than a single pixel on the Defpoints Layer.

- Lineweight should be set to BYLAYER for almost all entities. If you leave the default value of .25mm for most Layers, set the line weight for object lines at .5mm, although .20mm and .40mm for thin and thick lines usually give a good line weight distinction when plotted.
- Linetypes and color should also be set to BYLAYER for almost all entities to make it easier to manage Layers and visually distinguish among entities on different Layers.

Display Commands

I'm sure you frequently use the All, Extents, and Window options of the ZOOM command, but have you ever saved the results with a name so you can return to it quickly? If you must connect two small features that are widely separated, have you ever used VPORTS to place two views in the Model Space tab and drawn from one to the other?
Read on.

VIEW

It surprises me how few people use this feature. It allows you to save a specific view on the screen, or as defined by a window, so that you can immediately call it up later. This approach reduces the number of regenerations necessary. It's particularly useful in 3D drawings, and some people use this feature for plotting a specific view. If you haven't ever tried saving named views, try it. You'll be surprised at how useful it can be, particularly when using AutoCAD sheet sets.

VPORTS (in Model Space)

This function is essential in 3D drawing and often useful in 2D drawing as well. It allows you to divide your screen in Model Space into multiple views, all tiled neatly together.

You can draw across Viewports in Model Space, so it's possible to connect between areas of a large drawing by placing Viewports close to each other and zooming in close to those areas in each Viewport. You can save the result, but don't confuse saving Viewports with saving views. You can restore saved views within any Model Space Viewport and then save the Viewport configuration. Now, whenever the Viewport configuration is restored, it will contain the views as they appeared when the Viewport was saved.

The Middle Scroll Wheel

Roll the middle wheel away from you to zoom in or toward you to zoom out. The increment is controlled by the variable ZOOMFACTOR—the larger the value, the bigger the jump when using the wheel. If you hold down the scroll wheel, you can pan. If you hold down the wheel while holding the Ctrl key, you get a joystick pan—not that useful, but kind of fun. If you also use other software, like SolidWorks where the wheel zooms in the opposite direction, you can change that in AutoCAD with the ZOOMWHEEL variable or the 3D Modeling tab of the OPTIONS dialog box.

Using the middle wheel to pan is great for general viewing, but if you want to pan with much more control, use the command line version, -PAN. You select a basepoint and then you can enter absolute, relative, or polar coordinates, or the tracking tools. This is particularly useful in Paper Space where you want to pan in one Viewport while maintaining its alignment with another Viewport. See Fig 1.1 in this chapter.

VTENABLE (View Transition Enable)

VTENABLE can be set to smoothly change magnification or location when zooming or panning. This replaced the abrupt transitions of prior releases. Some people love it, and some people don't. VTENABLE allows you to turn it off for everything, for panning only, for zooming only, for scripts only, or for some combination. There are 8 settings. I find a setting of 7 useful when working with a drawing that contains raster-based images like OLE or XREF objects. It can also be set in the Display tab of OPTIONS.

Utilities

Utilities allow you to do things like rename files and purge unused Layers. There are just a few things here, but one that could be a huge timesaver is the use of wildcards in the RENAME command, particularly when you bind an XREF.

PURGE

The PURGE command is used to selectively remove unused Layers, Blocks, styles, Linetypes, and dimstyles. You can do this by name or using the Purge All option (ALL at the command line). If your drawing has nested Block definitions (Blocks used to create other Blocks), you can select Purge Nested Items. The Purge dialog box also allows you to select individual entities individually or in groups. To select a consecutive list of entities, select the beginning item, hold down the Shift key, and select the last item in the group. The Ctrl key lets you add or remove items from the selection set. (See Figure 1.21.)

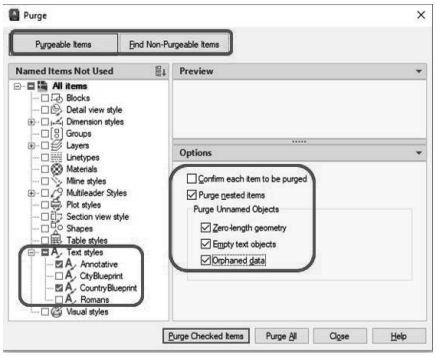


Figure 1.21 **Purge Dialog Box**

A line can be created with the same starting and ending point, which makes it hard to see, but it still has two endpoints, which can be selected if the ENDPOINT Osnap is active. An empty text object results from creating MTEXT and then later deleting the contents, but not the object. Orphaned data can be any number of things with a broken link. All should be purged.

Many people still use the WBLOCK command as a quick and complete PURGE command. That requires you to use the Entire Drawing option in the dialog box for WBLOCK, or the * option at the command line.

It is a good idea when you have finished a drawing to clean up any Layers, styles, or blocks that you don't need with PURGE (or WBLOCK) and OVERKILL. That may also be a good way to begin working with a drawing from another office before you start working with it.

RENAME

This is a great way to rename Blocks, Dimension Styles, Layers, Linetypes, materials, table styles, Text styles, UCSs, views, and Viewports. You can use wildcards, allowing you to quickly change all names created when binding external references (if you used the Bind option for binding, which I recommend you do, and not the Insert option). For example, if you bind an externally referenced drawing named XrefGuest to a host drawing, the Block definitions have names like refGuest0BIGTUB, XrefGuest0BIGBORDER, XrefGuest0CSIZE, STP123$0$D. If you want them to have the names BIGTUB and so on, type **XrefGuest0*** in the Old Name field and * in the Rename To field, as shown in Figure 1.22. Select the Rename button to update all.

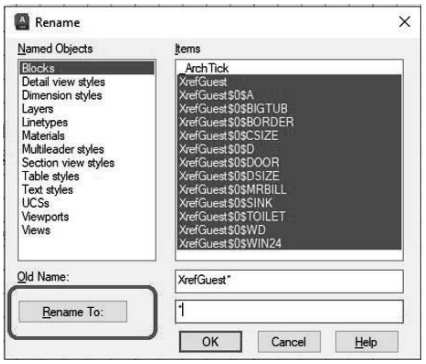


Figure 1.22 Using wildcards when renaming objects

PARTIALOPEN

You can choose to open part of a drawing to reduce regeneration time on large drawings. Highlight a file and then select the Open button in the lower-right corner of the Select File dialog box that's used to open an existing drawing. The Partial Open dialog box that's displayed can be used to select only those Layers or views you need. Once you've used PARTIALOPEN to open part of a drawing, you can use the PARTIALLOAD command at the command line or the File pull-down menu to select additional Layers. Note that you will have to turn on the display of the pull-down menu by setting MENUBAR to 1.

Express Tools

The Express Tools have always been popular with AutoCAD users because there are some fantastic timesavers here. Many native AutoCAD commands started as Express Tools, particularly many of the Layer options, while others are still external programs that work like native commands. You can find the external commands under the Express Tools tab in the pull-down menu or the ribbon. The Express Tools Reference listing all the external tools can be found by searching in the AutoCAD Help window.

If you don't have the Express Tools loaded on your workstation, try typing the command **EXPRESSTOOLS** at the command line. When AutoCAD is installed, they are placed here: C:\Program Files\Autodesk\AutoCAD 20xx\Express. Most of them are written in AutoLISP, with some being converted to .FAS files. Some are .ARX files. All can be loaded manually with the APPLOAD command in AutoCAD if you can't load them from the menus.

Note that AutoCAD contains a disclaimer for all 86 express tools. They are provided as a courtesy but are not supported. As a result, some offices reportedly delete them to avoid any problems or potential malicious code, so you may not have access to them, even if you load the menu using **EXPRESSTOOLS.** There is a discussion forum where you can post questions about them. https://forums.autodesk.com/

CHSPACE

CHSPACE evolved from an Express Tool into a native command. It lets you move objects between Paper Space and Model Space while retaining their relative scales. If you ever decide you placed something in the wrong space, you need this command. However, it doesn't work with annotative objects, which makes it less useful for Dimensions.

LAYWALK This is another tool that has become a native command along with many others. It allows you to step through the objects on each Layer to give you a visual clue as to what you've placed where. It's valuable when you're dealing with someone else's disaster drawing. The objects on each Layer are isolated temporarily while you cycle through the Layers. You can find them by expanding the Layer tools on the ribbon (Figure 1.23).

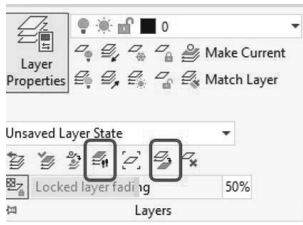


Figure 1.23 **Layer Walk and Layer Merge**

LAYMERGE This is also now a native command. I often advise new users and companies to use as many Layers as they may need so they'll have control over the related elements of a drawing. Sometimes this results in too many Layers, which is always easier to fix than too few.

TXT2MTXT is used to combine individual lines of text into a single MTEXT object. It doesn't format the result as individual lines, so you almost always need to do some editing. But if you want to group text from older drawings into a single object, it's nice.

FLATTEN I've recommended this command to a lot of people in companies that do civil design. They call me because they're having trouble with a drawing from someone else. Object snapping to endpoints produces odd results because, it turns out, the elements aren't all at the same elevation. FLATTEN quickly places every object at an elevation of zero. It isn't flawless, and you might find a more consistent version online, but it almost always does the job.

DIMEX and DIMIN These are related commands that allow you to export Dimension Styles and then import them into a different drawing. You can do this through AutoCAD DesignCenter (ADC) or by inserting a drawing that contains the Dimension style, but these Express Tools make it a little easier.

DIMREASSOC This is a nice tool. It permits you to individually or globally update Dimensions that have been overridden by the operator. In other words, the value in the Dimension and the actual size of the object don't match. I have overridden a Dimension purposely on rare occasions when I had an imminent deadline and had to plot something out immediately, but most of the time I use this to find non-associative Dimensions in drawings created by others. With DIMREASSOC I can find and fix the geometry.

You can also change the overwritten Dimensions automatically, which may cause even more problems if you don't notice what got updated. The problem is that the value is probably overridden because the geometry is wrong.

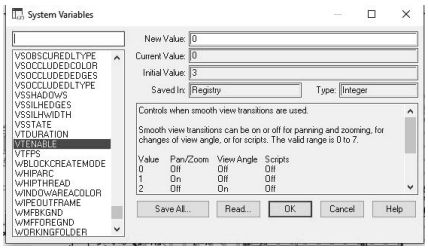


Figure 1.24 **System Variables**

SYSVDLG The System Variables editor is a wonderful tool. It gives you a concise list of each system variable, tells you where its value is stored, and describes what it does. Best of all, you can save the settings to an SVF file and restore them if you ever have to. If you do any training using computers that aren't your own, take this file with you; it will reduce the number of surprises you may encounter. (See Figure 1.24.)

Figure 1.25 **Overkill Dialog Box**

OVERKILL doesn't always work correctly. I've found that it works best if you use it for one kind of cleanup at a time (Pline, overlaps, co-linear) even though the dialog box allows you to check every type at once. Run it for each type separately to get the results you want. Note that it has a tolerance that requires that overlapping and co-linear elements be within a millionth of a unit. You can adjust that if you want.

BURST will explode a Block and turn any attribute values into text.

BCOUNT will list all Block insertions and indicate how many there are of each.

ARCTEXT will place text along an arc.

TCASE will change the case of selected text.

SUPERHATCH will create a custom Hatch from images or Blocks.

There are many more that might be useful to you. Take a look when you have a break and see what's there. It might save you a bucketload of time.

CHAPTER 2

Managing Your System

I'm a teacher and trainer, and my experience with AutoCAD users tells me you should know some things about your computer system that affect how AutoCAD functions. I often find that even experienced AutoCAD users run into confusing situations when they try to use backup or Autosave files. That's why I'm focusing this chapter on managing the system you use to work with AutoCAD. In this chapter, I'll take you through the Options dialog box and discuss the relationship between profiles and workspaces.

I'll also take you on a brief trip into the past, with a discussion of how knowing some old-fashioned DOS can save you a huge amount of time.

I want to encourage you to preserve knowledge that's fast fading away; fewer and fewer AutoCAD users remember (if they ever knew) how powerful these functions are.

I'll conclude this chapter with a discussion of the ACAD.PGP file: a humble little thing that still has great utility and provides the only hook to certain operating system features from the AutoLISP processor.

- Managing Files
- Managing AutoCAD
- Directories
- Why DOS Isn't Dead Yet
- External Commands