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Things Every AutoCAD® User Should Know - Expanded!

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GD41-2 There are fundamental things about the computer's operating system, the principles of technical graphics, and the software itself that everyone who uses AutoCAD should know. This workshop will fill in some gaps in your knowledge, provide useful techniques for using AutoCAD efficiently, and introduce you to customization techniques that can make you far more productive.

About the Speaker:

Dan has been involved with technical education for more than 20 years, the past 16 of which have been as a member of the Architectural and Engineering Design Department at Southern Maine Technical College. Dan trains industry professionals in AutoCAD use and management, AutoLISP, and geometric dimensioning and tolerancing. He works with Knowledge Development Solutions to help create technical exams for Autodesk. Dan earned M.S. and B.S. degrees in Industrial Technology from the University of Southern Maine, and a B.A. degree in Psychology from Swarthmore College.

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THINGS EVERY AUTOCAD® USER SHOULD KNOW

BACKGROUND

This workshop is for two groups: new users of AutoCAD® who want to produce good work that meets generally accepted standards; and experienced users who suspect they have some fundamental gaps in their knowledge. It is certainly not going to teach you *everything* you should know about applying this very useful tool to the process of designing and documenting work done in a wide variety of fields. It may, however, help you to avoid common problems and produce work that is more consistent, reliable and accurate.

This workshop should probably be titled "Things *I Think* Every AutoCAD® User Should Know," since there is no one set of rules for using AutoCAD® that is followed by everyone, and you will find people who disagree with some of the opinions expressed here.

A word about releases: most of this material applies to any release of AutoCAD®. However, there are a few places where a distinction must be made between release 2006 and earlier releases. I will make those differences clear where they occur.

There are four general areas that this workshop will:

- Software management
- Technical Graphics Standards
- General Design Applications
- AutoCAD® Standards

SOFTWARE MANAGEMENT

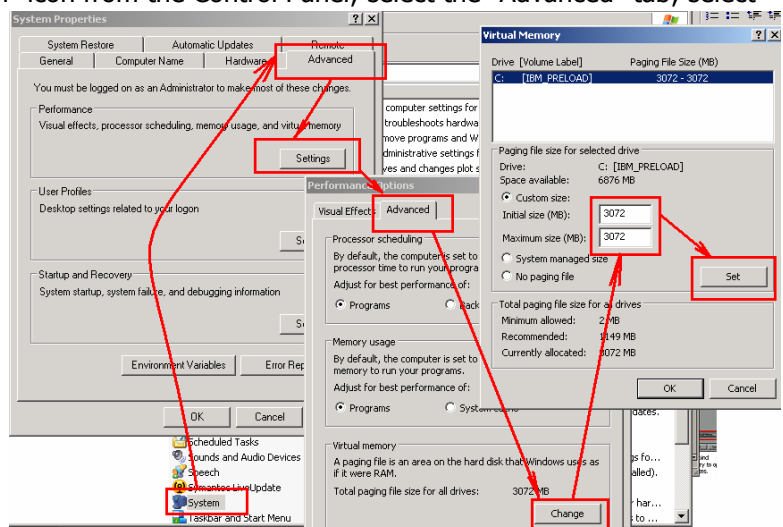
1. Ram, processor speed, and swap file size.

In many ways RAM is more important than processor speed. More RAM is good, and even more RAM is better. Since RAM is cheap, you should certainly have at least 512 Meg, and one or more Gig wouldn't be out of the question. Swap file size is also important for running AutoCAD®, but the default settings in Windows are too low. Set swap file size to at least 4 times the amount of RAM you have. If you have 512, set your swap size to 2048 (OK, it can be 2000...).

Swap file space is available on the hard drive for temporary placement of information when the computer runs out of RAM (hence the name "Virtual Memory"). AutoCAD® uses a lot of RAM, so it often stores information in swap files.

To increase Virtual Memory, select the "System" icon from the Control Panel; select the "Advanced" tab; select the "Settings" button for "Performance"; select the "Advanced" tab of this dialog box; select the "Change" button; specify a drive and size for your swap file. Make the maximum and minimum the same and the virtual memory will be contiguous. Do a scandisk and defragment your drives before doing this to get the maximum space possible.

2. Managing and copying files.



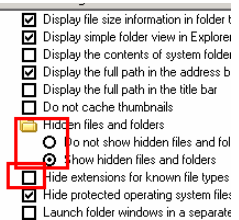
AutoCAD® uses many different file types. Unless you do programming or customizing, most of them don't concern you. However, when you find yourself in a bind because you have lost a drawing, AutoCAD® has "stopped responding," the computer crashed, or you try to open a corrupted file, you should know how to use the backup files and the autosave files that AutoCAD® creates.

In AutoCAD® 2006, there is a "Drawing Recovery" dialog box that will usually pop up the first time that you start AutoCAD® after a system crash (or other unusual termination). If you miss that opportunity use the DRAWINGRECOVERY command, but if you have set RECOVERYMODE to 0 or 1, or if you are using a previous release of AutoCAD®, you will have to find these files yourself.

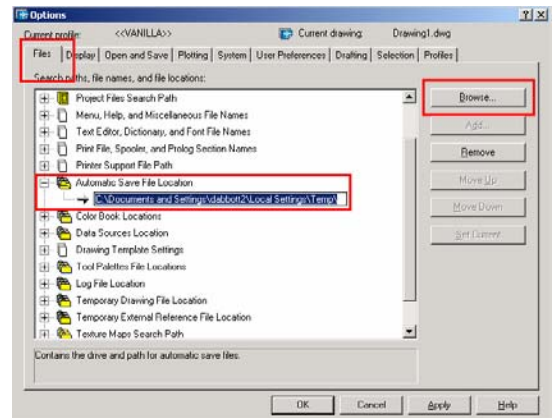
Backup files. When you save a drawing, any existing file of the same name is renamed by changing the *.dwg* extension to *.bak* before your drawing is saved. It will be in the same folder as your *.dwg* file. To use a backup file, rename the extension to *.dwg* and open it. Although some data may be missing, it is better than nothing. By the way, it is possible to turn off the backup feature off. That saves hard disk space at some risk of data loss.

Autosave files. The file created by the AUTOSAVE function does NOT have the same name, extension or path as the drawing itself. The name given to the file created by AUTOSAVE starts with the actual drawing name and contains incrementing characters. The extension is *.sv\$*. Like *.bak* files, the extension must be changed to *.dwg* before it can be opened as a drawing.

By default AutoSAVE files are in "C:\Documents and Settings\[user name]\local settings\temp." This is a hidden folder, so to find an autosave file you must make hidden folders visible. From Windows Explorer, select the "Tools" pull down; then "Folder Options;" then the "View" tab. Make the selections shown.



I recommend that you change the location of autosave files to a custom folder, using the "Files" tab of the "Options" dialog box. I also recommend that you set SAVETIME to an increment of 20 minutes rather than the default two hours. SAVETIME is saved in the system registry, not in the drawing. The *.sv\$* file is only available if AutoCAD® terminates abnormally. Otherwise, it is deleted upon closing AutoCAD®.

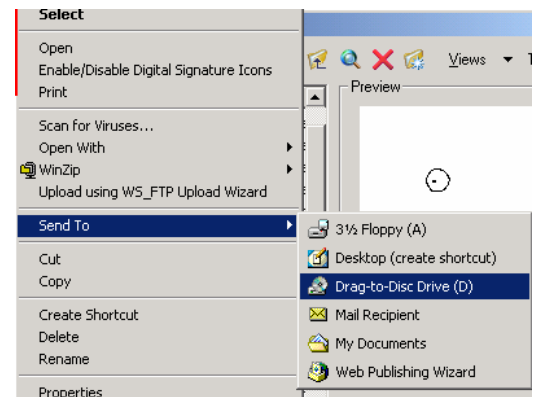


3. Working with files on a floppy disk or CDR.

Don't open *.dwg* files from a floppy or CDR. Don't use SAVEAS to save a drawing directly to a floppy or CDR. To use a file from a removable disk, copy it to the hard drive, and open it from there. Save files to the hard drive, and then copy them to the removable disk. Do this by right-clicking on the file name in any file management dialog box, and selecting the "Send to:" option. Otherwise the removable disk can fill up with *.bak* files, and the computer will do frequent searches on it while you are working. Files copied from a CDR may be "read-only." To change that, right-click on the file name, select "properties," then clear the "read-only" attribute.

4. Archiving files.

When archiving drawings on servers or local hard drives that have a limited capacity, reduce the file size as much as possible by using the PURGE or the WBLOCK command.

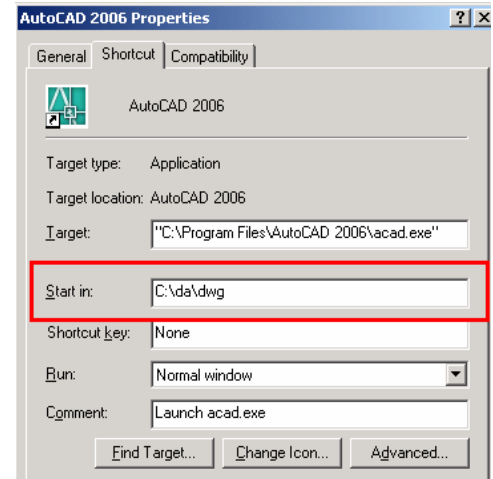


5. Set a default file-saving location.

Set the path for starting up AutoCAD® in the properties window for the AutoCAD® desktop icon. Display the properties window by right-clicking on the icon and selecting "properties" at the bottom of the list. Now you will go directly to the folder you specify (in this case "C:\da\dwg") when you save your drawings. Note that a profile can also specified on startup in the "Target" window.

6. Creating and using template files.

You can save a lot of work if you create and use AutoCAD® template files. To create a template file, open a drawing that is set up the way most of your drawings should be set up – with proper layer names, dimension styles, text styles, variable settings, layouts, and block definitions. Save it, using the SAVEAS command, by selecting *AutoCAD® Drawing Template File (.dwt)* from the *Files of type:* window. Give it a name that is logical and save it in either the default location given ("C:\Documents and Settings\UserName\Local Settings\Application Data\Autodesk\AutoCAD® 2005\R16.1\enu\Template," believe it or not, yet *another* hidden folder) or a custom folder. Now erase all the objects in the drawing and save it again the same way. I would recommend creating a template folder of your own. Use the "FILE" tab of OPTIONS to set the path for template files to your folder.



7. Managing the interface of AutoCAD®.

Options and Profiles

I won't try to go over all of the settings available in the Options dialog box, but you can customize the interface by changing things like the background color, the behavior of right-click menus, the locations of default files, etc. You should use the Profiles tab to save the resulting profile with a name, and export it to an ".arg" file so you can use it on another computer, or restore it to your current computer if your settings are changed.

Things to consider changing:

Files Tab - use the files tab to add a "custom" folder to the default path AutoCAD® will use to search for files. This will give you someplace to put your customized files for your own, or office-wide use over the network. Move it to the top so that your customization gets used.

You can also set the location for the Template files to your own folder or a folder on a network, have the Autosave files stored in a different location, and point to a different location for plotters or pen style tables.

Display Tab - hide scroll bars to give yourself more screen space and prevent the sudden shift in the screen display that sometimes happens when you try to make a selection near the edge of your screen. You can also change your background color, which I highly recommend if you are doing a screen capture or using the WMFOUT command to create a raster image, so that you will get a white background. Circle smoothness should be set to at least 2000, and the size of the crosshair can be increased up to 100% (just like R14!)

Open and Save Tab – set the default file format to R2000 if you exchange drawings with other companies. The round-tripping feature of 2005 will preserve those elements that are not in R2000.

I am often asked if there is some way to tell what release of AutoCAD® was used to save a .dwg file. It would be nice if it were part of the file properties displayed by windows, but it isn't. You can find out by opening the .dwg file in notepad. Only the first line is readable, but it contains the release in the following form: AC1018=R2006/2005/2004; AC1015=R2002/2000/2000i; AC1014=R14/14.01; AC1012=R13; AC1009=R12/11; AC1006=R10; AC1004=R9. Other codes were used for earlier, beta, and internal releases.

Setting AutoSave to 15 minutes will not only prevent you from losing too much work, but also give you a signal that it is time to look away from the keyboard and stretch your hands and wrists.

I would also increase the number of recently used files to list up to the maximum of 9. This is a poor substitution for the helpful history feature of the much-maligned "AutoCAD® Today," but since that baby went out with the bath water, it is all we have.

Plot and Publish – set a default output device and plot style table to those most commonly used. I also check off "Hide System Printers" to reduce the number of choices, and you can set plot stamp options here as well.

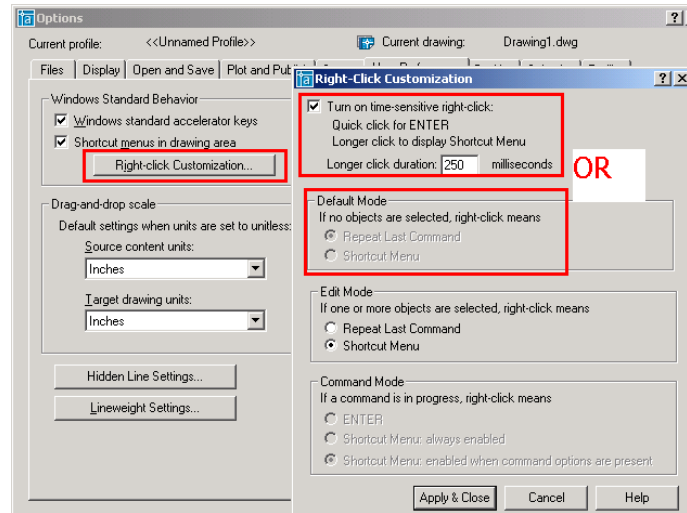
System – if you want the startup dialog box, this is where you can change the current behavior of AutoCAD®. Otherwise there is not much I would change here.

User Preferences -- if you want to be able to repeat the last command by right-clicking the mouse without having to make another menu pick, you have two choices. Try the time-sensitive setting to see if you like it (I do!) or select "right-click customization" and "repeat last command" for default mode. Otherwise, I would keep the rest of the right-click behavior to take advantage of the many right-click menus.

Drafting -- reduce the Aperture size to reduce the number of possible objects your cursor will select with running OSNAPS.

Selection -- make sure Noun/Verb is checked so you can change an object's layer easily using the layer window in the Object Properties toolbar. Reduce the pickbox size to make object selection more accurate. You might want to increase the number of objects for which grips are shown (default is 100) but I wouldn't. You might find that PICKADD, the variable that controls the Shift to Add option, mysteriously changes. If so, the culprit is probably the Properties Tool Pallet, which inexplicably has a toggle for that variable (see below).

Profiles – I like to save different profiles for different purposes. Once you have gone to the trouble of setting up the interface, you may as well save the result to an AutoCAD® Registry file (.arg).



8. Maintaining AutoCAD®.

Like any software, AutoCAD® 2006 has some bugs. There are several things you can do to maintain your installation of AutoCAD®, and find work-arounds for the bugs.

- Use the communications link in the lower right-hand corner of your screen to check for updates. If you have a live Internet link, you can set this feature to check daily for new updates, and then you will be notified. There is a service pack 1 for 2005.
- Find out about many bugs by reading Steve Johnson's "Bug Watch" column in Cadalyst. You can find it online at www.cadalyst.com.
- Keep backup copies of key customizable files, particularly the ACAD.cui (R2006), ACADDOC.lsp, and ACAD.mns (pre R2006). If your menus change unexpectedly, you can replace the appropriate files, or in pre-R2006 versions, reload the acad.mnu file using the MENU command. Change the "files of type" to .mnu. This will probably restore your missing menus to their original state, but you will lose any customizing you may have done.



TECHNICAL GRAPHICS STANDARDS

By "Technical Graphics Standards" I mean the rules of drafting that people used to know when they produced technical drawings by hand. Those standards were developed over many years to make technical communication more consistent and therefore more reliable. Since AutoCAD® does not automatically apply proper graphics standards, you must do so using appropriate techniques if you want your drawings to meet standards.

Standards

Although there is some variation among offices in producing drawings, every discipline has generally accepted universal practices. The most clearly defined standards are those in the mechanical design industry. Standards for drawings (and other engineering standards) are published by the American Society of Mechanical Engineers (ASME). These standards are voluntary, although they are often specified in design contracts between two or more firms, but widely used.

All other design disciplines in the U.S., including architectural, civil, surveying, electrical, electronic, piping, and welding, are based fundamentally on the same rules. If you use AutoCAD® to design or to document designs, I think you should know what those rules are.

Mechanical Dimensions

Open the Dimension Style Manager and create a new style with a name that makes sense to you. Select the tabs at the top one at a time. Modify *STANDARD*, after renaming it "Mechanical-Inch." Do NOT use the name *STANDARD*. It will cause problems later.

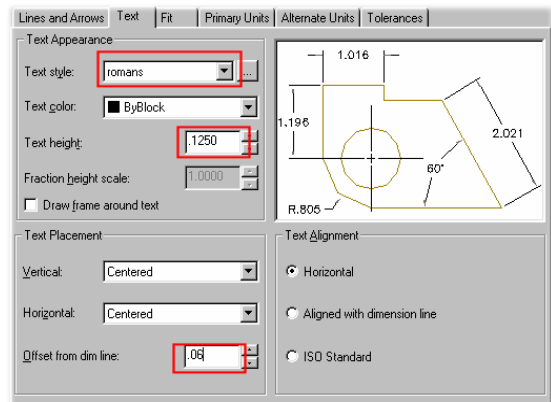
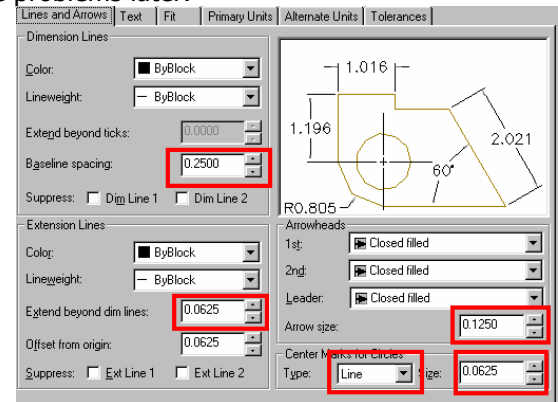
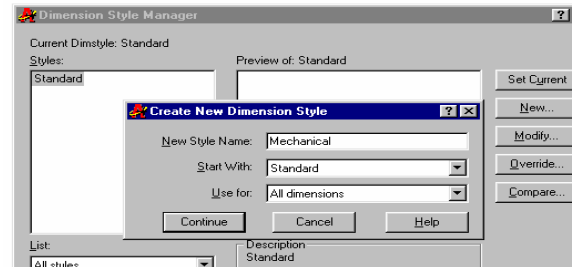
Lines and Arrows tab

Make the changes shown to baseline spacing, extension-line extension, arrow size and center marks for circles. Set it to "line" so that you can use the DIMCENTER (DCE) command to quickly place proper center marks. Set it to "none" when you create child variations for diameters and radii. *Note: there is a bug in R2002 and R2004 -- baseline spacing will not scale when dimensions are scaled to Paper Space. You must set a value based on intended plot scale if you plan to use baseline dimensions. This bug was fixed in R2005.*

Text tab

Set the text style to be used for dimensions. *DON'T USE "STANDARD."* It is based on the .txt font. (In fact never use "STANDARD" as the name for anything, even if you redefine the default version provided with AutoCAD®. You may have problems later trying to insert one drawing into another, or trying to bind a drawing to another one if used as an XREF.)

If you have not yet created a text style, select the button with the 3 dots (ellipses) next to the Text Style window. This will bring you to the Text Style dialog box. Create a new style based on the Romans.shx font, with a height of zero, and close the window. Now select your new style from the drop-down list. Change "Text height" to .125, and change "Offset from dimension line" to .04.



Fit tab

The Fit tab controls the manner in which AutoCAD® will fit the text and arrows for dimensions into the space available. You normally move the arrows outside of extension lines when there isn't room for both text and arrows, so check off "arrows" in the Fit Options. Select "scale dimensions to layout" so dimensions will be a consistent size when plotting.

Primary Units tab

Set the number of decimal places to what you need -- 3 for a general style. You can easily change precision for a particular dimension by selecting it and right-clicking. Suppress the leading zeros for decimal inch, but not for metric.

Alternate Units and Tolerances tabs

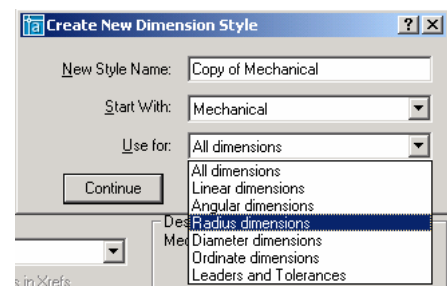
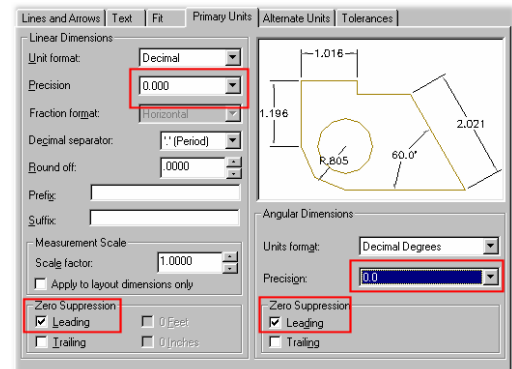
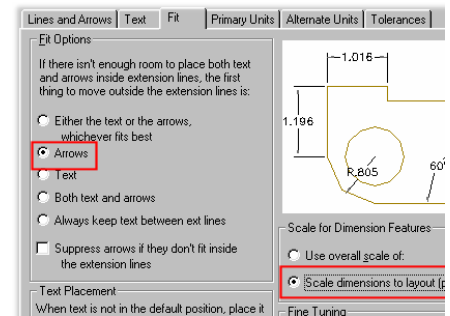
These tabs should not be modified unless you are using either alternate units or tolerances. The tolerance method I prefer is Limit Tolerance, but you might have a different preference. I leave the upper and lower tolerance values at 0, and change each individual dimension using the Properties Tool Palette, but if you have a drawing where the tolerances for a particular part are fairly uniform, you might want to set them here. You might want to consider using the excellent AutoCAD® Mechanical version if you do a lot of tolerancing.

Child variations

Pick the OK button at the bottom of the dialog box and return to the Dimension Style Manager. Pick the "New" button and then select Radial dimensions under the "Use for:" window. Press "Continue" and make the following changes for radial dimensions.

Lines and Arrows: set Center Marks to "none."

Fit: set Fit to "Both text and arrows," and set Fine Tuning to "Place text manually when dimensioning."



Now do the same thing for diameter dimensions. When you are done, you will have a new style with two child variations. They will work for most mechanical dimensions.

Metric dimensions - ASME

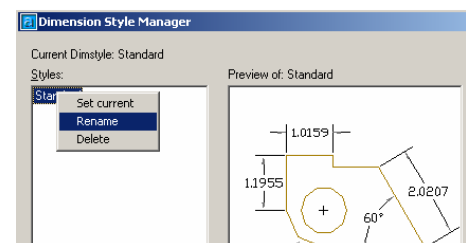
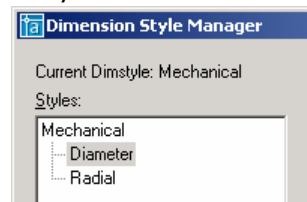
For metric drawings, create a "Mechanical-Metric" style as you did for "Mechanical-Inch" above but use the near-metric equivalent values as follows: for .0625 use 1.5; for .125 use 3; for .25 use 6. The only other change to make would be in the Primary units tab – DON'T suppress leading zeros for metric.

Metric dimensions – ISO

For the ISO system, start a metric drawing from scratch and use dimstyle "ISO-25," but change the text style by replacing *Standard* with one based on the *Romans.shx* font.

Architectural

Open the Dimension Style Manager. Rename "Standard" to "Architectural." Do NOT use the name *STANDARD*. That would cause problems later. Once the style has been renamed, you can select the "Modify" button and make the following



changes. If you already have a mechanical style created as indicted above, you can modify that style instead of the Standard style, which will save some time.

Lines and Arrows tab. Change "baseline spacing" to 1/2, "extend beyond dimension line" to 1/16, "center mark type" to lines, and center mark size to 1/16. Leave arrowheads set to closed filled" and set size to 1/8. You will set up a child style for linear dimensions later.

Note: there is a bug in R2002 and R2004 -- baseline spacing will not scale when dimensions are scaled to Paper Space. You must set a value based on intended plot scale for automatic baseline dimensions. This bug was fixed in R2005.

Text tab. Romans is an acceptable font for architectural drawings, but you may want to use the City Blueprint font. Avoid any font that doesn't come with the software if you ever send drawings to others. Change text height to 1/8, and offset to 1/16. Leave other settings as they are. Use a child linear style for dimensions that are to be aligned and centered. That setup appears later in this section.

Fit tab. Change the "Scale for dimension features" setting to "Scale dimensions to layout (paperspace)."

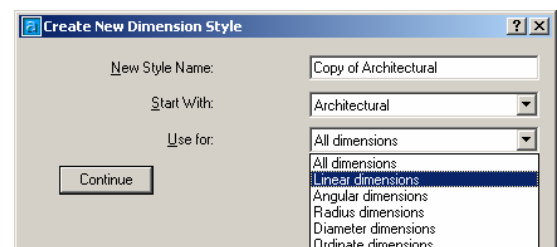
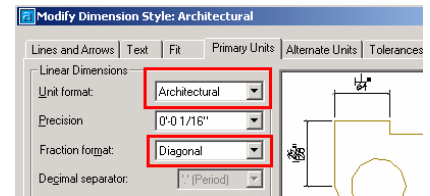
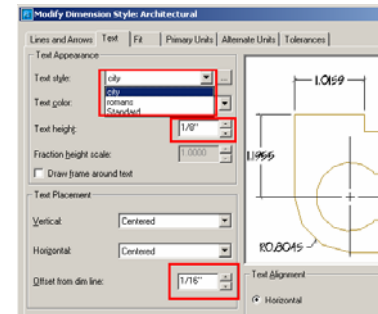
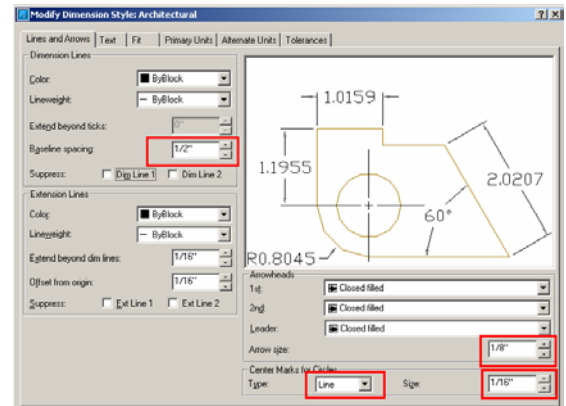
Primary Units tab. Change "units" to "Architectural." Note that the value you set with the UNITS dialog box will NOT be used for dimensions. You must set the units here. Set the precision to a small value. I use a setting of 1/256 so any drawing errors will show up when I add dimensions. This does NOT result in odd fractions on the drawing unless there is a mistake. I set fractions to "diagonal" and suppress 0 feet, but not 0 inches. There was a bug in R2002 that prevented any setting for "zero suppression" from being applied. To control suppression in 2002, change DIMZIN to 3 at the command line to display zero inches and suppress zero feet. Save the resulting override to the current dimension style. This bug was fixed in R2004.

Alternative and Tolerances tabs. Do not change any settings in these tabs.

Child variations. You will need to make three child variations for architectural dimensions: linear, radial, and diameter.

Linear child style. To create a child variation, highlight "Architectural" and then select "NEW" at the main dialog box. In the drop-down window labeled "Use for," select "Linear." Under the "Lines and Arrows" tab change the arrowhead type to "architectural tick" and the size of arrowheads to 1/16. Under the "Text" tab, change the "Text Placement" to "Above" and the "Text alignment" to "Aligned with dimension line."

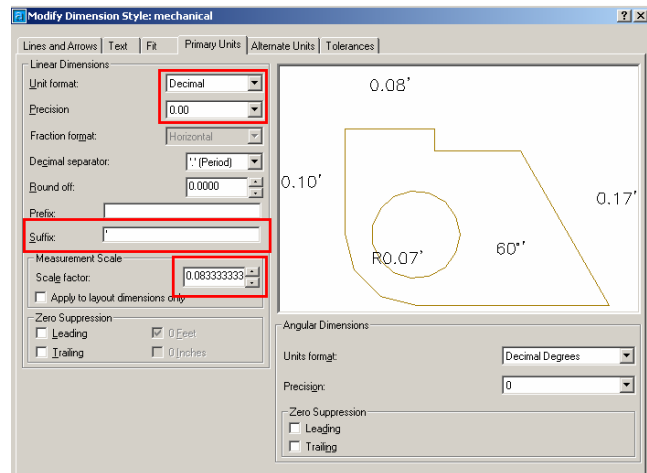
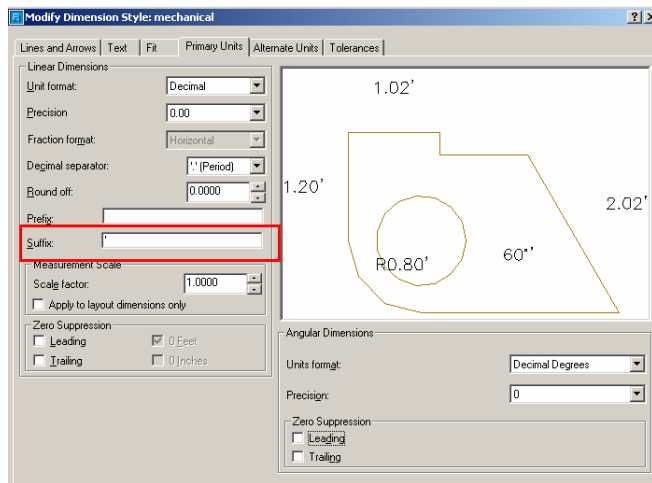
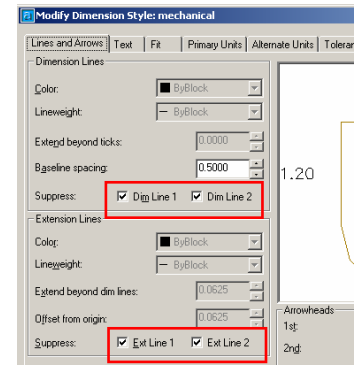
Radial and diameter dimensions. To set up child variations for radial and diameter dimensions, highlight "Architectural" and then select "NEW" at the main dialog box. In the drop-down window labeled "Use for," select "Radial." Under the "Lines and Arrows" tab, select "none" for "Center marks for circles." Under the "Fit" tab, check off "place text manually when dimensioning." Repeat these steps to create a child dimension style for diameter dimensions.



Civil/Surveying

To place dimensions indicating bearing and boundary length, create a style that is based on Mechanical with the following changes: suppress both extension lines and both dimension lines in the "Lines and Arrows" tab. Depending on whether the drawing was done in decimal feet (generally preferred) or Engineering or Architectural units, set "Primary Units" as follows:

For drawings done in decimal-feet, which is by far the most likely, use the settings shown on the left. For those done using Architectural or Engineering units, use the settings shown on the right, converting inches into feet by multiplying each dimension by 1/12 (0.0833333).



When placing dimensions for boundary lines, use the DIMALIGNED command (DAL) or tool button. Select the line or the ends of the boundary line, and then select the endpoint of the line again when prompted for dimension location. You will have a dimension in decimal feet over the boundary line.

3. Text Style

Don't use the TXT.shx font for anything. It does not meet any standard, although it has become a standard in some offices just because it has been the default AutoCAD® font for so long.

The only font shipped with AutoCAD® that meets ASME and ISO standards is ROMANS.shx (same font as SIMPLEX.shx). The one drawback to using ROMANS.shx is that there is no non-proportional version. If you need a non-proportional font, you have only MONOTXT.shx (same as TXT.shx) or a TrueType font named "Monospac821 BT" to choose from. There are fewer drawbacks from using a TrueType font than from using the very ugly .txt font, so use Monospac821 BT. For architectural or civil drawings, many offices use Country or City Blueprint. I wish that AutoCAD® came with an .shx architectural font, and a non-proportional version of all fonts, but it doesn't. Maybe next time.

It is permissible to use either vertical letters, or letters inclined at an angle up to 70 degrees. If you do use inclined letters, set an oblique angle of 10-20 degrees when creating your text style.

Letter height can vary, depending on what you are using the text for. I STRONGLY recommend setting a text height of 0 when creating a text style, so that your text can be placed at any height and automatically scaled for dimensions.

The AutoCAD® default text heights setting of .200 or .180 for English drawings is greater than the minimum required by ANSI Y14.2M for most text. The default setting of 2.5 for metric drawings is less than the minimum required. I recommend using the minimum heights specified in by ASME Y14.3M for all text, particularly

dimension text, since that allows you to place the most information in the drawing and still have it be readable.

Minimum text heights for mechanically produced text are as follows:

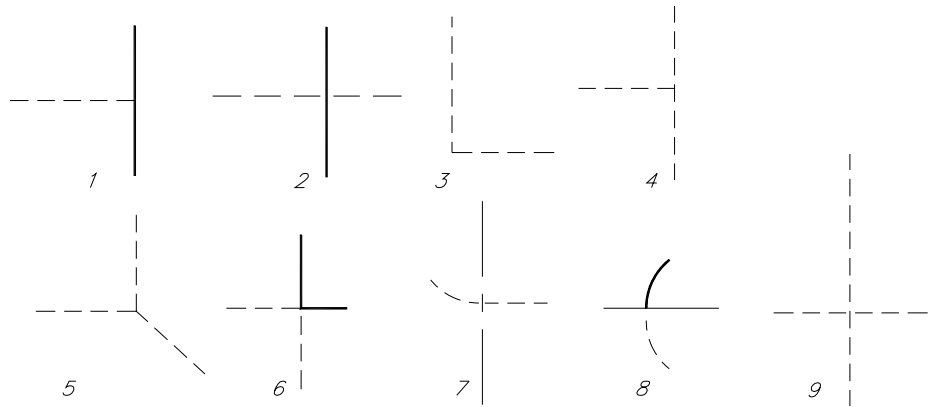
Use	Drawing Size	Metric	Inch
Drawing Number, Title and revision letter in Title Block	up through 22" x 17"	3	.12
	greater than 22" x 17"	6	.24
Section and View Letters	all	6	.24
Zone characters in border	all	6	.24
Drawing block headings	all	2.5	.10
Dimensions, tolerances, limits, notes, subtitles for views, tables, revisions, and zone characters in body of drawing.	all	3	.12

4. Hidden lines

Hidden lines have some special applications defined in the standards that are meant to convey specific information as illustrated below. These special applications can be difficult to accomplish automatically with AutoCAD®.

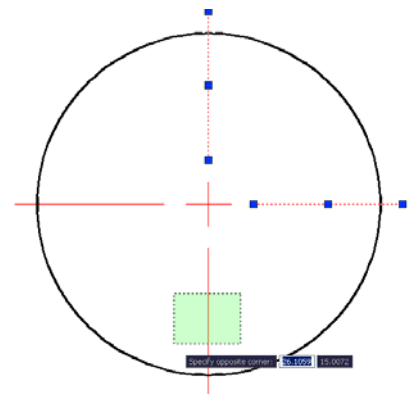
To get the results shown in 1, 3 and 5, you can normally just place lines on a layer with a hidden linetype. The results shown in the others take a little effort. You might have to move a line or change its linetype scale. I recommend using the Properties window to adjust the individual linetype of the lines after placing them. Some suggestions for linetypes:

- Adjust linetype scale for objects AFTER you have set up your views in Paper Space.
- Don't use LTSCALE to adjust linetypes – it is a global variable and will affect all objects.
- Set PLINEGEN to 1 so linetype breaks will be evenly distributed along PLINE, particularly if curved.

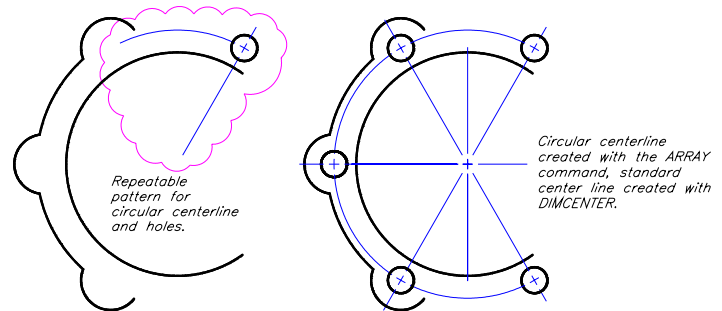


5. Centerlines

When centerlines are used to locate a center point, they should show a small cross, a break and extension lines (if necessary). To create proper centerlines, I recommend setting "line" as the value for centermarks in your dimension style with a value of -.0625 for inches and -1.5 for metric (a negative value for DIMCEN) and using the DIMCENTER (DCE) command to place crosshair centerlines on circles or arcs. This creates 6 separate lines that can be edited independently without changing the location of the crosshair.



Circular centerlines used for patterns like bolt holes should be created using the ARRAY command to be done correctly. The crosshairs on circular centerlines are actually arcs, not straight lines. To create a complete circular centerline for representing the locations of boltholes, draw an arc segment and straight-line crosshair for just one hole. Add an arc segment to each to the next hole and a radial center line and array the collection, as many times as there are boltholes. This is quicker than it sounds, and will always give you proper centerline results.



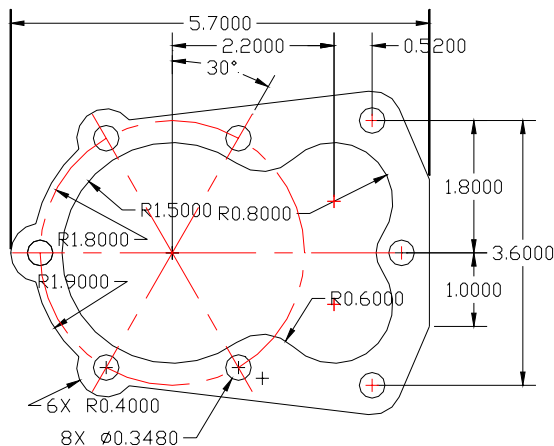
6. Lineweights

There are two standard lineweights – thick and thin. Use thick lines for visible edges, break lines, viewing plane lines, and cutting plane lines. Use thin lines for everything else. It is really surprising how much of a difference it can make to a plotted drawing when lineweights are used correctly. Set the lineweights BYLAYER using .4 or .5 for thick and .2 or .25 for thin. I like setting the default to .2 and using .4 for object lines. Architectural drawings often use more than two lineweights, by the way. Some offices use as many as six. (I'm not recommending it.)

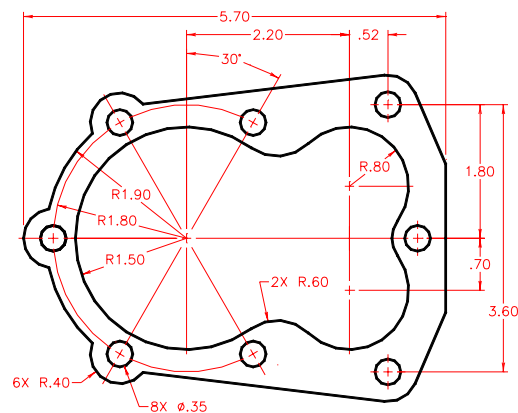
Applying Standards

Following standards results in drawings that are less likely to be misinterpreted. The illustration on the left is the result of using AutoCAD® defaults. The illustration on the right applies many of the practices described.

DESIGN STANDARDS



A few



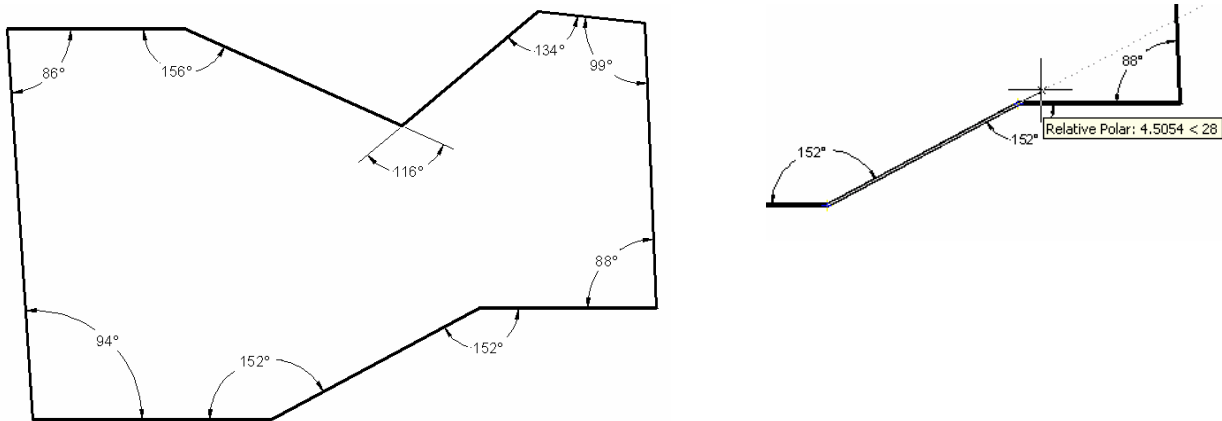
general practices for using AutoCAD® for designing.

1. **Draw actual size.** Even details can be drawn full size, if you use layouts. Use paper space viewports to scale them, NOT the SCALE command. Even a long conventional break can be used to represent objects drawn full size by using two viewports at the same scale, aligned and separated for the break symbol.
2. **Draw existing features "as built."** When designing for renovations or additions, creating geometry "as built" rather than "as designed" permits you to solve problems in the software instead of the field.
3. **Draw mechanical parts at MMC.** People often draw objects in the middle of their size tolerance range. I recommend using MMC (Maximum Material Condition) instead. This allows you to draw parts in the situation where they are most likely to fail, and check for interferences more readily. I recommend this approach for both 2D and 3D modeling, including when using Mechanical Desktop or Inventor.
4. **Use logical increments.** For architectural design, use whole inches, feet, two feet, or four feet when possible. For metric parts, use increments of 2 mm if possible.

AUTOCAD® STANDARDS

1. **Use blocks.** Blocks can dramatically reduce file size, allow you to quickly update large amounts of work, and make your drawings more consistent.
 2. **Don't explode blocks.** Especially true for dimensions. Once you explode a block, you increase the file size and lose the ability to update them. Dimensions will no longer update values when you modify geometry, and you cannot update dimension appearance with changes to Dimstyles.
 3. **WBLOCK creates a drawing file, NOT a block.** You don't need to use the WBLOCK command to create a drawing that you plan to insert into other drawings. You can just save the drawing. The WBLOCK command is very useful for saving part of a drawing, including a block definition.
 4. **Never override dimension values – redraw the geometry.** When you are adding dimensions to drawings, it is tempting to simply 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 is correct and then add an associative dimension. Otherwise, you and everyone else who ever uses your drawing will assume it is correct at some point in the future with potentially dire results. Using very small increments of precision will help flag drawing errors.
 5. **Use layers with logical and consistent names to separate geometry and 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 is essential to efficient management. If your layer names are logical enough, it is easy to manage multiple layers. Take the following example from a residential floor plan. Layers names all begin with a field that designates the floor of the residence: FL1, FL1-DIM, etc., FL2, FL2-DIM, etc. FND, FND-DIM, etc. Each floor has a number of associated layers for hidden lined, center lines, dimensions, appliances, electrical, etc. Because the layer names are uniform, the following syntax can be used, either at the command prompt, or within a menu, to thaw all layers, set layer "FL1" as the current layer, then freeze all layers that don't start with the characters "FL1." Note the use of wildcards "*" and "~."
- Layer;T;*;S;FL1;F;~FL1*;;
- The "*" is a wildcard meaning "all." The "~" (tilde) is a wildcard meaning "all except."
6. **Set ALL properties to BYLAYER.** Avoid the urge to use multiple colors, linetypes or lineweights on a single layer. If you get a drawing done by someone who has done this, use FILTER or QSELECT to move objects layers and then change their properties to BYLAYER.
 7. **Don't use SNAP and GRID unless it is really useful.** Users sometimes set a snap and grid for every drawing as an aid to layout, a holdover from an earlier CAD era. Unless the drawing you are doing is really modular, avoid making what amounts to rounding errors. Use direct-distance entry, object tracking, polar tracking, coordinates, and object snaps instead. There are exceptions – schematics, flow diagrams, and charts can be any size you choose, so it is smart to use a snap and grid. But those are special applications.
 8. **Learn to use direct distance entry, OSNAPS, object tracking, temporary tracking, and polar tracking.** They are great tools that can dramatically speed up your work once you understand how they work together. Check out the newer OSNAPS, like M2P, Temporary Tracking, Parallel, and Extension.
 7. **Use Paper Space.** These steps, followed in sequence, will help:
 1. Create your geometry FULL SIZE in model space, but do not add dimensions, text, or hatches.
 2. Setup a layout with all views at the proper plot scale.
 3. Add dimensions, text, hatches and schematic symbols to your drawing from (but not necessarily *in*) Paper Space.
 4. Use the express tool CHSPACE to move objects between paper and model space.
 8. **Draw perfectly.** At some point, someone using your drawing will assume you did.

9. **Set text 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 what you set the text height for in your DIMSTYLE. Setting a height of zero gives you control over text height every time you enter it.
10. **Don't use the name "Standard" for text or dim styles.** If you do, they may conflict with a text or dim style of the same name when inserting your drawing into another drawing.
11. **Be VERY cautious when using REFEDIT.** It is used to edit block definitions or external reference drawings and it will replace the originals if you tell it to. You had better mean it. To protect drawings from being edited with REFEDIT, set XEDIT to 0 before saving the drawing.
12. **Be careful when moving or renaming files.** Don't change file names or locations for Hyperlinks, XREFs, images, menu files, icon .bmp files, or other support files unless you know how to redefine the path used to locate them. Otherwise you will get blank rectangles for images, a line of text for XREFs, your menus won't load, many commands won't work, or you will see clouds or questions marks on your custom toolbars. Using the "Relative Path" option can help for images and xrefs.
13. **Plot from Paper Space.** Once you understand it, paper space can save you a lot of time and allow you to do things that you just can't do without it.
14. **Put dimensions in model space.** But don't place any dimensions until you have set up your scaled viewports in paper space. That way you can have the sizes scaled automatically.
15. **Use relative polar angle settings.** You can often reduce the amount of math you have to do in your head by changing the polar tracking mode from *absolute* to *relative* in the Polar Tracking Settings dialog box. You can also type any value as the incremental value in Polar Tracking, although anything less than one degree is hard to control.



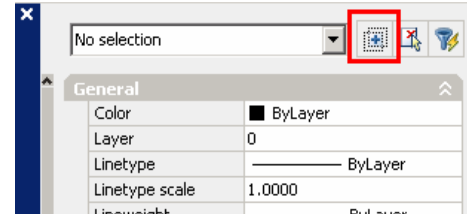
16. **Control English versus metric settings.** Starting from scratch with English (acad.dwt) or Metric (acadiso.dwt) units controls the files used for linetypes and hatch patterns. "ACAD.lin" and "ACAD.pat" are used for Imperial, and "ACADISO.lin" and "ACADISO.pat" are used for metric. You can change this using the MEASUREMENT variable. The Imperial setting is 0; the metric setting is 1. Changing this will NOT change the limits or the default dimstyle after the fact. When plotting a metric drawing, change the drawing area to "Metric" in the Plot dialog box for 2004. R2005 made a change in the plotting process. Whenever you are plotting a drawing started with the Metric template, (or when you have changed the setting for Measurement to 1) and you select a paper size measured in inches, the plot scale will automatically be set to 1:25.4. If you are getting odd results when plotting, check this value.
17. **Let AutoCAD® control your linetype scale.** Don't try to control the appearance of lines using LTSCALE. It is a global variable. Instead, set up your paper space viewports with PSLTSCALE set to 1 (the default). That way your linetypes will scale to the reciprocal of the ZOOM factor you used for each viewport, making them all look

the same in each viewport, even if they are at different scales. To adjust an individual line, use the PROPERTIES window.

18. Don't let drawing aids get in the way. You can quickly turn off Dynamic Input, running OSNAPS, polar tracking, and object tracking when they get in the way by using the buttons in the status bar, or the F3 key for OSNAP, the F10 key for POLAR, the F11 key for OTRACK, F12 key for Dynamic Input. But by all means, use the drawing aids available to you in AutoCAD® when you need them.

19. Don't select the + icon in the Properties Tool Palette. The Properties Tool palette has a slight design flaw that catches a lot of people.

If you find that you can no longer select objects repeatedly without having the last selection lose its highlighting, check the PICKADD variable. It will be changed if you try to close the Properties Palette by selecting the + button. This was improved in 2005 by moving the PICKADD button from the far right corner.

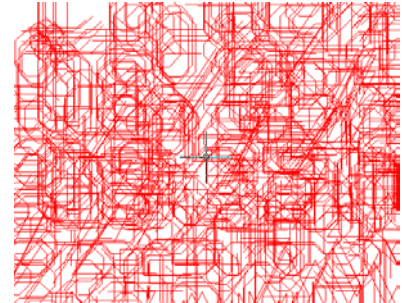
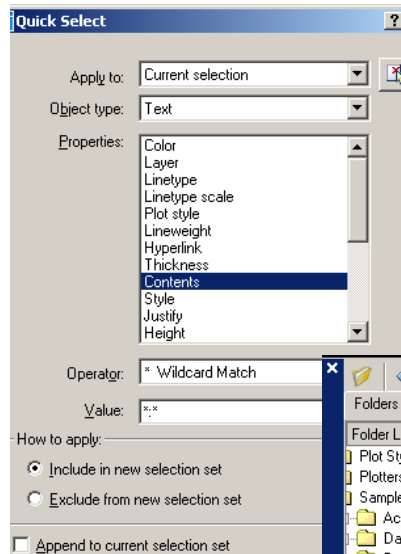


20. Learn to use QSELECT. It is VERY useful when trying to fix up a problem drawing. I once received a .dxf file from the engineer of our city hall. He had generated a large number of points using GPS equipment to map the location of sewers, drains, manhole covers, etc. The problem he had was that all the points, and all the text was placed on one layer. The text height was so large that all the text overlapped and was unreadable. The solution?

I used QSELECT to select all of points and put them on their own layer, which I immediately locked to protect the very valuable locations generated by the GPS software.

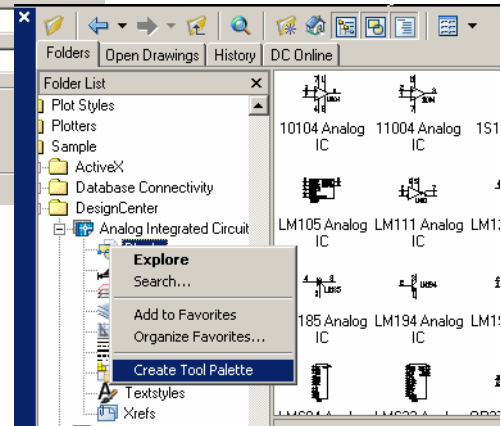
I then used QSELECT to select all of the text (height 294) and change it to 5 so it would be manageable. (See image)

I then used QSELECT to select all text containing a colon (all the dates) and put them on their own layer. This requires using the "*" Wildcard Match Operator (nice to know some DOS). By placing "*:*" in the Value window, you get all text containing a colon anywhere in the string.



21. Create Tool Palettes to enforce standards. You can create a tool palette that contains all of the blocks from a symbol library drawing, with a single selection using AutoCAD® Design Center. Locate the drawing in the browser window of ADC, right click on it, and select "Create Tool Palette."

Once you have 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 is 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.



22. Start other windows programs from AutoCAD®. Type "EXPLORER" at the command prompt to start Windows Explorer. Type "START" at the command prompt, followed by the name of the .exe file for any registered application and you can start any external program. The following are the .exe files I find most useful:

CALC	The window Calculator
IEXPLORE	Internet explorer, although the AutoCAD® "BROWSER" will do the same
EXCEL	MS spreadsheet program
MSACCESS	It would be nice if the MS products had consistent naming conventions, but...
WINWORD	MS Word, see line above

These, and other external commands, can also be created from within the ACAD.pgp file.

23. Solve some common puzzling behaviors.

Absolute Coordinates don't work. You start up R2006 and start to draw. When you enter an absolute coordinate, it is almost as if you had typed a relative coordinate...because, you did. By default, the Dynamic Input function of R2006 eliminates the need for the "@" symbol by using relative coordinates, because they are used much more often. If you want to enter absolute coordinates, precede them with the "#" symbol. If you always want them, change DYNPICOORDS from zero to one.

Hatch doesn't change when boundary is edited. HPASSOC must be turned on for your hatch patterns to be associated with the object you hatched. Then the hatch will update when editing.

Hatch boundary erases with hatch. If PICKSTYLE is set to 3, and you erase a hatch pattern, the object that you hatched will also be erased. PICKSTYLE also controls the selection of GROUPS. There are four setting for PICKSTYLE, so if you want to simply turn groups on and off, use the Ctrl-H combination instead. This used to be Ctrl-A, but that changed a few released back to comply with Windows standards.

Can't change object's layer with layer control panel. If PICKFIRST gets turned off, you can no longer use NOUN/VERB selection: i.e., you can't select an object and then issue a command. PICKFIRST must be on to select an object then picking the destination layer from the layer control list.

You get the error message "...not in the working set." Prior to R2006, if you double-clicked on a block or an XREF, the "Refedit" toolbar is displayed and you are able to edit the object. Unfortunately, it is very easy to close the related toolbar and be left in limbo. You may think you are back to drawing until you get the cryptic "...not in the working set." message. Type "REFEDIT" to bring the toolbar back up, then "cancel" or "save."



Typing "Y" when using PEDIT on a line results in "Invalid option keyword" prompt. PEDITACCEPT, added to R2004, allows you to eliminate the prompt "Object selected is not a polyline. Do you want to turn it into one?" when you use PEDIT. Since you have typed "Y" automatically for years, you still do it, but with PEDITACCEPT set to one, the "Y" becomes an invalid option.

Objects no longer show up while being moved or copied. DRAGMODE must be set to one.

Objects no longer highlight when selected. HIGHLIGHT must be set to one.

Dimensions have incorrect values when applied in Paper Space. DIMASSOC must be set to 2 in order for Paper Space dimensions to be associated with the geometry they represent. If you opened a drawing done before R2002, when there was no DIMASSOC variable, the setting for DIMASSO is used – probably 1 (hopefully!), and dimensions in Paper Space were based on the sheet, not the geometry.

24. Reprogram the F1 key. I don't know about you, but I often hit the F1 key when I mean to hit the ESC key on my laptop. To avoid getting the help system, I have changed the behavior of my F1 key. This is done differently in R2006 than in earlier releases. For R2006, use the customizable user interface (CUI) to define a command for cancel, then assign it to the F1 key.

For pre-R2006 releases, add the following line to the ACAD.mns file (If you edit this file, back it up first, and use a text editor like Notepad or WordPad to edit it.) The line goes at the beginning of the section labeled ***ACCELERATORS in the menu file:

```
***ACCELERATORS
["F1"]^C^C^C
```

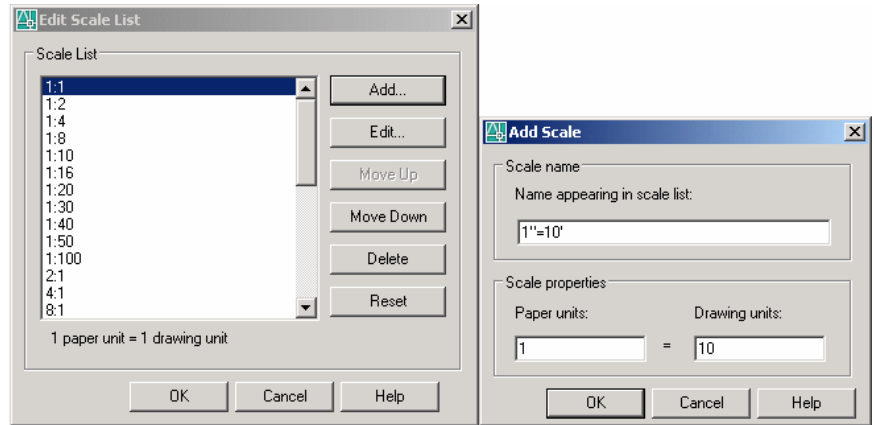
Now F1 is a triple cancel.

25. Control the new features of R2006.

With every new release of AutoCAD® we are faced with a decision: should we turn off the new behavior so that the software seems more familiar, or should we embrace the changes? That is your call. I always try to embrace the new features, because I feel I need to understand how to use them so I can teach others and answer questions. (Including the most common question: "How do I turn off...?" Some changes in 2006 to look for.

SCALELISTEDIT. If there is a scale you use missing from the standard list, you can now add it via this command. You can also delete the scales you don't use at all. To get them back, use the "Reset" button. I like it.

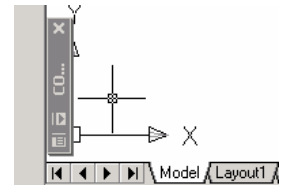
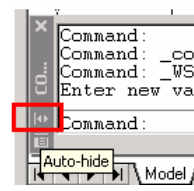
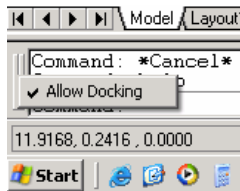
In this example, I have added an engineering scale by giving it the name that would appear in a title block, but assigning the scale factor to be used for drawings done in decimal feet, which is common for civil engineering.



Command Line. The command line can now be turned off. Thankfully, it can also be turned back on. There have been many calls over the years to *eliminate* the command line to make the AutoCAD® interface more up-to-date. R2006 has the right solution, in my opinion, since the modernists can get rid of it, and the rest of us can still use it --- but they have a point. It takes up room and you need to move your eyes down or up to refer to it, which you must do often. Try working without it to see how you like it. Here are your choices:

- Turn the command line on (COMMANDLINE), and turn Dynamic Input off (DYNMODE=0). This gives you the same behavior you have with earlier releases.

- Turn the command line on, but turn "Allow Docking" off, and turn "Autohide" on. Now it is out of the way unless you need it. Move your cursor over it, and it will expand. You can also toggle it on and off using the Ctrl-9 key combination.



- Leave Dynamic Input set for "All." You will now see the commands you type, the options available to you, and the dimensions that result, all at the cursor. This takes getting used to for three reasons: it is easy to lose the cursor on the screen; the cursor now drags a lot of stuff around with it; and you need to use the up/down arrows to scroll through options if you want to see them.

- Show only input, not dimensions at the cursor (DYNMODE = 1). This reduces the clutter, but gives you heads-up control of options.
- After working with R2006 for a while, my command line is at the top, undocked, with autohide on, and DYNMODE is set to "1" to show only input, not dimensions. When programming, I revert to the old command line behavior.

Use Workspaces and Profiles. The workspace concept is new to R2006. At first workspaces seem similar to profiles. However, they are different and generally unrelated. Workspaces don't control profiles, even though it implies that they do in the help system (I thought they would, and still think that they should), and profiles don't control workspaces. So treat them separately, but I highly recommend setting up at least one profile and one workspace and then using them both. Leave the default settings as they are so you can return to them if you should want to. Use them as follows:

Profiles:

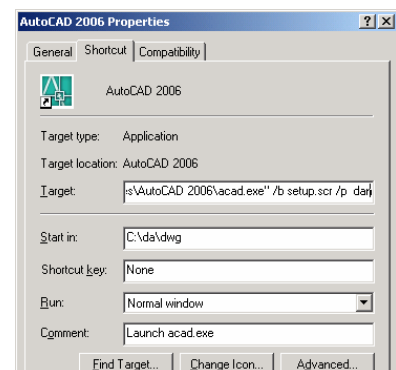
Use them to save settings you make in the "Options" dialog box, including file locations, drafting tool behavior, screen color, and many others. Changes you make take place in the current profile automatically. If you don't want to change the current profile, create a new one before making any Options changes.

Workspaces:

Use them to control the appearance and location of the command line, tool bars, palettes, ADC, and menus. Changes you make are not stored automatically in the current workspace. You must save them by selecting "Save Current As..." on the Workspace toolbar, then specifying the workspace by name. This would allow you to create custom toolbars for specific kinds of projects, and include only those toolbars in that workspace. Unfortunately, there appear to be some glitches in workspaces in this release. The state of ADC seems to change if you exit AutoCAD® with one workspace active, then open it and start another, and custom toolbars can misbehave, but those things will be worked out, and even with them, workspaces are nice.

Making them work together:

Well, unfortunately there is no command line switch for workspaces, but if you did want to have separate desktop icons for starting AutoCAD® up with both a specified profile, and a specified workspace, you can. Let's say you have a profile named "dan" and a workspace named "dan." First, create a text file with the following line in it: "WSCURRENT dan" and save it with the extension, for example "startup.scr." Now make a copy of the AutoCAD® desktop icon, and add the following to the startup line in the properties window: /p dan /b startup.scr. The whole line now looks like this: "C:\Program Files\AutoCAD® 2006\acad.exe" /b setup.scr /p dan. Now whenever you select this icon, you will have both your profile and your workspace.



Dynamic Blocks. Like other releases of AutoCAD®, R2006 utilizes round-tripping for those situations where you must open an R2006 drawing in R2004 or R2005 (which you can do). The new elements in R2006 will make it back intact. However, there is one wrinkle with the new dynamic blocks. They have two names in R2006, a real name and an anonymous name. The real name is the name given to the original block definition. The anonymous name is preceded by an * and is randomly assigned (*U9, for example) to each individual variation of the block when it is inserted.

When the drawing is opening in R2004, the real name of the block is not listed in either the "Insert" or "Block" dialog box. Although the name is given in the "Properties" pane or when you use the LIST command, you cannot insert a new instance of a dynamic block. You can, however, make a copy of a dynamic block, scale it, or explode it. REFEDIT will not work on an anonymous block. By the way, dimensions and hatch patterns are also anonymous blocks, and also have names beginning with an *.

26. Express tools. There are some great, even indispensable express tools. Make sure that you have them loaded. In R2006, you are prompted to do that during installation. In earlier releases, you have to select "Install Express Tools" in the installation wizard AFTER installing AutoCAD®. If the Express Tools aren't available, you MAY be able to get them to show up by typing "EXPRESSTOOLS" at the command line. If they seem to be available, but there is no pull-down menu labeled "Express" try the command EXPRESSMENU. If those attempts don't work, you must install them from the CD.

My favorites:

CHSPACE. This command is indispensable, and it is time it was migrated into the actual program. It lets you change objects between paper space and model space while retaining their relative scales. This makes it possible to place text or dimensions wherever it is convenient while working, and then move them to their permanent home later. If you ever decide you placed something in the wrong space, you need this command.

LAYWALK, allows you to walk through the objects on various layers to give you a visual clue as to what you have placed where. Very valuable when dealing with someone else's disaster drawing.

TXT2MTXT is used to combine individual lines of DTEXT or TEXT (same thing really) into a single MTEXT object. It doesn't format the final 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 is very nice.

FLATTEN. I have recommended this command to a lot of people in companies that do civil design. They will call me because they are having trouble with a drawing from "someone else." Object snapping to endpoints produces odd results because, it turns out, the elements are not all at the same elevation. FLATTEN quickly and, usually very thoroughly, places every object at an elevation of zero.

MKSHP and MKLTYPE commands give you the ability to create complex linetypes: those containing text or shapes. This process requires that a shape be created first (MKSHP) than the linetype (MKLTYPE).

LAYMERGE. I often advise students and companies to use as many layers as they MIGHT need so that they 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. When you do have too many layers, this command allows you to simply put them together into one.

DIMEX and DIMIN allow you to export dimension styles and then import them into a different drawing. This can be done through AutoCAD® Design Center, or by inserting a drawing that contains the dimension style, but these express tools make it easier to do.

DIMREASSOC is pretty nice. 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 do not match. My favorite application of this command is to simply find any dimensions that were overridden. I actually have done that on rare occasions, when I had an imminent deadline and had to plot something out. With DIMREASSOC, I can find those places and go back and fix the geometry. You can also change them automatically, which might cause more problems.

SYSDLG. The System Variable 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. The best thing is that you can save the setting 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 along with you. It will reduce the number of surprises you might encounter.

RTEXT. With RTEXT you can insert a constantly updated date or drawing name and path (among other things) into a title block, or anywhere else on a drawing. This is great for releases that don't support fields. See the EXPRESSTOOLS help system for the syntax.

OVERKILL. Gets rid of overlapping lines, line segments that connect in what appears to be a single object, and duplicate objects on top of other objects. Can really clean up a mess of a drawing.

27. USING AUTOLISP

I don't think that everyone who uses AutoCAD® must know how to do AutoLISP programming, but it is not that hard to do, and really does expand your control of the software. However, I think that you should be able to find and *use* existing AutoLISP routines, even if you don't understand how they were written.

This is a good time to mention an organization that every AutoCAD® user should join: AUGI, is the Autodesk User Group International. It is an independent (although they receive some support from Autodesk) group of people who use Autodesk software. Their website (AUGI.com) contains forums for various Autodesk software, including AutoCAD®, where you can get information, and get questions answered. They also publish tutorials, and have classes available on virtually all Autodesk software.

It is also a good place to get AutoLISP code, which are programs written in the LISP programming language that automate some function within AutoCAD® or any of the vertical programs that are based on AutoCAD®.

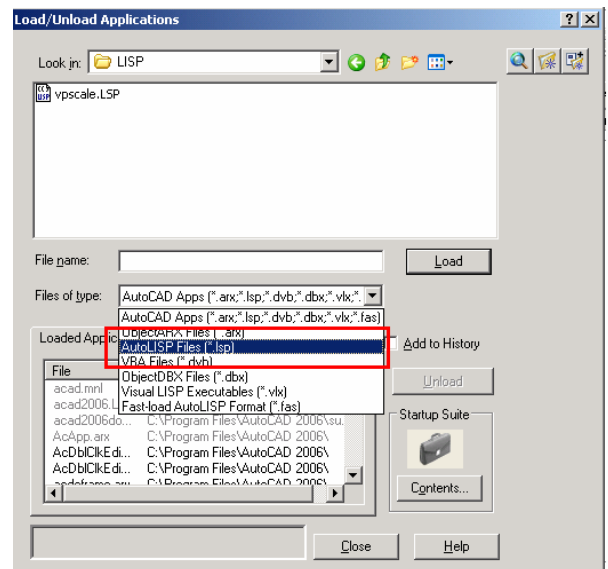
There are other sites as well, but if you type "AutoLISP" in Google, you will get 380,000 hits, many of which offer AutoLISP code for download. Be more specific about what you want a program to do, and you may find just what you want. So, what do you do once you find an AutoLISP program?

An AutoLISP program is a text-based file (unless it has been compiled) that is saved with the extension ".lsp." To use such a program, you must load it in an AutoCAD® session, either manually or automatically. Once you have saved it, type APPLOAD (alias is AP) at the AutoCAD® command prompt, and you will get a dialog box that allows you to load a variety of different file types. Pick lisp from the list, and use the top portion to find your file.

If you want your program to load every time, use the "Contents" button to place it in the "Startup Suite."

That loads the program, which probably means that you now have one or more new commands at your disposal. The command names could be anything. To find out what the commands created by this program are called, just open the file (right-click right in the "Load/Unload Applications" dialog box and pick "Open."). With the file open, search for the following combination "defun C:" "C:" denotes the name of a new AutoCAD® command, NOT a drive letter. So if you find "defun C:CC" the new command name is "CC" and it can be typed at the keyboard.

Here is a sample program that I wrote for a company that wanted to be able to place text in a Paper Space viewport and have it automatically scale to be the size specified by DIMTXT when the drawing was plotted. If you type this program EXACTLY as shown, you can use it in AutoCAD®. Note that the command name created by this program is STXT (see the entry "defun c:STXT").



Things Every AutoCAD® User Should Know - Expanded!

Any text preceded by one or more semi-colons can be omitted when you type in the program. They are simply annotation and not part of the code. When you do type this, it is easier to use the Visual Lisp editor within AutoCAD® than to use NOTEPAD, because it uses color-coding. But Notepad will do.

```
;;;VPSCALE.lsp
;;;Dan Abbott
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;;;March 12, 2005
;;;Places text in model space that is scaled to the viewport it is placed into.
;;;Text will plot at the size specified by DIMTXT
;;;Provided "AS IS," with all faults. You are free to use this code in any way you want.
;;STXT [ScaledTeXT]command
```

(defun c:STXT (/ txtht cvp vhms ss1 entlist vhps ps p1)	;local variables
(command "style" "romans" "romans" "" "" "" "" "" "")	;creates text style
(setq tm (getvar "tilemode"))	;determines if in model or paper space
(if (= tm 1)	;start in model space do the following
(setq p1 (getpoint "\nSelect insertion point of text: ")	
txtht (getvar "dimtxt")	
)	;if NOT in model space do the following
(setq txtht (getvar "dimtxt")	;gets the current dimstyle text height
cvp (getvar "cvport")	;gets the number of the current viewport
vhms (getvar "viewsize")	;gets the model space height of cvport
ss1 (ssget "X" (list '(0 . "viewport") (cons 69 cvp)))	
	;selects just the current viewport
entlist (entget (ssname ss1 0))	;lists properties for the current viewport
vhps (cdr (assoc 41 entlist))	;gets the paper space height of cvport
ps (/ vhms vhps)	;sets ps to the ratio between the ms and ps sizes
txtht (* ps txtht)	;multiplies ps times dimtxt
p1 (getpoint "\nInsertion point of text: ")	;gets the starting point of text
)	;endssetq group
)	;ends if
(command "dtext" p1 txtht "")	;uses DTEXT to place text at proper height
)	;end defun

So, those are things that I think every AutoCAD® user should know.

