



Autodesk
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Stop Wrestling and Start Dancing with AutoCAD®

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GD401-2 This class is for those of you who go home at the end of a busy day and wonder just what you accomplished. Have you ever spent hours working on something that you thought would've been a simple task? Why do some users seem to dance with AutoCAD while others seem to wrestle? In this session, we'll cover several ideas on how you can take what you know and use it in clever ways to increase productivity. If you've ever wanted to turn a week's worth of work into a 5-minute task -- and we'll show you examples -- then this class is for you.

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About the Speaker:

Based in Minnesota, Darren has held a variety of positions over the last 15 years such as CAD/CAM engineer, CAD administrator, and CAD/CAM systems developer. He founded a consulting and development business called Minnesota CADWorks. While Darren's true interest is the automation of manufacturing systems, his experience ranges from manufacturing to architecture, and has led him to projects varying in scope from dress patterns to gas turbine piping. He's been a technical editor for the 2002, 2004, and 2005 versions of the "AutoCAD Bible" (by Autodesk Registered Author Ellen Finkelstein) and has been a regular contributing author for "Inside AutoCAD," since 2001.

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Introduction

Over the years, I've found that most users don't put a lot of deep, self-reflecting thought into how they use their software. After all, you're not paid because you know how to use AutoCAD; you're paid because you know how to construct a building, design a machine, or how much a grade needs to slope for proper drainage. As a result, we often fall into a rut. We're busy; we're doing things, so we must be working productively right? Wrong. This rut is what makes it possible for us to continue working as we always have, without exploring that "new" feature in the next release. It allows us to know what that old command is, without ever really knowing what it does. It's why it personally took me 3 releases of AutoCAD before I started using Grips for editing.

This class isn't about tips and tricks to make you more productive, even though most of you will learn a few during this class. Instead, it's about teaching you to think differently about how you view your software. While going through this course, I challenge you, don't just think about the concepts, tips, techniques and suggestions and their direct result. Think about what's happening at their core, and how parts, if not all, could apply to other areas of what you're doing. Perhaps even more importantly, think about where some of these ideas or suggestions came from. Exactly how do I (or any instructor at Autodesk University for that matter) know all this stuff? The answer is that we tap every resource we can, reflect and think differently about what we learn.

It's a widely held belief that if you study a topic for 15 minutes a day, in 5 years you'll have accumulated enough knowledge, that you could be considered a national expert on that topic. That's not a lot of time, but the average person spends less or even no time doing this, which is what makes this possible. Maybe you don't want to be a national expert, and frankly there's no need. The point is, commit to yourself to spend 15, 10, or even 5 minutes a day, reflecting and thinking about your software, how it works, what it does, and how you can apply it in your environment. You'll find in short order that you'll be more productive, efficient and accurate in your daily work. Doing this just may result in me sitting in one of your classes at next year's Autodesk University and nothing would please me more, than having that happen. So on that note, let's get started...

Productivity Settings – Fine Tune Your AutoCAD Experience

Part 1: That's just plain annoying

Under the annoying heading, here are a few system variables that may or may not speed up your day, but it sure won't seem as long if AutoCAD isn't constantly aggravating you. Simply type the system variable name at the command line and enter your desired value.

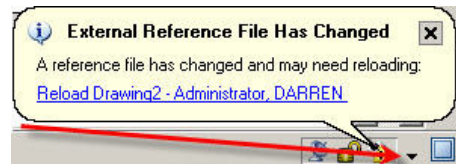
TRAYTIMEOUT – Default 0 (zero) leaves system tray notifications for services up until you close them. Set from 1 and 10 to indicate how many seconds the notification should be displayed. Change this once and AutoCAD will remember from session to session.

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TRAYNOTIFY – Default 1 (one) tells AutoCAD to display system tray notifications for services. Change this to 0 (zero) and get rid of them all together. Change this once and AutoCAD will remember from session to session.

Alternatively, you can change either of these settings by clicking on the little triangle and clicking Tray Settings from the popup menu.

XREFNOTIFY – This is similar to TRAYNOTIFY. But while TRAYNOTIFY enables or disables tray notifications for services; XREFNOTIFY enables or disables Xref notification as one of the services that are used. In other words, if you want service notification but don't want Xref change notifications, set this to a value of 0 (zero), set it to 1 (one) to use Xref notification with alerts appearing only in the system tray, or 2 for tray and bubble alerts. Set this value once and AutoCAD will remember if from session to session.



DATALINKNOTIFY – Again similar to TRAYNOTIFY. But while TRAYNOTIFY enables or disables tray notifications for services; DATALINKNOTIFY enables or disables Data Link notification as one of the services that are used. In other words, if you want service notification but don't want Data Link change notifications, set this to a value of 0 (zero), set it to 1 (one) to use Data Link notification with alerts appearing only in the system tray, or 2 for tray and bubble alerts. Set this value once and AutoCAD will remember if from session to session.

LAYERNOTIFY – Again similar to TRAYNOTIFY. But while TRAYNOTIFY enables or disables tray notifications for services; LAYERNOTIFY enables or disables layer reconciliation notification as one of the services that are used. In other words, if you want service notification but don't want layer reconciliation notifications, set this to a value of 0 (zero), set it to 1 (one) for notifications during PLOT, 2 during OPEN, 4 for Load/Attach/Reload Xrefs, 8 for /retore of layer states, 16 during SAVE, and 32 during INSERT. These are bit coded flags, if you want notification for several settings, just add them up and use that value. Set this value in each drawing where you want it to take effect.

(setenv "XNOTIFYTIME" "#") – This isn't a system variable, it's an environment variable that needs to be set with the AutoLISP syntax as shown. This setting tells AutoCAD how often to scan your drawing for changes to Xref it uses. By default, AutoCAD does not set this value in which case it defaults to 5 minutes. You can change this "#" value from a setting of "1" (one) to "10080" (seven days). The number must be enclosed in quotes and "XNOTIFYTIME" is case-sensitive. Once set, you can verify its value by typing at the command line in AutoCAD, the AutoLISP code (getenv "XNOTIFYTIME"). If you change this once, AutoCAD will always remember it from session to session.

PUBLISHALLSHEETS – Default 1 (one) tells AutoCAD to automatically add Model and/or Layouts for all open drawings when Publish is started. Change this setting to 0 (zero) and AutoCAD will only add the Model and/or Layouts of the current drawing when Publish is started. Change this in every drawing you want it to apply.



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PAPERUPDATE – The default setting of 1 (one) warns if the printer doesn't support a paper size used by the layout. Set to 0 (zero) to disable this warning. Change this once and AutoCAD will remember from session to session.

PEDITACCEPT – Default 0 (zero), changing this to 1 (one) disables the “Object selected is not a polyline. Do you want to turn it into one? <Y>” prompt issued by the PEDIT command when you select objects that are not Polylines. Change this once and AutoCAD will remember from session to session.

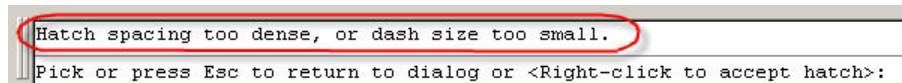
Part 2: Still annoying but at least AutoCAD's trying to be on your side

Here are a few more settings, some obvious, some less obvious. While these may seem annoying when you encounter them, AutoCAD uses them to help protect the user from doing things to themselves that they might not be aware of.

(setenv “MaxHatch” “#”)

– This isn't a System Variable, rather it's

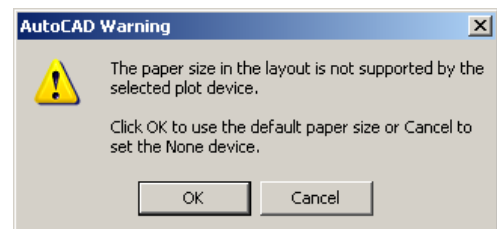
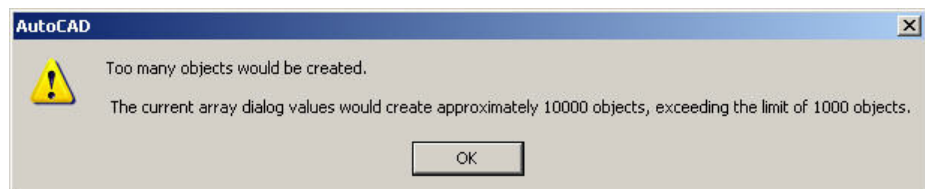
AutoLISP code that changes one of AutoCAD's environment variables. While AutoCAD says it supports values of “#” = “100” (one hundred) to “10000000” (ten million), you can enter any whole number contained in quotes, even invalid negative numbers (not recommended by the way). This setting won't allow you to create a single hatch containing more than the specified number of objects. Doing so can impact the performance of AutoCAD's Hatch command as well as your drawing. Note, the “MaxHatch” environment variable is case sensitive and must be typed exactly as shown. To check the existing setting, type (getenv “MaxHatch”) at the command line. Like some of the system variables, AutoCAD remembers this setting from session to session.



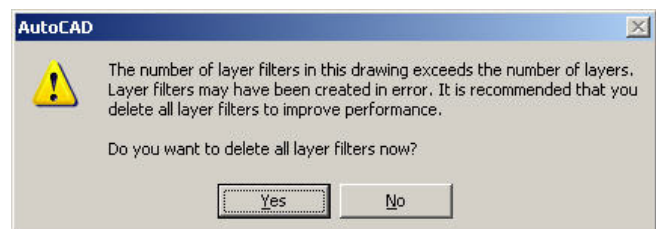
(setenv “MaxArray” “#”)

– Just like “MaxHatch”, the “MaxArray” environment variable

does the same thing only for the Array command, limiting the number of copies your array will produce. As is cousin above, the “MaxArray” environment variable is case sensitive and must be typed exactly as shown and takes the same types of values formatted the same way as “MaxHatch” does. You can check the existing settings by typing (getenv “MaxArray”) at the command line.



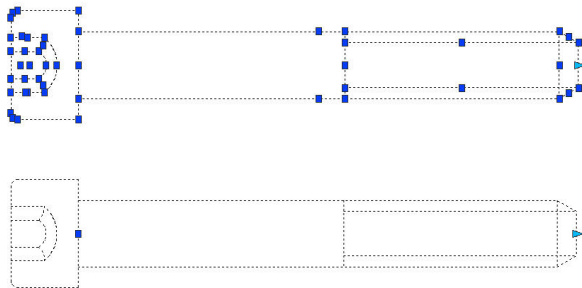
LAYERFILTERALERT – This system variable is intended to prevent your drawing from becoming bloated with too many layer filters that it's sluggish. AutoCAD criteria for determining this and takes action based on



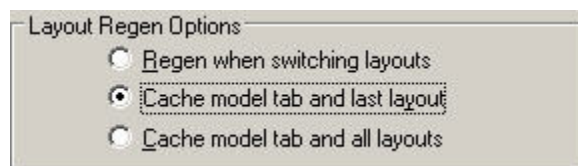
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this setting if the number of filters is over 100 and there are more filters than layers. A setting of 0 (zero) does nothing a setting of 1 (one) deletes all layer filters upon opening the Layer Properties Manager no questions asked when AutoCAD's criteria is met. When set to the default value of 2, upon opening the Layer Properties Manager you are prompted if you want to delete the layer filters or not. Choosing to delete them removes them all. I recommend setting this to a value of 3. When set to 3, you are warned upon opening the drawing, not upon opening the Layer Properties Manager and offered the option of which to delete, and which to keep. Furthermore, there's a bug in AutoCAD 2007 & 2008 that deletes all "Group" layer filters leaving only the "Properties" layer filters, no questions asked when this is set to 2. In other words, if there are Group layer filters that exceed AutoCAD's threshold, settings 2 & 1 (one) act the same. AutoCAD will remember your changes from session to session.

LAYOUTREGENCTL – With a default setting of 2, AutoCAD regenerates each tab (Model and Layout) the first time you access it. After that, it uses cached memory to restore that tab when you return to it. The more layout tabs you have in a drawing as well as the longer you work in it in one session, AutoCAD will gradually and continually slow down. This slowdown is magnified if you're running a "memory challenged" computer. Setting this to 0 (zero) regenerates each and every time you switch Model or Layout tabs. A good compromise is a setting of 1 (one) that only caches the Model and last layout tab. You can also make this change in the Display tab of the Options dialog.



GRIPOBJLIMIT – With a default of 100, this setting controls how many objects can be selected before not displaying grips. Valid settings are between 0 (zero) and 32,767. A setting on 0 (zero) means grips are never displayed. A setting of 1 (one) means grips will be displayed only when one object is selected. Like many others, AutoCAD will remember this setting from session to session.



Bit Coded Values

Bit coded values might seem odd when expressed with decimal numbers. That's because their origins are binary. To understand how Bit Coded values work, let's express them in Binary form using a partial listing of the OSMODE System Variable settings, that define which Osnaps are set.

Decimal	Binary	Description
0	0000	None
1	0001	ENDpoint
2	0010	MIDpoint
4	0100	CENter
8	1000	NODe

Now, let's look at the values added together...

3	0011	END,MID
10	1010	NOD,MID
11	1011	NOD,MID,END

If you look at the binary result, there's just "On" (one) and "Off" (Zero) switches. We use the decimal value for the variable settings.



Part 3: Get what you need the first time

OSOPTIONS (formerly OSNAPHATCH) – While OSNAPHATCH still works; it's being phased out by the introduction of OSOPTIONS. In fact, making changes to OSNAPHATCH changes the value of OSOPTIONS. A value of 1 (one) tells AutoCAD to ignore Hatch objects when snapping to other objects. A setting of 2 tells Osnap to ignore geometry with a negative Z value during the use of dynamic UCS. This is a "bit coded" value. This means that you "add" the values together to get functionality you want. E.g. If you want both, settings 1 (one) and 2, just add them together and use a value of 3. AutoCAD will remember this setting from session to session.

OSNAPZ – While this might seem the same as the 2 option from OSOPTIONS, it's not. OSOPTIONS configures Object Snaps to ignore negative Z geometry during the use of Dynamic UCS. OSNAPZ on the other hand, does not ignore negative Z geometry; it only ignores the Z-value (keeping the X & Y) of negative Z geometry. It substitutes the current elevation for the current UCS with the X/Y values from the geometry snapped to. Note, this setting is NOT saved other than in memory. The next time you start AutoCAD, it will revert back to its default value of 0 (zero). If you need this set frequently, create a tool button that sets it. You could also add it to your Acad/AcadDoc.Lsp if you want it always set to 0 (zero).

DELOBJ – AutoCAD ships with a default value of 1 (one). This deletes much of the original geometry used to create 3d solids once the solid is created. A setting of 2 deletes even more. -1 (minus one) and -2 do the same thing respectively but prompt before deleting the original geometry. If you've even been frustrated but not having the original geometry when you make 3d solids, try a setting of 0 (zero) which retains it all or at the very least, a negative value so you're prompted each time. AutoCAD remembers this setting from session to session.

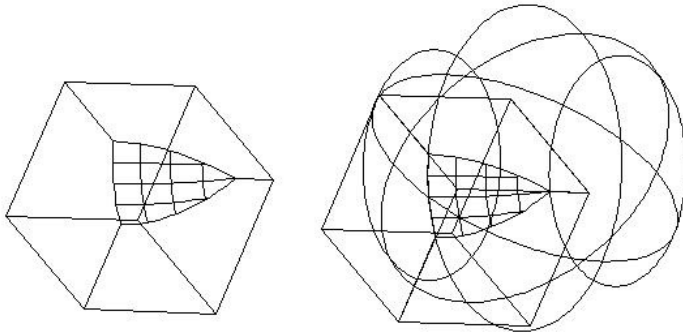
BINDTYPE – This system variable is not saved and only remains during the AutoCAD session it was used in. Its default setting is 0 (zero) which means, that the funky layer names Xref's create, stay when the Xref is bound to the drawing. That is, an Xref layer name of "xref1|one" becomes "xref\$0\$one". When set to 1 (one), layer names are converted to something simpler when the Xref is bound to the drawing. The same Xref layer name in the previous example, "xref1|one" becomes "one" when bound.

PELLIPSE – This system variable controls whether AutoCAD should create true Ellipse objects, the default setting of 0 (zero), or if it should draw them as polyline objects, a setting of 1 (one). Actual Ellipse objects are more mathematically accurate but that object type doesn't allow as much in the way of editing options as Polyline. For more "editability" of ellipse shapes, change this to 1 (one). There are, other ways to deal with Ellipse objects, which we'll cover later. Also note that this setting is saved in the DWG file itself. If you change it, the drawing you are in when you change it will remember, but others will not. To change this for all drawings, you'll need to change its setting in your template drawing (DWT) file or change it within your Acad/AcadDoc.Lsp file.

SOLIDHIST – This system variable controls whether composite history of the solid is recorded or not for all new and existing solids. Composite history is a record of the geometry that created

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the solid. The graphic below shows the same solid, one with composite history shown and the other with it not shown. In both cases, however, the composite history is recorded. If you turn off composite history recording with a setting of 0 (zero), all existing composite history of solids that is not displayed will be removed and can not be recreated. By default, AutoCAD records composite history by using the setting 1 (one). Changes to this variable affect only the drawing it's changed in. To change this setting for all drawings, you'll need to make the change to your drawing template (DWT) or add AutoLISP code in your Acad/AcadDoc.Lsp file to change the setting upon opening each new file.



Acceptable values for...

INSUNITS
INSUNITSDEFSOURCE
INSUNITSDEFTARGET

Value	Units
0	No Units Specified
1	Inches
2	Feet
3	Miles
4	Millimeters
5	Centimeters
6	Meters
7	Kilometers
8	Microinches
9	Mils
10	Yards
11	Angstroms
12	Nanometers
13	Microns
14	Decimeters
15	Dekameters
16	Hectometers
17	Gigameters
18	Astronomical Units
19	Light Years
20	Parsecs

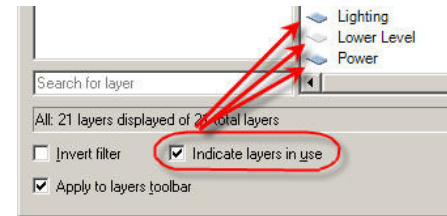
INSUNITS – This variable defines what units the drawing is in for purposes of automatic scaling of Blocks and Xrefs. When a block or Xref is inserted into a drawing, both the Block/Xref's INSUNITS value and the parent drawing's INSUNITS values are examined and the block automatically scaled appropriately to compensate for any different in units. AutoCAD defaults to a value of 1 (one) but can take a value of 0 (zero) – 20 depending on the units needed. This setting is drawing specific. To change its default, update its value in your template drawing (DWT). It is not recommended, however, to add code in your Acad/AcadDoc.Lsp to update this value because you won't always know that every drawing you open was drawn with a particular unit in mind.

INSUNITSDEFSOURCE, INSUNITSDEFTARGET – These two variables go hand in hand. Like INSUNITS above, they define the units of a drawing. However, they only take effect if the source and/or target drawing have a value of 0 (zero) for their INSUNITS system variable. In other words, if INSUNITS wasn't set, these are what AutoCAD should assume. Unlike INSUNITS, these settings are saved in the registry, which means that AutoCAD will remember them from session to session.

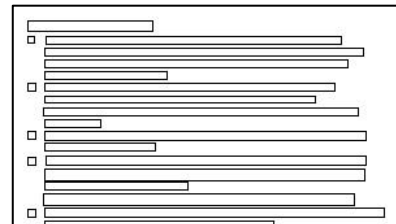
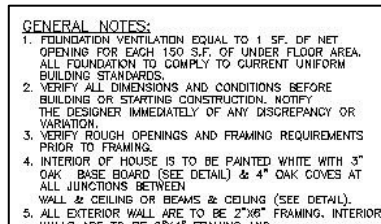
Part 4: Pure productivity

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SHOWLAYERUSAGE – While the default setting of 1 (one) is off, setting this value to 1 (one) uses different icons in the Layer Properties Manager to show which layers are used (contain objects) and which do not. Turning on this option can slow AutoCAD's performance, but occasionally it's a good trade off to turn it on briefly when needed. You can also access this from the toggle in the Layer Properties Manager for a little quicker access. Like many others, AutoCAD will remember this setting from session to session.



QTEXTMODE – This system variable is set to 0 (zero) by default, which tells AutoCAD to display text. If your drawing has a lot of text, you can greatly improve its performance by temporarily changing this value

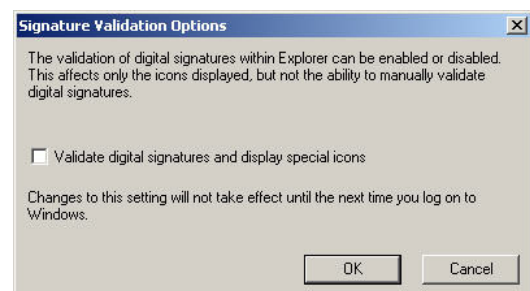


to 1 (one) which; tells AutoCAD to display text as boxed outlines instead of displaying the characters. Zoom, Pan and Regen will all be significantly faster. This value is saved in each drawing file, however, you typically only set it temporarily when you need the performance and/or are not working with the text itself. You'll also note that changes to this system variable are not immediately visible until a Regen is performed. You can also change this setting by typing the QTEXT command. If you hadn't already guessed it, the "Q" in "QTEXT" stands for "Quick".

Hidden Secrets of AutoCAD - What You Don't Know Can Help Or Hurt You

Digital Signatures – Verifying Network Resources More Than DWG Authenticity

If you're running AutoCAD and you have drawing files residing on a server (not your computer), AutoCAD's ability to digitally sign files can unknowingly slow your system performance, even outside AutoCAD, such as Explorer or any program that displays a file list. By default, AutoCAD installs an extension that allows it to display different icons if the DWG file is digitally signed or not. But in a network environment, this can really tank your system performance because AutoCAD needs look into each file briefly to see if it's signed or not. You can turn off this feature by right clicking on a DWG file and selecting the "Enable/Disable Digital Signature Icons" option from menu. This will open a dialog box where you can clear the toggle and click Ok to disable this functionality.



Hatching – The Less Hatch Has To Choose, The Faster It Is For You To Use

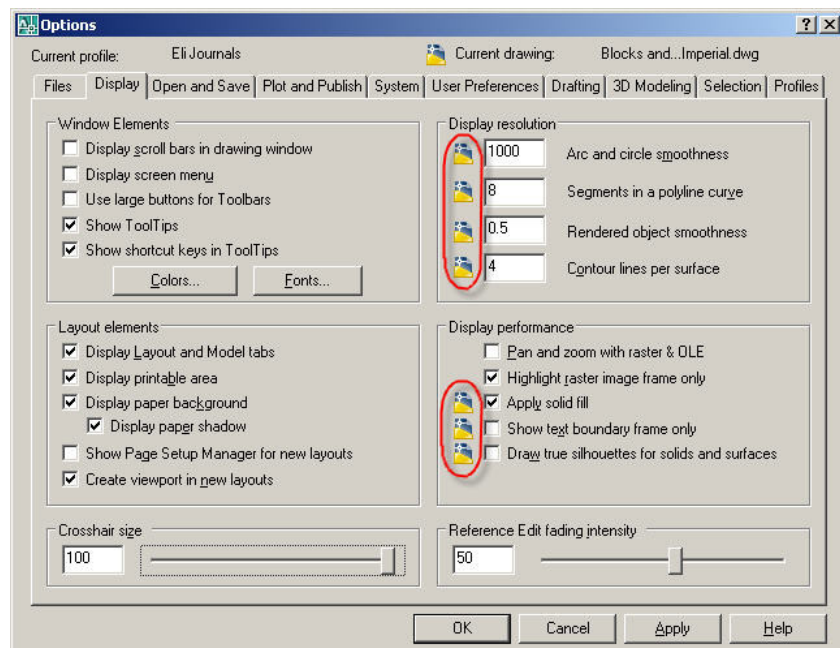
It seems obvious, but we all occasionally forget. When you hatch, AutoCAD analyzes everything on your screen when you select an internal point of an area you want to hatch. You can increase the speed of your hatching by either zooming into an area slightly larger than that which you want to hatch, or use the “Select Object” option instead, which only analyzes what you select, instead of everything on your screen. Keeping these methods in mind will minimize the times you select an area to hatch and your system temporarily locks up with processing the screen geometry.

Associative Dimensioning – Sometimes Its Wise To Disassociate Yourself

The new, improved associative dimensioning in AutoCAD has been a hit since it was introduced. Dimensions that move and change with your geometry even across PaperSpace and Xref, what's not to love? But did you know, that when the DIMASSOC system variable is set to 2 for full associative dimension, this isn't officially supported when dimensioning several different types of objects? True associative dimensions do not officially support other Dimensions, Leaders, Tolerance objects, Multi-lines, any block inserted with the Minsert command, Solids, Traces, 3d Faces, Polygon Mesh Vertices, Polyline Mesh Vertices, Polyline Meshes, Polygon Meshes, 3d Face Records, Viewports, Hatches, Mirrored or Non-Uniformly scaled Block references, Images and Custom objects. Whew! How's that for a list? If you'd like to read up on this more, you can search Autodesk's web site for the knowledge base article “TS71853”. Unlike it says in the article, when attempting to dimension some of these object types, AutoCAD does not create a non-associative dimension. Instead, it often creates an associative one and everything appears fine. That is, until the next timer you open your drawing, and your Dimension's text value seemingly gets out of whack on a hit-or-miss basis.

AutoCAD's Options Dialog – Will AutoCAD Remember This Setting?

There's a lot of places in AutoCAD you can make changes to its settings in a dialog box instead of directly changing its system variables. A lot of times, you won't even know which variable it is that's being changed so it's hard to look it up in the help to know if AutoCAD will remember the change or if it gets saved in the DWG file and needs to be made in all files. As a result, AutoCAD gives you some clues but it's so



Stop Wrestling and Start Dancing with AutoCAD® painfully obvious right if front of you that if you're like me, you probably missed it. If you look at some of the settings in the Options dialog, you'll notice that some of them have a DWG icon next to them. This is your clue that those values are not retained from session to session; rather apply only in the current drawing.

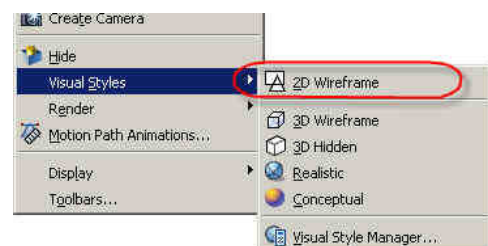
Object Conversion – Unload That Legacy Baggage

If you deal with old legacy drawings or block libraries, files from vendors/customers, or third part applications, you might not be aware that you could be carrying excess baggage. There's a lot of updated object information in the newer releases of AutoCAD that could make your drawing more editable, smaller and faster. You just need to know where to look. Here's a rundown of some of the commands that will help you clean up some of that baggage.

Command or System Variable	Explanation
CONVERYPOLY	This command will convert between Polylines (heavy weight polylines) and LWPolylines (light weight polylines), Use LWPolylines to keep your drawing as small and fast as possible.
CONVERT	Use this command to update old hatches and polylines into the newest format. Not all Hatches will be able to be updated. Polylines don't always need to be updated depending on the PLINETYPE system variable setting which tells, if set properly, can tell AutoCAD to update all Heavy Weight Polylines to the Light Weight Polylines upon opening a drawing. Note, that 3 rd party applications or inserting old blocks can still introduce old Hatches and Heavy Weight Polylines into your drawing. Use this command on your block libraries and archive drawings too to prevent this from happening.
PLINETYPE	This system variable controls whether AutoCAD should automatically convert old Heavy Weight Polylines into the new LWPolylines automatically when your drawing is opened. By default (a setting of 2), AutoCAD will automatically convert these for you. Still it doesn't hurt to check and verify this is set how you'd like it.
CONVERTOLDLIGHTS	Use this commands to update lights you've used in the past on your old drawing to the new light object in AutoCAD 2007. Not all lights are updated properly; you might have to adjust settings like intensity for example.
CONVERTOLDMATERIALS	Use this command to update old materials to the format now used by AutoCAD 2007. They won't always update perfectly, so you may have to adjust material mapping or tweak a few other settings upon conversion.

Pen Weights & Color – Not As Black And White As It Use To Be

It use to be, that all you needed to do to plot Black & White was make use of the proper CTB or STB file. With AutoCAD's Visual Styles, that's no longer the case. CTB and STB settings typically only apply when your view style is set to "2d Wireframe" or if your plot dialog is set to plot in "Draft" mode.





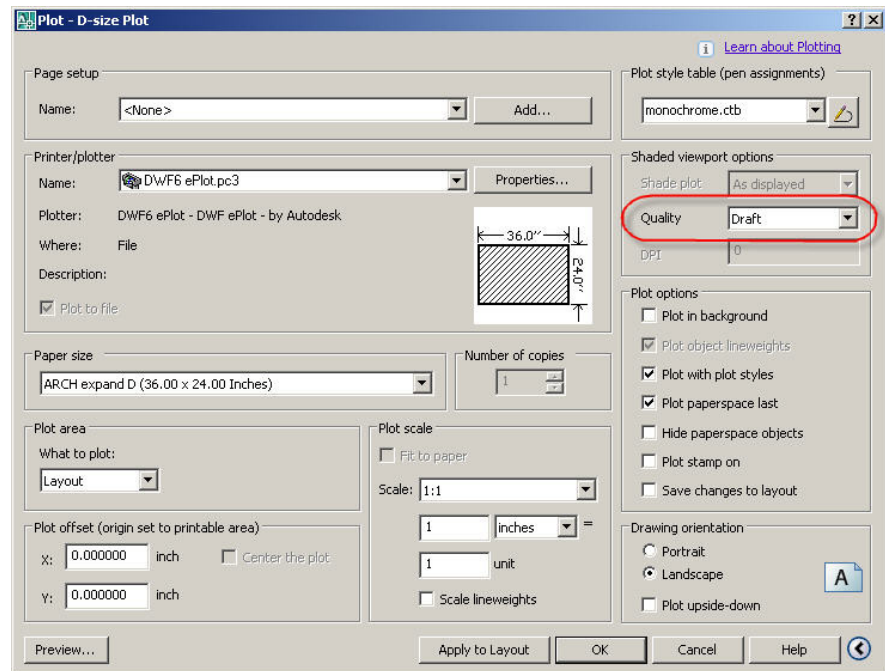
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Using the plot quality setting of “Draft” mode doesn’t always plot what you expect and we’re not just talking quality here. You might even see different graphics missing especially if what you are trying to plot is 3d.

2d Wireframe isn’t the end all here either. It’s sometimes necessary to use 3d Wireframe, Hidden or any of the other Visual Styles to achieve the look you’re after.

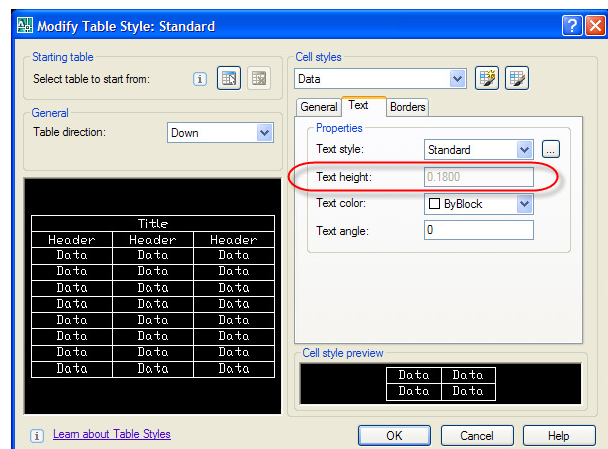
The solution is as easy as modifying the properties of the visual style, or making a new visual style that plots Black & White or however you want your plot to look. Another consideration is that Visual Styles do not get

deployed as easy as copying a copy onto each of your systems. Visual Style information resides in the DWG file. If you are going to use you modified Visual Style on many drawing, you should made the modifications in your template (DWT) drawing. You also drag the visual style to a Tool Palette and deploy the Tool Palette to your other systems. Your users, however, will then need to drag it from the Tool Palette into each drawing they want to use. For this reason, it’s recommended to add your own custom Visual Styles to your drawing template.



Text Height – Wears Many Styles For Different Looks

It bears repeating as most use one or the other method. When you encounter a drawing using a method opposite of what you are use to, you typically have to stop and figure it out what went wrong. The methods, I’m referring to are how you can define the height of the text your dimensions or tables use. All Dimension & Table Styles refer to a Text Style so the Dimension or Table Style knows which font to use. However, if the Text Style the Dimension or Table Style uses has a fixed or specified height, the text height settings in the Dimension and Table Style are ignored. If you want to specify the height of the text a dimension or table using the height setting of the Dimension or Table Style, the height of the Text Style it uses must be set to 0 (zero).

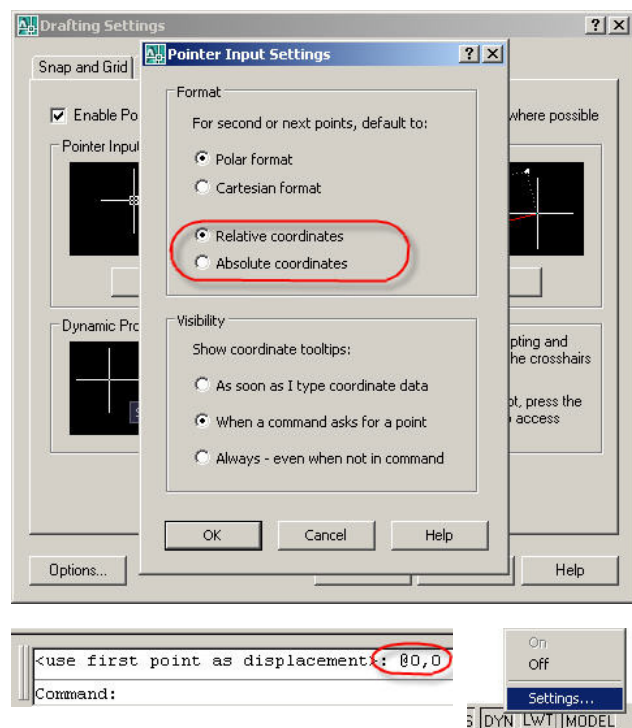


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Typically, you don't run into this being a problem unless someone changes a Text Style's height property from 0 (zero) to a fixed height or from fixed to zero. You might also run into this when your drawing uses Text Styles that have a 0 (zero) height for some and fixed for others and the Text Style the Dimension or Table Style uses is changed. What's nice about Table Styles however, is that the Table Style Manager will disable the height edit box and show it's the actual value that will be applied when the Text Style has a fixed height. This doesn't happen in the Dimension Style Manager, which always has the edit box for the text height enabled regardless of the Text Style's setting. This leads people to believe the value they see will be applied when it may not.

Dynamic Input – When Move Won't Budge

With the introduction of Dynamic Input in AutoCAD, users have a whole new way of entering coordinates into AutoCAD. Unfortunately, this can lead to a lot of head scratching in some cases if you don't fully understand how it works. Try this...draw a circle someplace in AutoCAD. With Dynamic Input turned on, try moving the circle from its center point to the coordinate 0,0 by entering "0,0" (zero comma zero). Unless you or someone else changed it from its default settings, the circle doesn't move. The reason, is that AutoCAD's Dynamic Input defaults to "Relative" data entry, you know, like when you type the "@" at the command line. Relative input assumes the entered coordinate is a distance from where you picked prior. In this case, AutoCAD's assuming you want to move the circle 0 units in both directions from its center point.



You can change this default behavior by right clicking on the Dynamic Input button in AutoCAD's status bar and selecting the Settings option. Then, choose the Settings button from the Pointer Input section of the dialog. Here, you can change the default behavior to Absolute Coordinates instead of Relative. Doing this means that when using Dynamic Input, you'll not need to explicitly enter the "@" to force Relative coordinate entry just like you're always use to from AutoCAD's command line.

But let's think about this for a moment, during most of your drawing, most of your keyboard entry is likely always Relative. By defaulting to Relative Coordinate entry, you eliminate typing "Shift-2" to get the "@". The question then becomes, how to force Absolute Coordinate entry from Dynamic Input? The answer, is by typing the "#" instead of the "@". Doing this tells AutoCAD that you're indeed using Absolute Coordinate entry. This is likely the best option for

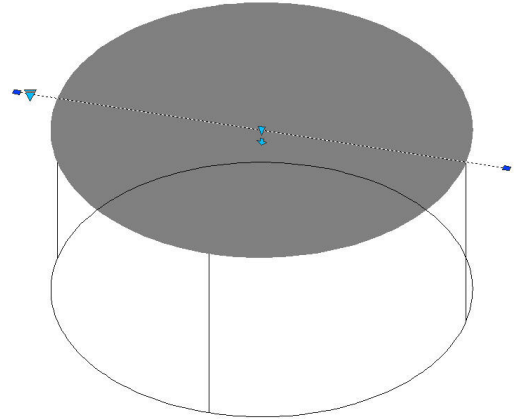


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most drafters because the bulk of their use is Relative. However, if you frequently use Absolute Coordinate entry, like a CNC machine operator or a drafter preparing CAD geometry for CNC operations where machine coordinates are important in relation to geometry, then changing the default settings is possibly a better option. As an alternative, you could also temporarily disable Dynamic Input with the F12 key anytime you need to make Absolute coordinates. One thing to keep in mind, however, is that while the command line accepts the “@” symbol, it does not accept the “#” symbol which can only be used when Dynamic Input is active.

Live Sections – They’ll Leave You Dead

With the major theme of 3d in AutoCAD 2007, one of the great improvements was the addition of Section Plane object. Section Planes will allow you to show a section through geometry and/or generate 2d or 3d geometry and blocks, which represent the section geometry. However, as with anything that seems powerful, you need to respect its use. With that power means your system will need power to make it all work. If you have large, complex solids, or have a lot of solids within the same file, you should limit your use of Section Plane objects. Even if their live sectioning properties are turned off, you might find tomorrow, that the file you worked in all day today, takes a LOT longer to open, possibly even hours. Proceed slowly before pushing any new functionality or even features you’re not familiar with to the limits. You never know when they’ll rise up and bit you. And chances are, it’ll be when you least expect it and there’s a big deadline looming.



Those Pesky Bugs – Can’t Fillet Dot Linetypes

If you run into this it’s likely wasted a little of your time until you finally gave up. There’s a bug in AutoCAD 2007 (fixed in 2008) that prevents you from Filleting and linetype made up of nothing but dots. This includes the default DOT, DOT2, DOTX2, and ACAD_ISO07W100 linetypes. Your workaround, turn the object you are filleting to any other linetype first, and then change it back. For those using AutoCAD Mechanical or Mechanical Desktop, there’s a handy AMFILLET2D command that does not have this limitation.

Wildcard – They’re Up Your Sleeve And You Didn’t Even Know It.

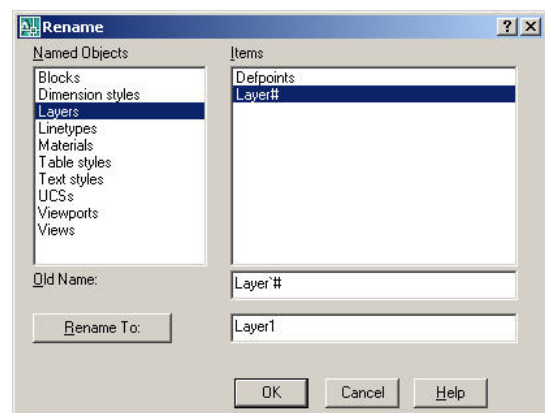
Wildcard are a very useful tool. Whether you’re trying your luck here in Las Vegas or back at home using AutoCAD. The trouble is, you’re more familiar with those used in Vegas than the ones you have in each hand of AutoCAD and the ones in AutoCAD have better odds of helping you than those at the Poker table. Most people are familiar with at least 2 Wildcards, those supported by Windows and DOS. Those are the dollar sign (\$) and the asterisk (*). But did you know, there are 11 of them and you can use them in all sorts of combinations? And something else you might not have known is that they are supported all over AutoCAD, in all sorts of places, just sitting there quietly, waiting for you to use them. Seems off doesn’t it, that there’s so many and there’s seemingly no mention of them in the AutoCAD documentation? Actually,

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unless you're a programmer, they are in the documentation but only someplace where a programmer might go. For a detailed explanation of them, check out the AutoLISP function WCMATCH (stands for Wild Card MATCH) in the AutoLISP Reference. For now, here's a list of the various Wildcard characters that you can use and a brief example or two.

Wildcard	Explication	Example
#	pound - matches any single numeric digit	Floor# would match Floor1, Floor2...etc... Floor## would match Floor11, Floor12...etc...
@	at - any single alphabetic character	@103 would match A103, B103...etc... @@-103 would match AA-103, AB-103...etc...
.	period - any single non alphanumeric character	A.Floor would match A-Floor, A_Floor...etc...
*	asterisk - any character sequence	*1 would match Arch1, Room101, 321...etc...
?	question - any single character	Room? would match Room1, RoomA, Room-..etc...
~	tilde - if 1st character in pattern, matches anything but pattern	~A-Room would match B-Room1, C-Room1...etc... Would not match A-Room1.
[...]	matches any one of characters enclosed	[ABC]-Room would match A-Room, B-Room, & C-Room. Would not match D-Room, E-Room, ...etc...
[~...]	matches anything but one of characters enclosed	[~ABC]-Room would match D-Room, E-Room...etc... Would not match A-Room, B-Room, C-Room
-	hyphen - used inside brackets to specify range for a single character	Part[0-9] would match Part1 thru Part 9. Would not match Part10, Part11,...etc...
,	comma - separates 2 patterns	Room#,Part# would match Room1, Part4...etc...
`	reverse quote - reads next character literally. Used if the character you want to match happens to be a wildcard character also.	Room`# would match Room# Would not match Room1, Room2...etc...

With these Wildcard, you can do a lot of things. When used with the command line version of the Xref command, it would allow you to Bind a series of Xrefs matching the wildcard all at one time. When used in the Rename dialog, can help you change the prefix on several layer or block names all in one operation. Even in the Layer dialog, when you add a new layer, as you are trying a layer name, when you are finished typing the name, type a comma followed by another layer name (e.g. "Floor01,Floor02,Floor03"). The reason this works, is because the Layer dialog expects you to enter a layer name, and the comma wildcard, tells AutoCAD that you are entering multiple because the comma wildcard separates patterns. Also, as with the example in the image to the right, you could not rename "Layer#" without using the reverse quote otherwise Rename thinks your using the # as a wildcard.



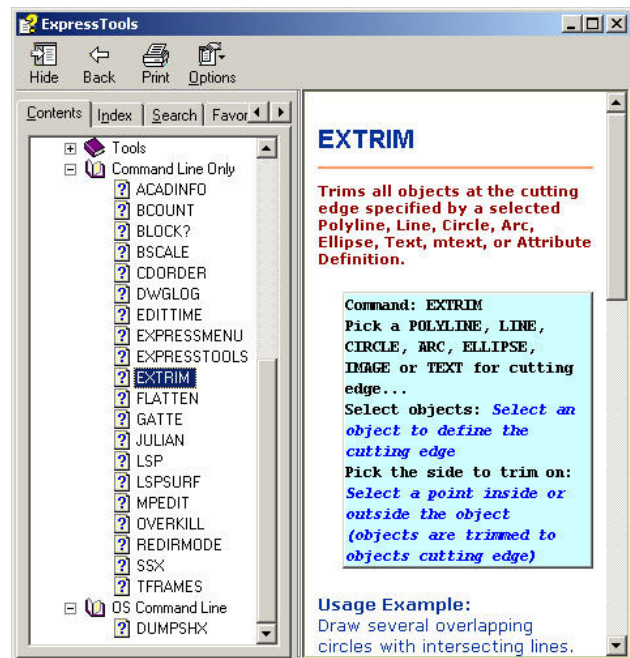
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Sadly, however, some of the newer functionality, like the Reference Manager Palette, doesn't always fully implement or honor these Wildcards. As we pointed out earlier, this information is so hidden, even some of Autodesk's programmers didn't know about it. Despite this, there's a lot of places to work and do wonder.

Standards & Express Tools - There To Do The Dirty Work

Extended Trim – Turns Trim Into A Cookie Cutter

It's been a while since the Express Tools moved from a "Subscription Only" add-on to

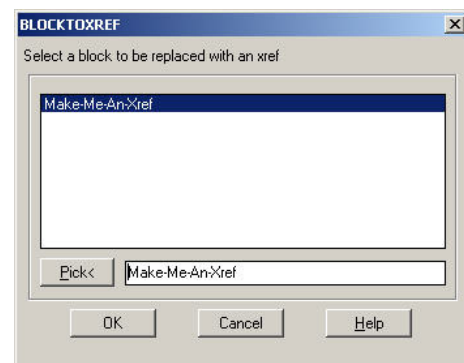


part of core AutoCAD. Many of the items in core AutoCAD now, were once part of the Express Tools, previously called Bonus Utilities. What you may not know is that there's several Express Tools items not listed in the Express pull down menu. Many of these items without a menu link have been rendered less functional by the addition or enhancement of other AutoCAD commands but not all. To find out more about these items, click on Help at the bottom of the Express pull down menu. Look in the "Command Line Only" and "OS Command Line" sections for a list.

One such command is the EXTRIM or Extended Trim command otherwise known as a Cookie Cutter Trim. It does just as its name implies, trims all objects that cross a perimeter. This makes pulling a section of geometry from a larger area and cleaning it up a snap.

Block to Xref – Add Attributes To Your External Reference

Here's a little trick you don't often hear about. AutoCAD's drawing database supports attributes with Xrefs. AutoCAD was never designed with this in mind, but its design does allow it to happen. For Blocks to support attributes that can have independent values of other instances of the same block, each insertion contains sub-entities to handle the attribute values. An Xref is handled by AutoCAD the same as a block. In fact, if you use the List command, it lists it as a "Block Reference". So, because Block References can contain Attribute sub-entities, Xrefs therefore can contain Attributes and live happily within the DWG database. If you've ever wished you could have the power of Attributed Blocks and Xrefs together, now you can.



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The trouble is, AutoCAD's Xref command doesn't have any code to add these attribute sub-entities so when you attach an Xref with AutoCAD's standard Xref tools and the file you are trying to attach has attributes defined within it, the get stripped out. To make an Attributed Xref, you'll first need to insert the file as a regular Block. Once you've inserted it, use the Express Tools "Convert Block to Xref" command to convert it to an Xref.

One draw back to using Attributed Xrefs, is that most of the programmers at Autodesk weren't aware that this is possible and as such, built in code to a lot of the newer Block tools within AutoCAD to automatically reject Xrefs. This means commands like Block Attribute Manager (BATTMAN command) will filter this out and not work. Likewise, double clicking on the block...um...I mean Xref...will not launch the Attribute editor. However, the Enhanced Attribute Editor (EATTEDIT command) despite not being called automatically during a double-click, as well as the old Dialog Attribute Editor (DDATTE command) both support editing of the attributes attached to the Xref. The Properties Palette also gives you easy access to the attributes value so this limitation is more than easy to live with.

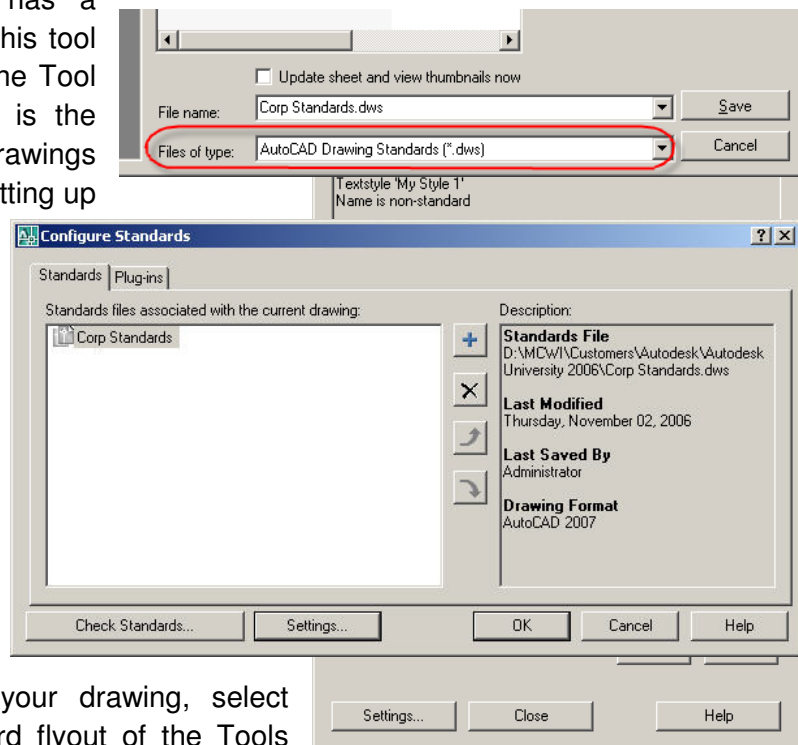
Standards Checker – Helping You Tidy Up Your Drawing

The CAD Standards Checker has a couple good uses. You can find this tool in the CAD Standards menu of the Tool pull down menu. The first use, is the most obvious, to check your drawings standards. You do this, first, by setting up

a standards file for this program to use. And to do that, you just create a drawing, and setup Layers, Dimension Styles, Linetypes and Text Styles that you want to define as your "Standard". Once you've finished, use the Saveas command and change the File Of Type drop down to AutoCAD Drawing Standards (DWS). Now, anytime you want to have a

standards file associated with your drawing, select Configure from the CAD Standard flyout of the Tools pull down menu to add a reference to your standards file. You can also reference a standards file from your drawing Template (DWT). From here on out, use the Check option of the CAD Standards menu to check your drawing against the standards you've selected.

You can use the Settings button from the Configure or Check Standards programs to adjust how the CAD Standards Checker operates. For instance you can tell AutoCAD to automatically notify you with a bubble or icon in the system tray any time something happens in your drawing that



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makes it violate the standards. You can also tell it to try to automatically fix errors and/or ignore certain issues that you've flagged as known violations that perhaps you might need to let slide and ignore.

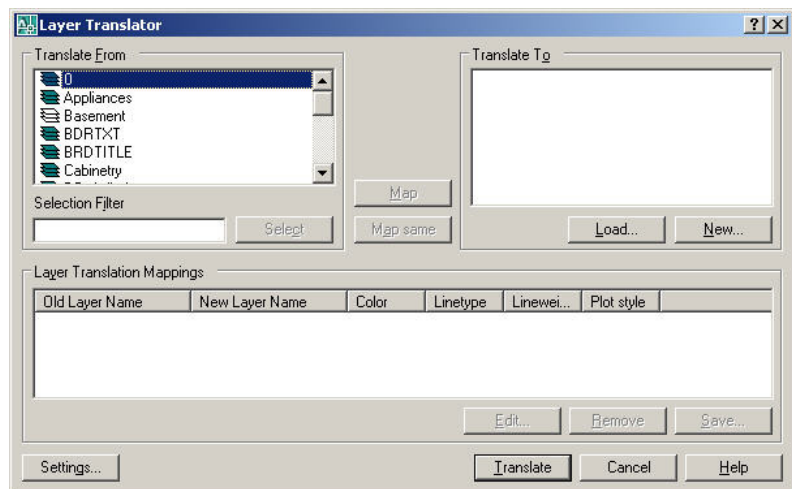
While most people think of the CAD Standards functionality in terms of keeping and/or verifying that their drawings remain in compliance with corporate CAD standards, the second use is something you might actually find more useful. If you've ever dealt with customer or vendor drawings, they almost certainly aren't to your standards. You can use the CAD Standards functionality to convert drawings to your standards. Even if you don't have and set standards, you can define settings on the fly, save them in a Standards file (DWS) then use that in the vendor/customer drawing with the CAD Standards Checker to convert your vendors or customer's standards (or lack there of) to something your prefer to work with.

One of the nice things about the CAD Standards checker is that it tells you what's different between what's found in the file being checked, and the standards file you defined. If you run into something unexpected, that you hadn't addressed in your standards file, you can always and edit the DWS file to handle this and rerun the Standards Checker. It might also be a good idea to have several, smaller standards files, each with less settings. You can attach several standards file to a drawing, and by using several smaller DWS files, you can more easily define the settings you want in your new file. A DWS file can even have as few as 1 layer or text style, or dim style, or linetype defined. It might take a little planning and though, but the CAD Standards Checker is well worth you time and effort to explore.

Layer Translator – Helping You Tidy Up Your Drawing

If the CAD Standards Checker doesn't give you the flexibility you need with Layers, you can use the Layer Translator. It's right next to the Check and Configure Standards commands on the CAD Standard flyout of the Tools pull down menu. Using the Layer Translator, you can quickly and easily translate one set of layer settings to another.

Using the Settings button, you can configure exactly how the Layer Translator works. Take special notice of the "Show Layer Contents When Selected" toggle in the settings. With this toggle checked, the contents of the layers you select if reflected in AutoCAD. This mimics the Laywalk (Layer Walk) command but has one bonus. The Layer Translator displays different icons next to the layer name to indicate if it's in use or not. Layer Walk (Laywalk command) on the other hand, does not. Now when you preview a layer and you



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don't see anything, you don't have to worry that something might be there but that it's too small to see, hiding behind your dialog box, or is off the screen.

Another nice feature of the Layer Translator is that you can load a list of layers from and DWG (drawing), DWT (template) or DWS (standard) file you may have as well as define new layers on the fly. Once you've defined layer mappings, you can save those mappings to a DWS to DWG file for layer use. Very handy if you get files routinely from a customer or vendor that you always need to translate layers from their standards to yours.

I Knew That! - ...But Never Thought Of It That Way Before...

RefEdit? – Sometimes It's Best To Forget It!

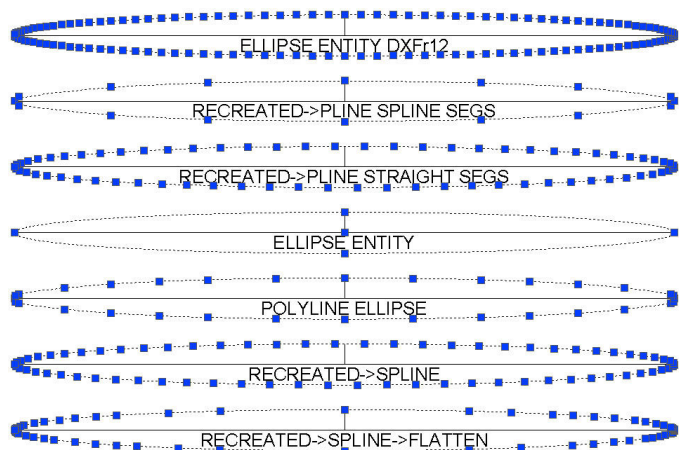
Most AutoCAD users are familiar with the RefEdit command. It's the command that allows you to edit in place, an Xref or Block insertion without binding and/or exploding it. It's a very useful tool. However, there's times when RefEdit just doesn't cut it. When you use RefEdit, you edit a scaled instance of the Xref or Block. That is, if the Xref was attached or the Block inserted at a scale factor other than 1, you edit the Xref or Block at the scaled instance, not in the base units the Xref or Block was created in.

If editing at the Xref or Block in the base scale (scale factor of 1) you have a couple options. For Xrefs, you can open the Xref's file and edit and save it. Upon returning to the parent drawing that references it, simply reload the Xref to make your changes visible. While this is nothing new, many don't know there's the Xopen command that will prompt you to select an Xref and automatically open it which makes this process a lot quicker.

That takes care of Xrefs, what about blocks? For Blocks, you can use the new Bedit (Block Edit) command that was introduced with the Dynamic Block feature of AutoCAD. While designed for authoring your own Dynamic Blocks, you don't have to use or make Dynamic Blocks to use the Bedit command. When you edit a block in the Block Editor with the Bedit command, you are editing the block at its 1-1 scale, not it's scaled instance like RefEdit does. There's Model Space (Model tab), there's Paper Space (Layout tabs), think of the Block Editor and the Bedit command as "Block Space".

Don't Just Get Fed Up – Really Explode!

There's times when AutoCAD is just too complex for its own good. Sometimes, you just have to have simpler geometry. This could be because the more complex objects don't support the editing methods you need, or perhaps you are going to export the file to a different software package that doesn't support them. It might even be that you want better control





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over the number of the more basic entities that you end up with to maximize efficiency in CNC machining. Whatever the reason, there's a lot of options for you to consider.

Take for example, the graphic earlier. The center object is an actual Ellipse object in AutoCAD. There's not a lot of editing you can do with ellipses but they are the most mathematically true geometry. If you change the PELLIPSE system variable to generate a Polyline representation of an Ellipse, and overlay the two, you can see if you zoom in close that the two don't match completely. The error is greater the greater the ratio is between the 2 axis of the Ellipse are. There's also an example that was broken down by the old method of using a release 12 DXF which creates a lot of small line segments. If you use the Pedit command on a Polyline ellipse, you can use the Decurve or Spline options to generate yet different results.

Now let's look at the very bottom 2 example. The top of which is a Spline object that was created from an Ellipse. This was done by offsetting the ellipse which creates a spline. You can then, erase the original ellipse, and offset the Spline Ellipse back to it's original position. Looking at the Ellipse that was recreated by a Spline, its geometry is quite accurate, but its not very editable, just like the Ellipse, and it has a lot of small segments much like some of the other options. At this point, this recreated Spline object doesn't give us anything much better....or does it? The very bottom example was the same Spline, but then processed further with the Flatten Express Tools command and entering No to the Remove Hidden Lines prompt. In this case, you have a very accurate Ellipse, in the form of a Polyline. It uses as large an arc segment as it can and smaller ones where it needs them to maintain its accuracy. This yields the least number of segments with the most accuracy possible.

So what else does this tell us exercise tell us? It tells us that if we need to break down Spline objects, one of the most accurate ways with the least number of resulting segments is the Express Tools Flatten command choosing to not remove hidden lines when prompted. Watch your results, however. If the Spline object is "almost" straight and all the points are in the same Z-axis, Flatten could potentially turn it into a Line when you really need Arcs. If this happens, simply perform an Undo back to where you had the Spline. Using the Properties Palette, change some of the Control Points to have various Z-coordinates. You don't need to change them all, just a few. When you then run Flatten again, it will retain its Arc segments in the form of a Polyline.

While all this might seems trivial to some, the extra short segments can significantly increase file size, and even processing time for things like plotting or CNC machining. One such file I've seen CNC Water Jet run time reduced from over 30 hours to less than 20 by doing nothing more than reducing the number of segments the machine needed to process.

Can't Explode Or Flatten? – Don't Fret, Just Think A Little Bit

There are times when Explode or Flatten just won't work. There are still other options but there's no single answer. In cases such as these, you have to think about all the tools AutoCAD has and how they work. Exploding Text into lines for example has many options. For starters, there's the Explode Text Express Tool. That doesn't give you the results you've looking for, try



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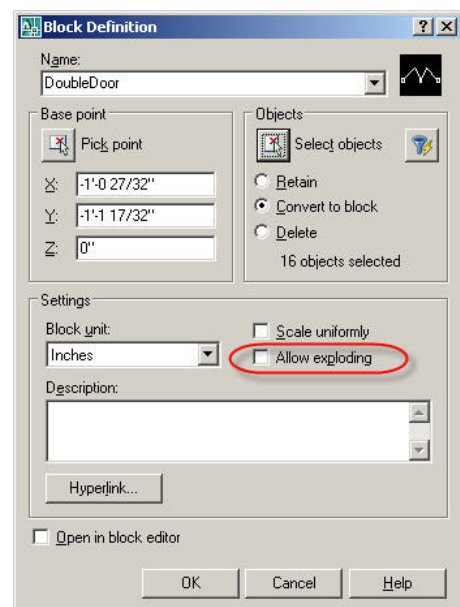
plotting to a DXB file and re-importing it. DXB files don't handle accuracy well over large areas, so if accuracy is what you need, try scaling up (yes, up) the geometry, making several small DXB plots, piece them back together with the DXB Import and rescale back to their original size. You can also plot to a plot file with most any HPGL printer driver and re-import the PLT file using the PLT To DWG Express Tool. Yet another method is to export the Text using WMFOUT into a Windows Metafile (WMF). You then can use WMFIN to re-import the file, and explode the WMF Block Reference. If the Text is using a True Type Font (TTF) instead of an SHX based font, you'll need to mirror the text first before exporting it to a WMF file. Just make sure the MIRRTEXT system variable set to 1 (one) so the Text actually mirrors. This needs to be done because WMF file support True Type Text and by Mirroring it, it gets exploded down instead of being retained as a Text object. Using the WMF method, you'll most likely need to rescale and possible clean up a little bit but it might just give you the results you are looking for.

Things like complex linetypes can often be broken down with some of these same methods as well. You might even run into situations where you need to use one method for Text and another for the Complex linetypes and merge the two results to get the best results possible. Do you have access to other software like Corel Draw, Adobe Illustrator or other CAD/CAM package? Try exporting from AutoCAD into these packages. From there, export the data again, then re-import into AutoCAD. You'll often get different result because each software translates objects a little differently. You might even try exploding or other processes in these packages before exporting them back to AutoCAD. Try even the slightest variations in your process because sometimes the slightest change can make the different between poor geometry, good geometry needing some cleanup, and the perfect results you are looking for.

Other things like 3d objects can often be flattened with tools the Express Tools Flatten command, the new Flatshot command in AutoCAD 2007, Section Plane objects by generating the geometry from the object (another AutoCAD 2007 addition) or even the old Solview, Soldraw and Solprof commands that have been around for a long time. Have a polyline with a width that you want to be 2 profiles representing the edge of the width? Offsetting isn't your only option, it doesn't take long to setup an Multi-Line (Mline) Style and trace over it ands explode the Mline. The point is, there is no right or wrong answer, just many answers with different results. Mix and match any of the ideas you can think of and check the results. There's likely one "Right" answer for any given situation. In time, you'll be better any knowing what situations and what end results you need and which "Right" answer(s) will get you there.

Unable To Explode Blocks? – Refedit To The Rescue

There's an option when creating a block that you can uncheck, which defines a block in such a way that AutoCAD will not explode it. The trouble is, is that sometimes you still might need to. If it's a simple matter of changing the Block's definition, Refedit (Reference Edit)



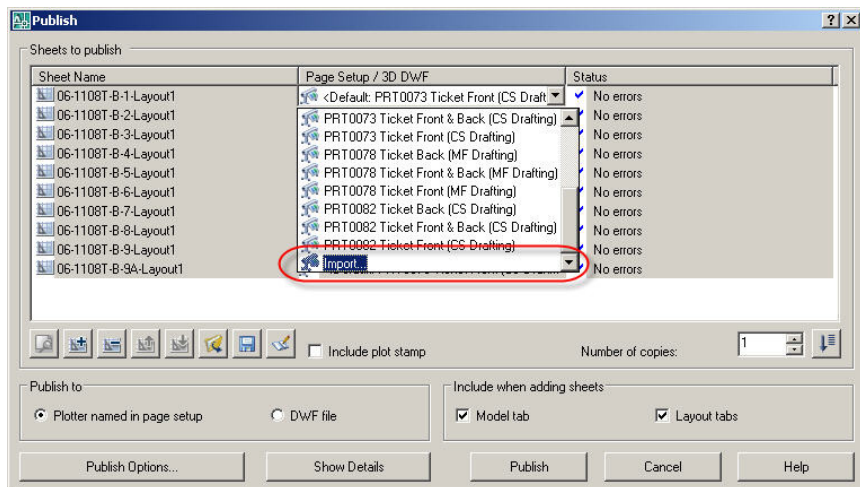
Stop Wrestling and Start Dancing with AutoCAD® and Bedit (Block Editor) will work. But what if you need the geometry to modify elsewhere and not change the original block definition? You could always use the Wblock command to write the block to a separate file, open that file, then copy the geometry to the Windows Clipboard, go back to your original file, then Paste it in. You don't have to be a rocket scientist to see that's a convoluted cumbersome process.

For a quicker option use something you're already familiar with and that's the Refedit command. With Refedit, select the block that you'd like to explode or extract geometry from. Once Refedit becomes active, make a copy of the geometry (all or some) that you want. Then, use Refedit's functionality to remove your copied geometry from Refedit's working set. Then, simply close Refedit canceling any changes to the block. You'll have your original Block (not exploded) and have a copy of the individual geometry that made it up. If you don't want the block, simply delete it and move your individual geometry where you want it.

If your drawing doesn't contain multiple instances of the Block that you need to keep, you don't even have to make copies of the geometry in Refedit. Simply remove all the geometry from the working set. You will, however want to create one new piece of geometry that will remain in the Block's definition. Now, instead of canceling Refedit, choose to Save the changes in Refedit. The geometry you want to keep will be outside the Block's definition; in the exact location it was prior. And your block, not has some geometry, do you don't end up with an empty Block definition making it more difficult to erase. This variation will update all other instances of the block so only use it if you don't have multiple insertions of the block that you want to keep. Otherwise, use the previous method discussed earlier.

Batch Printing On The Fly Make You Sigh? – It's As Easy As Pie

Since the Publish command was introduced, it's been a great tool for batch printing drawings. The trouble is that it requires a Named Page setup to work. That's not a big deal for most people as they've usually defined some Named Page Setups for their regularly printed to printers. But what about when you need to batch print to someplace different, or to a file like a DWF or PDF? If there's no Named Page Setup for these situations, many users have felt the pain.



But it doesn't have to be that way. It's very quick and easy to make a new Named Page Setup on the fly. Simply open just one of your drawings that you want to batch print, and create a new, temporary named page setup that reflects the settings you want to use. When finished, use the

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Saveas command and save it somewhere temporary with a new name. Once this is done, you can then run the Publish command and import the Named Page setup for all your drawings from the temporary file you just saved. You typically do NOT want to import a Named Page Setup from a file that's included in the list of files you want to print. Doing so will cause that file to not print, because it tries to open that temporary file so that it can import the Named Page Setup, but it can't because the file is already opened by the Publish command. This is why saving to a temporary file helps.

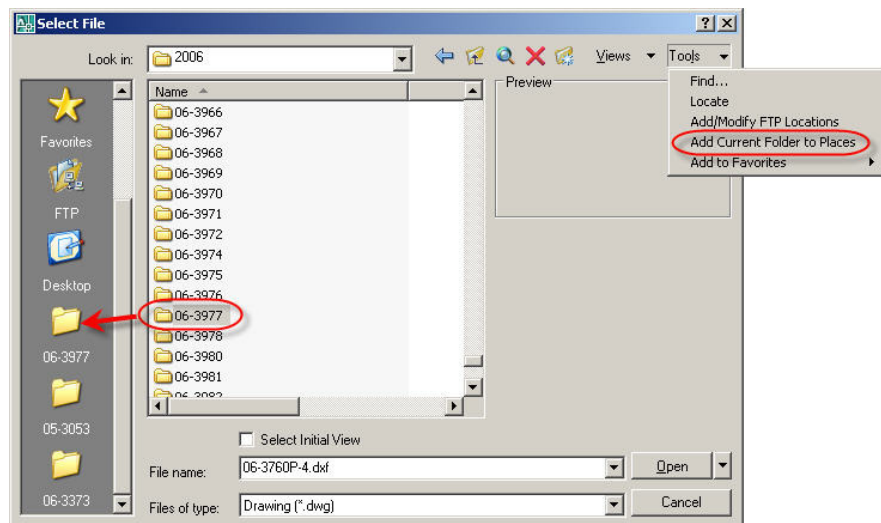
Smart Planning - Up-Front Thinking Saves In The Long Run

Single Object Selection – Not As Difficult As It Appears

Here's a change that happened in AutoCAD 2007 and there's a good chance very few noticed. Object Cycling has been a method for single object selection when multiple objects all reside in close proximity to one another. In AutoCAD 2006 and prior, Object Cycling was accomplished by holding down the Control Key and picking with your mouse button. Each double pick cycled selection to the next object. With AutoCAD 2007, things changed. AutoCAD now uses the Shift key followed by the pressing the Spacebar repeatedly to cycle through objects which are close to one another. The Control key, when pressed, now allows you to select the face and edges of 3d Solids. You can cycle through overlapping faces and edges the same way only holding the Control key and repeatedly pressing the Spacebar. Now when you hold the Control key while picking with your Left mouse button (which performed Object Cycling in AutoCAD 2006 and prior), you select sub-objects of 3d Solids like Edges and Faces.

Is AutoCAD Putting You In Your Place? – That Can Be A Good Thing

Here's something you'll find either people use, or don't and most don't. It's the Places list of your file dialogs. To take advantage of this, all you need to do is drag a folder to the places list. That's it. You can also use the Tools pull down menu and select the Add Current Folder To Places option but that requires you to be in the folder you want in



Places and it's not as fast and easy. To remove them, just right click on the Places shortcut and select Remove.

Selection Sets – There’s a Lot To Select From

I’ve watched people spend forever and a day selecting objects one by one. Zoom pick zoom, pan, zoom, pick, repeat. Other times the select by Window and/or Crossing, getting something other than the desired results and canceling out of their selection and starting over. But did you know, there’s 18 different options to the familiar “Select Objects” prompt? Here’s a quick rundown...

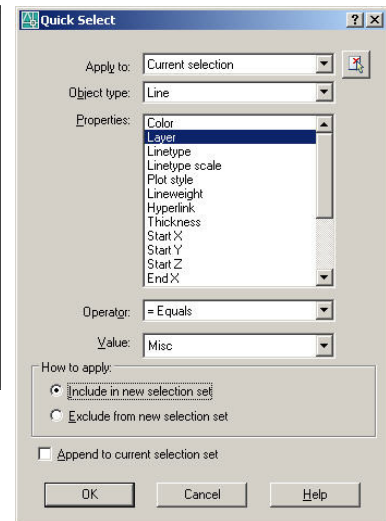
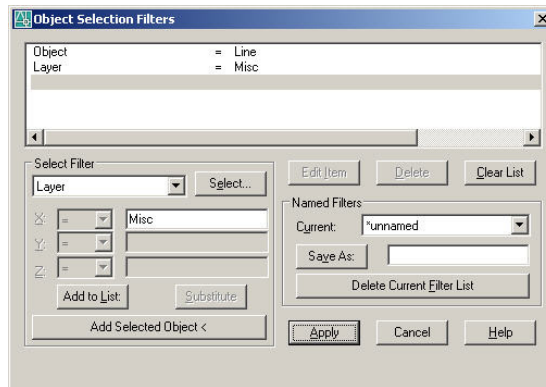
Selection Option	Description
Window	Forces Window selection (all objects completely reside within the window frame) regardless of which way you drag your cursor when selecting points.
Last	Selects the most recently created visible object.
Crossing	Forces a Crossing selection (all objects completely within or crossing into the windows frame) regardless of which way you drag your cursor when selecting points.
BOX	Default - This is the same as the implied Window/Crossing you’re typically used to. Pick points from right-to-left for a Crossing selection, or left-to-right for a Window selection.
ALL	Selects all objects on thawed layers
Fence	Selects all objects which cross the fence line
WPolygon	Performs a Window selection of a polygonal area
CPolygon	Performs a Crossing selection of a polygonal area
Group	Selects all objects within a named group
Add	Default – Modal – Additional selections are Added to the selection set
Remove	Modal - Additional selections are Removed from the selection set
Multiple	Used to select complex objects without highlighting them until all your selections are made. Used to increase performance during selection of complex objects.
Previous	Selects the same objects that were in the previous selection set
Undo	Cancel selection of the object(s) most recently added/removed from the selection set
AUto	Default – Switches to automatic selection where pointing to an object selects that object, pointing to a blank area initiates a box (crossing/window) selection.
SIngle	Switches to single selection mode where no further prompts for more objects are provided.
SUbject	Allows selection of faces, edges or other parts of 3d Solids or composite solids. You may need (but not always) to display the Solid’s History to use this option. This is the same as selection when holding down the Control key.
Object	Ends the ability to select Sub-Objects

If you run some vertical applications, you may have even more options. It use to be that users knew these options because each “Select Object” prompt listed them all. As AutoCAD added more options though the years, there were simply too many to list. As a result, most users are unaware of most of these options but they should be.

If you’ve ever wanted to pick from left to right for a Crossing or right-to-left for a Window, simply type “C” or “W” at the Select Objects prompt to force those selection modes. Another less used method is using the Remove option. Sometimes, instead of selecting what you want, it’s easier to select everything and remove what you don’t. Mixed with other methods, this can be a very useful tool. Lastly, let’s not forget about the Select command. If you ever need to make a selection set in a couple phase, you can easily create selection sets without doing anything to

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them by using this command. You can then make a second selection set, and while building that, add or remove the previous selection speeding up your overall process.



Selection Filters

– A More Refined Selection Process

If you need more control over your selection sets, such as by properties for the items you are selecting, then selection filters might be just the thing you are after. There are 2 basic methods for doing this. One is the Qselect command otherwise known as Quick Select.

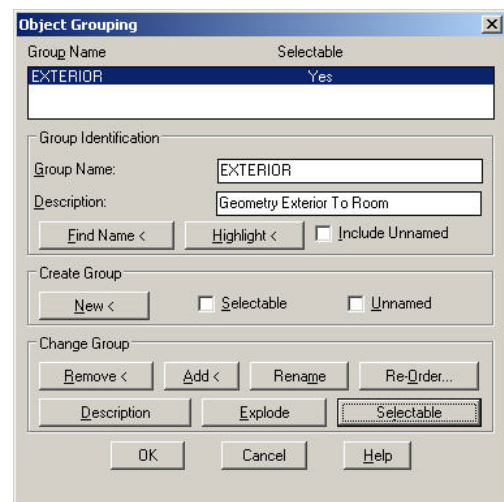
It's the newest and most user friendly way.

The other way is with the old Filter utility. The filter command is more flexible and powerful but it's less intuitive and most likely overkill for most of your needs. Qselect is also available from places like the Properties Palette. Otherwise, you can just type Qselect at the command line to call up the dialog box. When finished, you can use the Previous selection method to select what you'd previously grabbed with Quick Select.

The Filter command was originally based on AutoLISP and has since moved more into the code programming code stream. It's interface isn't as fresh and new but it's just as functional. You can use the Filter command by itself to create a selection set you can use later by accessing it with the Previous object selection method, or you can use it transparently within a command by preceding it with an apostrophe (e.g. 'Filter).

Groups – When Nothing Else Will Do

If all the selection methods and filtering methods don't offer much help in making your selections quick and easy, you still have options. Sometimes you need to take the extra time to select the object you want. For this reason, there's the Group command. The Group command offers you a way to group objects, possibly that can't be easily selected or filtered by properties. You can create groups that use names or groups that are nameless (unnamed groups). You can also toggle if the group is Selectable or not. If Selectable, selecting one object within the group selects the entire group as long





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as the Pickstyle System Variable is set to 1 (one). If set to 0 (zero), selecting an object within the Group selects only that object.

You can quickly and easily toggle the Pickstyle system variable if you frequently need to select the group as a whole, as well as just individual objects. You can also use the Group object selection method at the Select Object prompt to select a group regardless of the Pickstyle System Variable setting.

It's important to note that if you copy a Group, the resulting new geometry is not in the same group, it's in a new, unnamed Group. You can then rename that Group if you like. If you copy an item from a Group, the new copy is not in any group, not even an unnamed group. Lastly, if you copy all items in a Group individually, it's the same as copying the group, a new, unnamed Group will result for the new geometry. The Group dialog box offers full flexibility of adding, removing or exploding groups.

It's also important to note that your drawing performance can diminish if you become too full of unnamed Groups from copying operations. You may not be aware the unnamed Groups are being created so it's best to use them only when you need or to convert them to Un-selectable groups when you don't need them but may want to retain that information for later use. Un-selectable Groups do not create new groups (named or unnamed) when their objects (even if all of them) are copied.

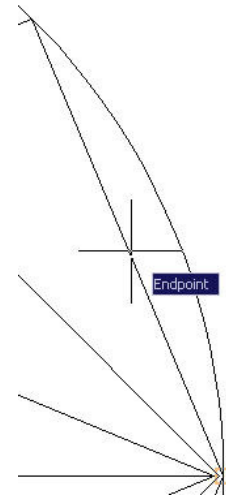
Script Files – Predictably Dumb

If you haven't used Script files, you might want to look into them. They are a very simple and easy way to automate AutoCAD. A script file is nothing more than an ASCII text file list of commands as AutoCAD would accept them from the command line. It's as its name implies, a "Command "Script". The reason a lot of people shy away from Script files, is that they offer no intelligence. If different situations in AutoCAD present different input needs, your Script will error out. A good trick to solving this is to put any operations that involve "variable" input into a Lisp program and call that from the Script file. But most of us are not AutoLISP programmers and don't have the time to become an efficient one. For that reason, you can do little things in your Script file to make it more predictable.

A good example of this is the prompt you sometimes get when you try to close a drawing. If the drawing has not had any changes since it was last saved, it will close, otherwise it will prompt if you want to save or discard those changes. Sometimes in a Script file, you'll run into such situations and the Script file can't handle it. Or can it? In these cases, you can add commands to your Script to make the unpredictability go away. If for each drawing you performed a Zoom by 0.99%, your drawing will always be changed. You can then reliably answer Yes or No to that prompt. Or better yet, if it doesn't matter if the DWG files are saved, just always perform a QSAVE just prior to close. Need to set a layer current that may or may not already exist? Use the Layer command's Make option to Make the layer if it doesn't already exist and set it current. If it does already exist, it just sets it current. Using tricks and techniques like these can make your Script file usage a lot less problematic making it more beneficial for you.

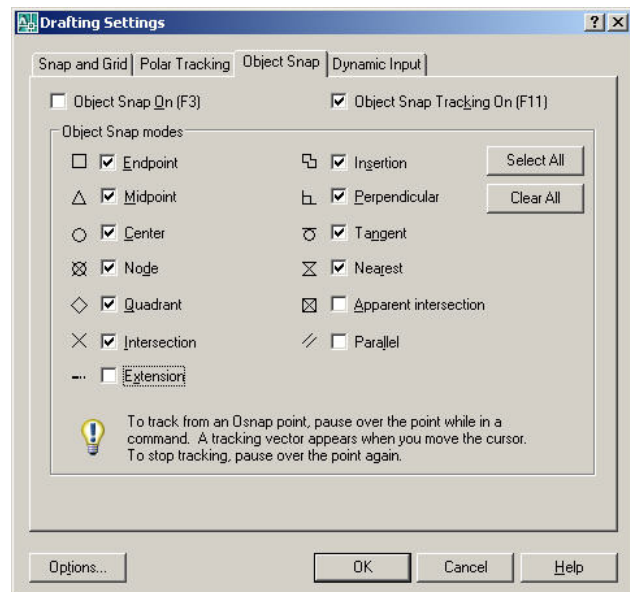
Osnaps – Let Them Do The Work

Have you ever watched anyone drawing in AutoCAD? Give it a try sometime. You get a whole different perspective on productivity. For example, take Osnaps and the image to the right. It should be obvious yet it goes against human nature. If you want to select the Endpoint of a Line, you pick the Endpoint. After all, who would think to *NOT* pick the Endpoint to get the Endpoint? This scenario plays out all too often with AutoCAD users. Looking closely, at the image, you'll see that the Endpoint of the Line is in deed ready to be selected but that the cursor is nowhere near the Endpoint; it's just barely one side of the Midpoint. Even if you have both Midpoint and Endpoint Object Snaps running, you still only need to pick just to one side of the point half way between the Midpoint and Endpoint. Watch your and your staff's use of Object Snaps. Changes are good there's room for improvement and it'll reduce your selection of the wrong line as well as zooming and panning.



Osnaps Part 2 – ...But Don't Get Too Carried Away

Yet another Osnap misstep is running too many Object Snaps at the same time. Does your Object Snap Settings look like the image to the right? Are many if not all selected? This is a sure fire way to loose productivity and accuracy. This is especially true when the Nearest Object Snap is set because it can grab anything anywhere and you might not know it.



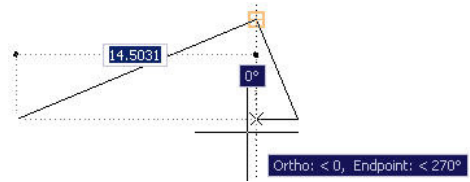
Having too many running Object Snaps also leads to AutoCAD users fighting the Snap selection until the right one applies. I've watched some users move their mouse for 30 seconds trying to get the correct Osnap to take before giving up and Zooming in. This issue combined with the behavior previously discussed above can really be unproductive. In fact, having too many running Object Snaps can often force users to pick close to the point they need because there's so many running object snaps, doing anything else would cause the wrong snap to apply.

Besides, at any selection prompt, you can type a few letters or use the right click menu to force an object snap that you need very efficiently. There's really no need to have that many turned on at one time.



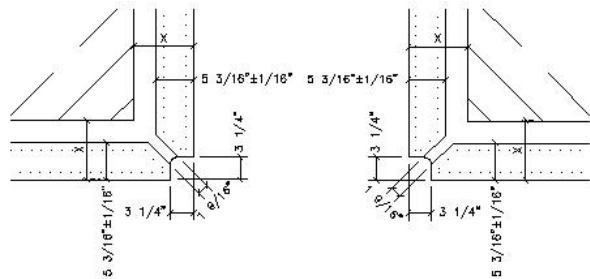
Otrack & Point Filters – Selecting Use Of Object Snaps

Yet another of the least used features in AutoCAD is Object Tracking. As you watch AutoCAD users draw, you'll see many of them jump through hoops with the editing and creation of construction geometry all in an effort to be able to Snap to a particular point. While construction geometry is useful and many times under used, thought should be taken as to when it's helpful and harmful to your productivity. If Object Snaps don't give you the point selection you need, using Point Filters (e.g. .X/.Y/.Z) to Snap to parts of coordinates building your point can often be just as if not more effective. And there's also Object Tracking, where with this feature a brief pause of the cursor and you can acquire much of what you need all without the typing or right-click menu selection of Point Filters.



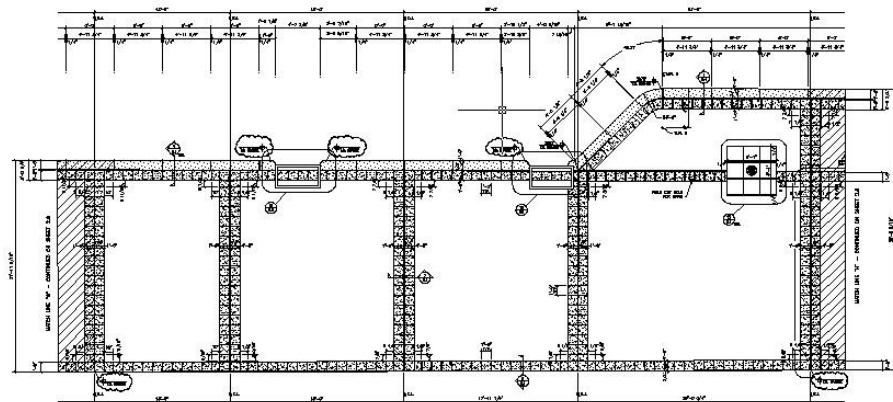
Mirrored & Rotated Geometry

Another area you can increase efficiency is using Paperspace (Layouts) differently or even using it at all if you're still using only Modelspace.. Take, for example, the image to the right. A lot of AutoCAD users will make copies of geometry for mirrored or rotated examples. This not only increases the size of your of your drawing making it perform more slowly, but you lose efficiency by creating what's essentially duplicate geometry. The image on the right only was created from a drawing that only contained geometry for 1 (one) of the 2 conditions. The other was created with a Viewport that was looking at the back side of the geometry, creating a mirrored effect. You could also rotate (either alone or in addition to the mirror) the Viewport as well for other orientations. And with fully associative dimensions, you can copy the original Viewport, dimensions and all. All you then need to do is adjust the Viewport's viewing angle and correct the placement of some of the Dimensions. And future changes, will update both automatically.



Dimension & Annotation Space – Paper or Model?

Among the many questions I'm often asked, is if dimension and annotation objects should be placed in Paperspace or if they go in Modelspace. The answer is that they go where they belong. There is no right or wrong answer and these



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types of objects have a lot of definite advantages and drawbacks in both places.

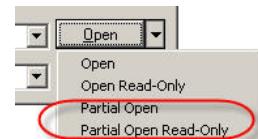
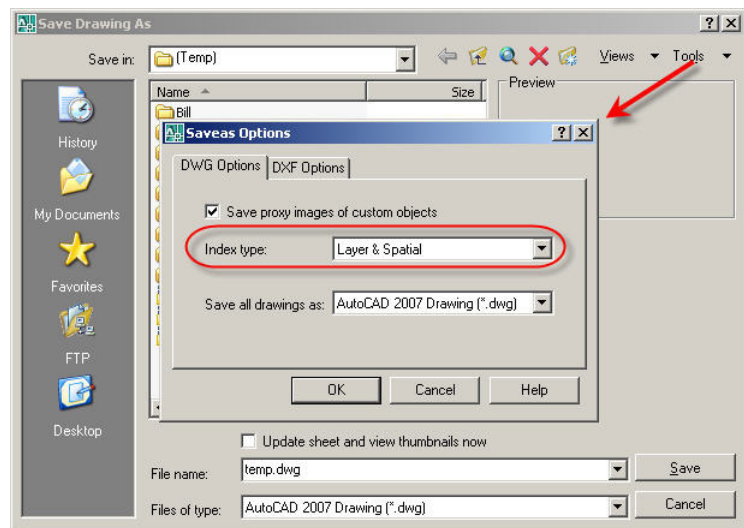
If you place notes, symbols, and dimensions, in Modelspace, they'll almost never be out of line with your geometry because you see how changes to your geometry affect them and you can adjust accordingly. With the true associative dimensioning, this is less important for dimensions because if they are associative, they will update. Notes and other symbols on the other hand do not. But putting them in Paperspace does get them out of the way of your geometry making it easier to create and edit.

In the previous example of a mirrored Viewport, dimensions in Paperspace meant we could copy them without re-dimensioning the geometry. If they were in Model, your mirrored Viewport would show the dimensions from the back side without toggling layers on a per Viewport basis and dimensioning twice, once from each side of the geometry. On the other hand, in the example above, the top string of dimensions does not extend all the way down to the geometry it's dimensioning. Placing these dimensions in Paperspace means they're values won't be correct because they're not associated to the model geometry. Also, this section of streetscape extends much longer than is shown and is carried onto the next several sheets. Placing the horizontal dimensions in Model means a lot more steam lined dimensioning process. The vertical dimensions on the other hand, are placed in Paperspace. That means they won't overlap geometry under them which makes reading this drawing much easier. In this case, dimensions were placed in both Paperspace and Modelspace.

In a nutshell, there is no proper place for your dimensions and annotations. They belong where ever they make the most sense to give you the results and efficiency you need. While dictating their use in one location or another simplifies your drawings, there will be places where a one-size-fits-all approach will limit your efficiency.

Drawing Structure – How You Draw Can Affect Your Productivity

This last section is both a new topic and a summary of everything that was previously covered. It's about drawing structure. There's no one idea or topic you should implement, nothing you aren't doing that you should, no one-size-all approach. Instead, it's about getting you to think about what you and your particular company does, and how you do it. Do I use block? Should I use Blocks? Am I using them the most effective way? Should they be Xref's instead? How do I name them? Do they each go on a separate layer? If things change, what will change and how will it be the easiest to implement





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that change? Will my drawing get too big and slow down? How can I structure it to make it less large and keep its efficiency up?

All these are good and important questions to ask yourself so you can review what you do each day. As an example, if you use Xrefs, you may have heard of demand loading of Xrefs which is

set in the Options dialog box or by using the XLOADCTL System Variable. Maybe you've already set this setting because the Help file says it increases performance when loading Xrefs. But did you know that this setting does nothing? Nothing, that is, unless the file you are Xref'ing is saved with a Layer and/or Spatial index. A Layer and/or Spatial index adds a little extra size to your drawing, but when used as a demand loaded Xrefs, it gives it the added information it needs to keep from loading geometry that's outside the coordinate (spatial index) area or on a different layer (layer index) that the parent drawing does not need. This makes loading your Xrefs faster which makes loading the parent drawing faster. Saving files with a Layer and/or Spatial index also makes partially opened files load and open faster.

Maybe Xref's aren't the way to go. But are you using blocks where you should? Will you need to update geometry that's duplicated elsewhere in your drawing? If you're building things with 3d Solids, are all your solids in 1 (one) or 2 DWGs or is 1 (one) Solid in 1 (one) file? Are the solids all on 1(one) or 2 layers? Or are they each on their own layer? How are the layers named? Will you be able to filter them efficiently so you can manage dozens if not hundreds of layers quickly? Is there 1 (one) layout or sheet in a file or multiple? The next time you're drawing, consider these and other questions and ponder how they could affect your productivity on the overall scale of your project. You really might be surprised how many choices you make on your next project and the performance and efficiency implications it can have.