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AUGIWorld

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TIPS & TRICKS FOR USING AUTOCAD 2006

*Lynn Allen dives into
AutoCAD 2006 with
some great tips for
boosting your productivity*

Also in this issue

- Four simple rules for finding
via documentation
- Creating block-based MVParts
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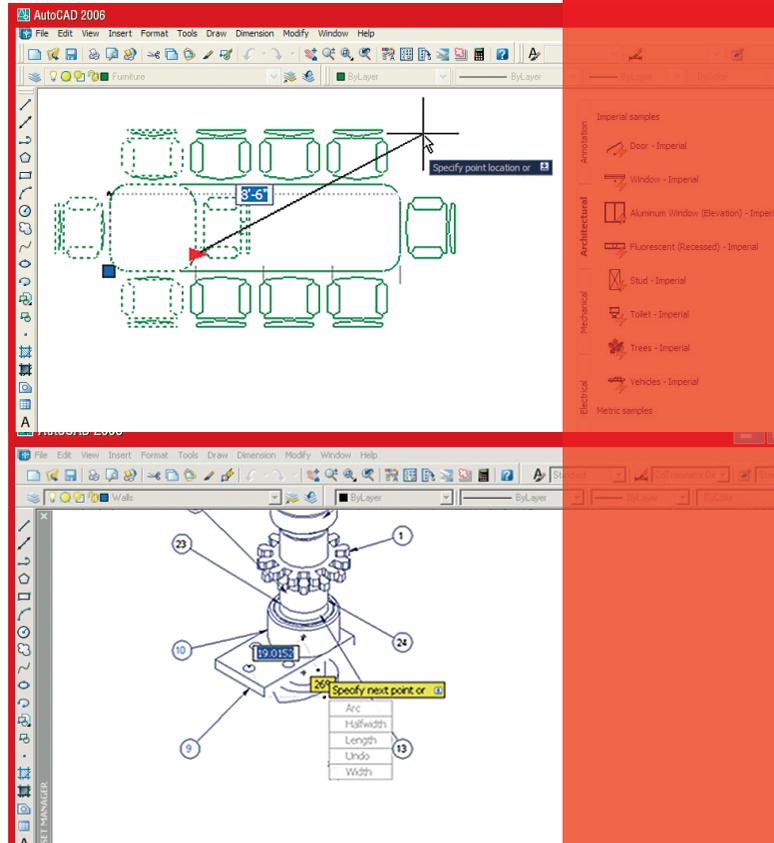
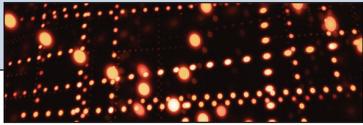


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TIPS & TRICKS FOR USING AUTOCAD 2006

Lynn Allen dives into AutoCAD 2006 with some great tips for boosting your productivity



AUGIWorld

Editorial

Managing Editor

Marilyn Law
marilyn.law@augiworld.com

Technical Editor

David Harrington
david.harrington@augiworld.com

Contributing Editors

Lynn Allen
R. Robert Bell
Nick Fuller
Tommy Huckabee
David Kingsley
Mitchell McComb
Matt Murphy
Mike Partenheimer

Production

Tim Varnau
tim.varnau@augiworld.com

Advertising

Dan Teeter
dan.teeter@augiworld.com

Controller

Larry Teeter
larry.teeter@augiworld.com

Publisher

Rich Uphus
rich.uphus@augiworld.com

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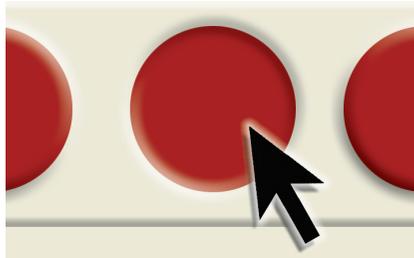
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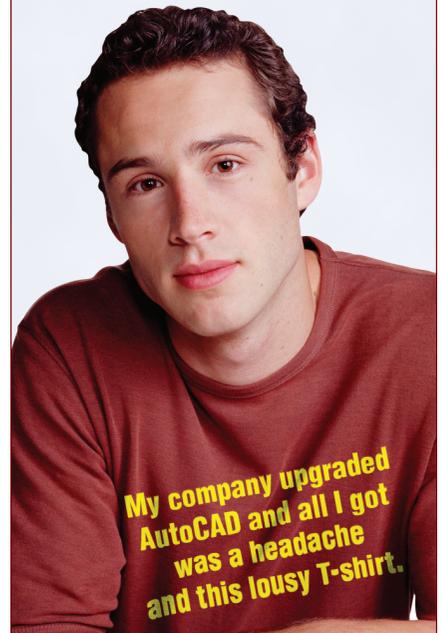
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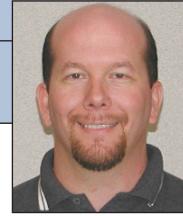
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The Cad Manager



David Harrington

Certification, Part Deux

In my last column I broached the subject of AutoCAD® certification and how it is sorely missed from the profession at the moment. I suggested that if you agree, you contact your Autodesk vendor and see what collective momentum can be generated.

I got a decent amount of feedback, which is always nice. Some readers sent a simple “I agree,” while others offered up improvements on my thoughts. Seemed to be enough there to warrant a follow-up article!

Well, for starters, yes, there is a huge desire to have something that proves that person A knows more than person B. Obviously, it is person A who wants this. I doubt person B will be supportive of the realization that he or she doesn't know all that much. Although, frankly, person B may not care (which is a subject for another time).

I was a bit surprised to learn that the cost of this certification was a concern. Some felt it should be free to take, while others felt if it didn't cost something then it wouldn't be worth the paper used to

print the certification. I will offer up my suggestions to this issue.

Should the test be free? No way. No company or organization is going to do this and spend money doing it. There has to be a baseline expense to cover the production process of the test, be it a website that houses this test or the real documentation materials. Also, having a baseline fee for the test will eliminate the casual CAD guy who wants to get an easy certification.

I would propose that the AutoCAD Level 1 Intermediate cost \$99 per test. If you fail, you are out the cash. This test would be timed via a website. Because there are many people who would take this test, it would need to have wide access. We would trade the possibility of cheating for ease of access. The \$99 cost is a lot to pay to allow your smarter CAD buddy to take the test for you. It might happen, but I doubt often. This test, being done online, would probably take about an hour in total, with perhaps four segments. There would be a section with general-knowledge, multiple-choice questions. This would be timed such as to limit your ability to look

...there is a huge desire to have something that proves that person A knows more than person B



up the answers. There could be a drawing test section. It would offer up instructions on what to draw and then require that you respond with values or measurements to determine if you did it right. Could be quite fun I would think.

Next up is AutoCAD Level 2 Expert at \$195. Here we start to get a bit tougher. Of course, you need to have passed Level 1 first. This test will require physical presence testing via a trip to an authorized Autodesk Training Center (ATC). In this venue you would have two sections, perhaps two hours in total. Part 1 would be questions, but you provide the answers, not pick from a list. The second section would be a drawing test that branches into 3D concepts, but since it requires physical presence the validation is much more reliable.

And then finally we have AutoCAD Level 3 Master, the really tough one. The cost continues to rise to \$295, commensurate with who is taking this test. You are likely to do this only if you see a reason to have the title, similar to Sir in the United Kingdom. It also would be conducted at an ATC. You would have to be pretty smart to get the chance as well, passing Levels 1 and 2. I would even go so far as to propose a Level 2 passing grade of 95 percent or better. This test would comprise nothing but questions testing your massive cranium's powers of deduction. You would be given problem scenarios to solve accurately. Knowing that some problems in AutoCAD happen in various ways, you have to know all the ways things go amiss, not just one way. Can you imagine what this test would prove? Probably that you know more than many of the programmers at Autodesk! Try not to get a big head, okay?

Now what about updates? AutoCAD is getting revised every year and so should you! I'm leaning towards a \$50 update test given online to keep you up to date—one

for each of the three levels. However, if you skip a release, you're out. You will have to start over, back to Level 1, buddy! Similar to other licenses—once they expire you lose the perk of easy renewal.

So after all the testing and, assuming you pass this massive test, what do you get? First up would be a listing on Autodesk.com in some section naming really bright people. So if you are pushing for a raise or a new job, you can offer up information that can be verified easily. If I said I was a member of MENSA, could you check that easily? Not as easily as this would be. And of course it would group the names in the Level they belong. The smart people are at the bottom. Hmmm. Perhaps a whole other page instead.

Next up is naming rights. Similar to other qualifications from Autodesk, you would be entitled to add blurbs to your signature. So instead of trusting you know what you think, you can put it right there for everyone to see.

David J. Harrington
AutoCAD 2006 Level 3 Certified

Well, I'm not going to go for just Level 2, am I? And I would hope that the cats

over in the Marketing Design group at Autodesk could make up some cool-looking icons for this, too! Would or could you do a signature block for Level 2? Sure. But for Level 1 I'm not so sure it is saying much. I would expect that someone who uses AutoCAD everyday for about two years would pass Level 1 easily. Level 2 would probably take four or five years with a component of customization as well. And then Level 3 would probably take six years or more years of experience, combined with a strong CAD manager and technical support element.

So, what do you think? Does this sound interesting and challenging? Would you expect AUGI to do this independently of Autodesk or not be involved at all? What would you change or add? Drop me an email and tell me your ideas. This has really got me going and I want to see it happen!

David Harrington, technical editor for AUGIWorld, works for Walter P. Moore and Associates in Tampa, Florida, USA. David is the author of Inside AutoCAD 2005, Inside AutoCAD 2002, and original programmer for REV-CLOUD. He can be reached at david.harrington@augiworld.com.

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Education & Training



Matt Murphy

Getting the Most Out of Your Software Investment

I'll be heading down to Florida soon for the Autodesk Boot Camp to get training on the 2006 products. But before I go, I've taken the time to read the press releases and white papers. I've also gone through the New Features Workshop and tutorials in order to get the basics of what's new. I don't always understand the full reason behind the new features, especially something that is revolutionary. That's why I go to training—but not until I've prepared myself mentally for it. Then after the training, I'll spend hours practicing and reviewing what was covered. I'll even send a couple of follow-up emails just to make sure I really get all that the new features offer and to tie up any loose topics I didn't quite understand.

Remember Tool Palettes in AutoCAD 2004 and Sheet Sets in AutoCAD 2005? I know too many people who closed these palette windows when they installed the software and never opened them again. Maybe you're one of them. Why? Because you don't understand them. It's far easier to dismiss them as something you don't need because there is already a way to insert Blocks and to

print multiple Layouts. But when something is incorporated into the software that dramatically changes the way you work, you need to take a dramatic approach to learning it.

So how do you learn, adapt, and implement the "new" techniques? It is not easy. There is no "EASY" button to purge out the old and plug in the new. We are creatures of habit and many old habits are bad habits. So how do you learn and incorporate the features for which you've paid?

Prepare and implement a productivity strategy

The entire purpose of upgrading your software is to take advantage of the features included with the latest version of the software. Merely installing the software in itself does not make you more productive.

There are many people who are proactive to learning and feel their first step should be training. This is not necessarily true! Training is definitely important and you should get it. But rushing to generic or "out-of-the-box" update training and then returning to work without a plan to use the new features means you and your team will surely slip back into old tech-

niques and your new training knowledge will soon be forgotten.

In fact many users fail to become more productive because they don't develop a plan. A truly successful plan will include four key stages: a current process analysis, a training program based on the gaps in the current process, an implementation phase, and then finally a review and follow-up. When all four are included in a comprehensive plan, I call this process-based training.

Process-based training

Process-based training incorporates all four of the key features that are required for successfully implementing a productivity strategy. Let me take you through the steps.

Step one: analysis process

A training provider will come into your company and assess what you're currently doing. What is your current skill level, what are your processes for using the software, and what is the pain in your current process? This is sometimes referred to as a "skill gap analysis." Everyone has holes and gaps in their knowledge and use of the software, yet you don't need to know everything. A good process analysis looks at all aspects of the way you're doing things now.

Step two: training process

Once the analysis is done, the training provider will match your current process with the techniques and processes that are incorporated in the current release of the software. In many cases, if you haven't received formal training in a few releases there are probably many features and functionality that you've missed and haven't implemented.

The provider will develop and deliver training that includes drawing problems and exercises based on your actual data. It's important to submit actual drawings to the training provider so when you practice the new techniques you're learning practical solutions. If you cannot provide actual drawings, then industry-specific examples should be used.

Is out-of-the-box training the best way to become productive?



Step three: implementation process

Now it's time to return to the office and apply what you have learned. As the daily pressure to complete jobs mounts, there will be a tendency to return to the old technique. It's familiar, it works, and sometimes there is not immediate productivity gain when you're still trying to apply the new method. Try to stick with it. If you've taken the training with other people in your office, use them as a support. Do some lunchtime group support or learning vignettes to keep you focused.

Step four: follow-up process

This is 30-day check-up done to ensure that the processes taught have been implemented by the training participants. In many cases this is an interactive session, allowing the group to share successes, but also to address the problems and pitfalls they have encountered. The training provider will evaluate the successes and failures of the implementation and then tie together any loose ends to bridge additional gaps in the training.

Regardless of the processes and techniques you've used before, this type of custom training is tailored to eliminate one thing, the number of clicks and picks you

make. The entire purpose of learning new processes is to gain productivity. Evaluate the new processes to determine if you are reducing the number of clicks and picks you do on a daily basis. Don't worry about how fast you are. Speed comes later.

Other keys to obtaining productivity

Which would you prefer: training by immersion or training by vignette?

Training by immersion

Who said that the best way to get update training is to do it in two consecutive days? Is it because it's the optimum way to learn or is it because it's for optimum scheduling? If you're like me, by the end of the first day of training I need to start to practice and apply what I just learned. By the second day I'm on overload. The mind can more easily absorb and digest small segments of learning with personal practice and review time between new episodes of learning.

Training by vignettes

With today's hectic schedules, doesn't it make more sense to learn in smaller increments? How about one day of training followed by a few days between ses-

sions? Half-days of training provide time to practice and absorb the material more easily. How about taking your lunch hour, sitting down with your colleagues in a conference room and have brown bag learning. Lunch and learn!

Final recommendations

1. Be familiar with the new features before you take your training.
2. Insist on practical real-world examples and exercises.
3. Evaluate and do a follow-up to determine if what you've learned is saving you time by reducing the clicks and picks of your previous methods.

If you have a training success story, I would like to hear about it. Don't forget to login to www.AUGI.com for my "Training the Trainer" articles in *AUGI HotNews*.

Until next time, don't close that new dialog box!

Matt Murphy, chair of the Autodesk Training Center Advisory Board (ATCAB), is an Autodesk Certified Instructor (ACI). He teaches AutoCAD productivity seminars at ATCs, Autodesk University, AUGI CAD Camps, and private companies. He can be reached at matt.murphy@ACADventures.com

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TIPS & TRICKS FOR USING AUTOCAD 2006

**Lynn Allen dives
into AutoCAD
2006 with some
great tips for
boosting your
productivity**



For people like me who've been using AutoCAD® for more years than I care to remember (okay, more than 20, to be precise), it's kind of amazing to see how technology has transformed the fundamental work of drafting for engineering, architecture, and building. What's even more exciting is that there seems to be a never-ending stream of innovation, when it comes to AutoCAD 2006.

In fact, all the cool new features and functions in this release are part of a roadmap to continually enhance five key aspects of computer-aided design: core drafting tasks, publishing, presentation, workflow, and customization. *AUGIWorld* readers got a preview of tips and tricks to customize AutoCAD 2006 in the April issue of *AUGI HotNews*. Here, I've selected more techniques and timesavers from dozens collected in my new booklet, which you can get free when you register and attend the Autodesk Realize Your Ideas tour coming to a city near you!

Heads-Up Design: Dynamic Input

A participant in one of the Autodesk usability studies once said that using older versions of AutoCAD was a little like watching a film with subtitles: The user is continually looking up at the main picture and then down below at the command line, as he or she works.

Save that fancy glancing for the big screen and a bag of popcorn! AutoCAD 2006 has dynamic input and pointer input capabilities that let you enter information at the cursor instead of at the command line.

1. Simply turn Dynamic Input ON from the status bar by clicking DYN, or hit the F12 key. Select the arrow down key to display any command options (or use the familiar right-click shortcut menus).
2. Right-click on DYN to view the Dynamic Input tab of the Drafting Settings dialog. Here you can indicate how much (or how little) information you want to display.
3. To display the command line input and prompts from the cursor, make sure that "Show command prompting" and "Enable Pointer Input" are toggled ON (Figure 1).
4. You can control the input format for coordinates and input values by selecting Settings. You'll probably want to stick to Relative and Polar (Figure 2).

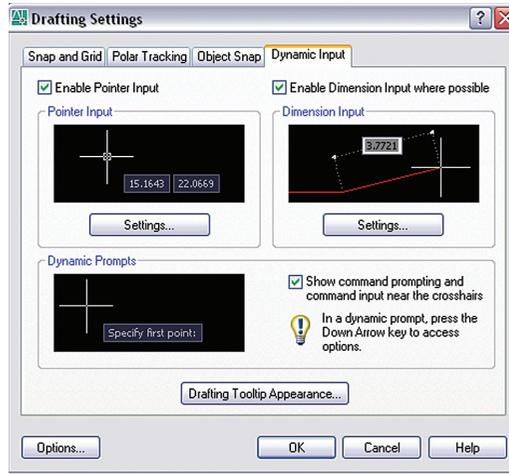


Figure 1. Click to enable point input and dynamic prompts, so that your command line follows your cursor.

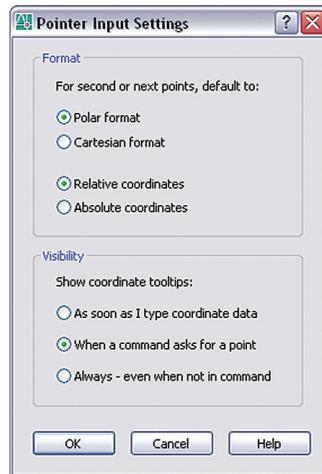


Figure 2. Choose how and when you would like pointer input to appear.

5. Want a quick way to turn the traditional command line on or off? Just hit Control+9 (for you adventurous souls).

Direct Input to Change Objects

Now you can change the length or angle of existing objects when creating existing geometry or grip editing simply by directly inputting the new value on the screen (Figure 3). Here are a few tips for direct input.

1. Toggle on "Enable dimension input where possible."

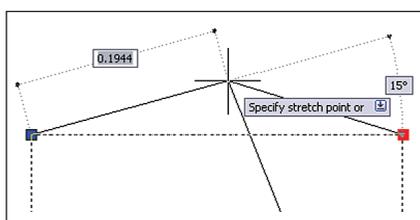


Figure 3. Input value changes directly when grip editing.

2. Use the tab key to toggle between the two values.
3. Choose from five different fields you can display for grip editing: Resulting Dimension, Length Change, Absolute Angle, Angle Change, and Arc Radius (available by selecting Settings under Dimension Input). Tip: More information isn't exactly better (and can get pretty overwhelming—I am content with just showing two dimension input fields at a time).
4. Set the system variable TOOLTIPMERGE to 1 to display both the indicated object snap and the command prompts.

Quick Command and Recent Data Access

Some other features that help make AutoCAD 2006 more intuitive to use include capabilities for finding recent tasks and data, and switching between drawings.

The new AutoComplete functionality will finish your input for you, which is handy when you forget how to spell a specific system variable or infrequently used command. Use the tab key to cycle through all the possibilities. And a new Recent Input option appears on your shortcut menus, making it easy to recall a previously used value (Figure 4).

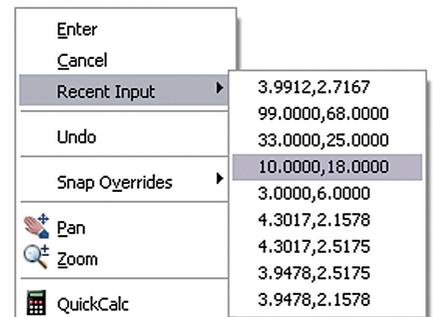


Figure 4. Click on Recent Input in the shortcut menu to pull up values you've used.

Finally, when multiple drawings are open, each drawing will display as an icon on the Windows taskbar. To switch drawings, simply select the drawing icon—just be sure TASKBAR is set to 1 (this works in AutoCAD 2005 as well).

Tables

No more drawing grids by hand! AutoCAD 2005 added a new intelligent TABLE object that eliminated a lot of the tedium and saved you time. Here is a review of tables along with some new capabilities added into AutoCAD 2006.

TIPS & TRICKS FOR U

Getting Started

Just as you do with dimensions and text, you first set up your table style in the Table Style dialog box.

1. Select Tablestyle from the Format menu.
2. Select New to create a new table style.
3. Set up your table style to reflect your needs, such as table direction, text height, alignment, border properties, and so on.

Create Your Table

1. Select Table from the Draw menu.
2. Select your table style.
3. Indicate the number of rows and columns.
4. Input specific column width and row height, or let AutoCAD determine it by the table size.

Populating Tables

In-place editing makes it easy to fill in the cells.

1. The Tab and arrow keys move across cells.
2. Double-click a table cell to enter text using the MTEXT editor.
3. You can also insert fields and symbols from the shortcut menu.
4. Clicking in a table cell permits you to insert a block from the shortcut menu. You can let AutoCAD software fit the block in the cell or specify a scale factor and the table will adapt accordingly.
5. Right-clicking also allows you to merge cells, add and delete rows, and so forth (Figure 5). You can use grips to modify the table location, column width, and row height.

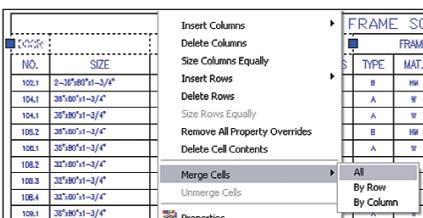


Figure 5. Click in a table cell to manipulate its format.

Accessing Tables from Excel

1. Copy Excel table data to the clipboard.
2. Select Paste Special from the Edit menu.
3. Select AutoCAD Entities.

4. Place the table in your drawing.
5. Formulas will also come across in AutoCAD 2006.

Exporting AutoCAD Tables

The TABLEEXPORT command will save an AutoCAD table out to a comma separated value (CSV) which can in turn be imported into Excel or Access.

Extracting Block Attribute Data to a Table

1. Execute the Attribute Extraction Wizard (EATTEXT).
2. Indicate which blocks to extract (Figure 6).

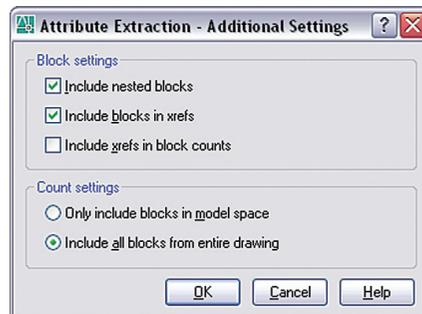


Figure 6. Choose which blocks to extract to your table, using the wizard tool.

3. Select the attribute information to extract.
4. Select "AutoCAD table" as the output. The information in the table is linked to the attribute data and is updateable (Figure 7).

DOOR SCHEDULE								
SYM.	WIDTH	HEIGHT	STYLE	REF#	MANUFACTURER	QTY	COST	TOTAL
1	3'	6'-8"	TWO PANEL	TS 3010	TRU STYLE	2	189.00	378.00
2	3'	6'-8"	TWO PANEL	TS 3010	TRU STYLE	7	189.00	1323.00

Figure 7. The table you create presents data that's linked to block attributes.

Performing Calculations

Don't bother with spreadsheet formulas. AutoCAD 2006 lets you apply simple numeric operations such as Sum, Average, and Count, create arithmetic expressions, and set cells equal to other cells. Select Insert Formula from the shortcut menu (Figure 8).

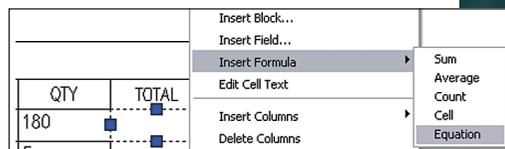


Figure 8. Choose a formula to calculate the value of data within a table cell.

Dynamic Blocks

Managing blocks can seem like a full-time job in itself. When your team has to plot, count, cost, and order the items the blocks represent, from dozens of windows and doors to thousands of fasteners, there's plenty of risk of error. AutoCAD 2006 drastically changes all that, with new capabilities for dynamic blocks.

Now blocks are powerful, flexible, data-driven objects, and one block can have multiple definitions of size, material, and more. Coupled with the new table creation capabilities, AutoCAD 2006 eliminates the countless manual updates to ensure finishes and sizes are consistent and quantities are accurate. Here are just a few advantages in addition to enormous gains in productivity and accuracy.

- One block with multiple definitions can drastically reduce the size of your block libraries. For example, one bed block, many options (Figure 9).

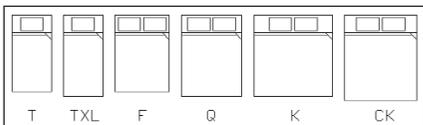


Figure 9. Define multiple options for a single dynamic block.

- Dynamic blocks can automatically align with existing geometry (Figure 10).

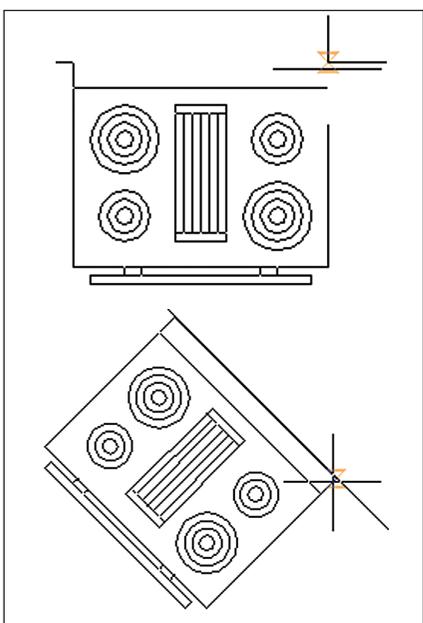


Figure 10. Align dynamic blocks with the geometry of each placement.

- You can edit individual components of a block. You can even assign specific increments for stretching or rotating with minimum and maximum values (Figure 11).

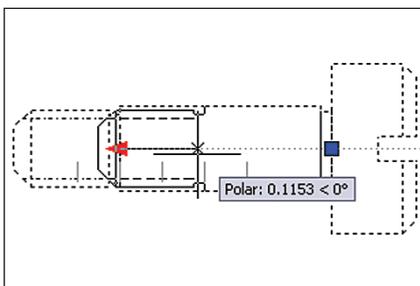


Figure 11. Edit the components of each dynamic block.

- You can create blocks with multiple insertion points (Figure 12).

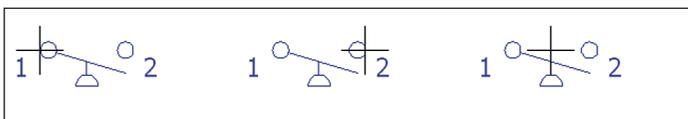


Figure 12. Create multiple insertion points for each dynamic block.

Defining Dynamic Blocks

The new Block Definition Editor (BEDIT) makes it easy to create dynamic blocks or edit your existing blocks in just two steps:

1. Select an existing block.

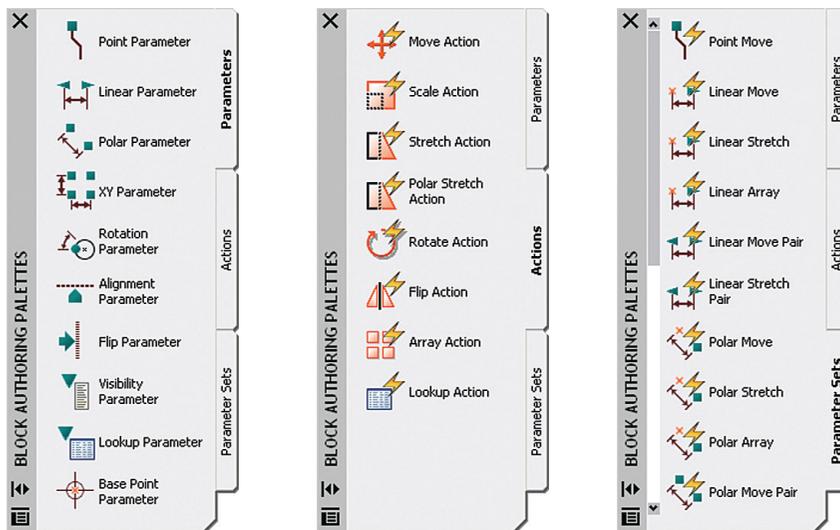


Figure 13. Assign actions to your block authoring parameters, to see dynamic block characteristics.

2. Right-click and select Block Editor from the shortcut menu.

The Block Authoring Palettes contain the tools to make your blocks dynamic (Figure 13). The process looks like this:

1. Select a Parameter and assign it to a portion of your block.
2. Select the action you wish to assign to that parameter. Try one parameter/action pair at a time to ensure you are getting the proper results.
3. Place the Action somewhere near the parameter. Most Parameters must have at least one Action.

The following definitions of aspects of dynamic blocks will help you understand their capabilities.

Parameters

You can think of parameters as dimensions that drive the block geometry (Figure 14). For example:

- A linear parameter to a door block will drive the width of a door (when combined with Stretch Action).

TIPS & TRICKS FOR U

- A rotation parameter to a chair within a “table and chairs” block permits individual rotation of the chair after insertion (when combined with Rotate Action).

Some useful tips:

1. Selecting the Properties of a parameter allows you to specify increments as well as minimum/maximum values.
2. Visibility Parameters are used to assign multiple definitions to one block.
3. Lookup Parameters can be used to assign multiple sizes of a specific block. A simple right-click allows the user to change from one size to another.

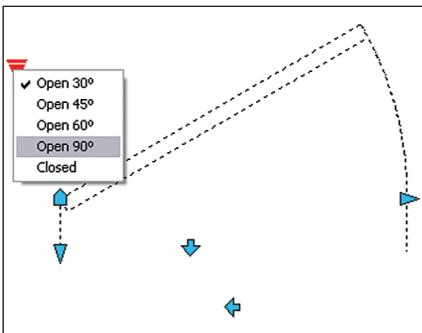


Figure 14. Parameters allow you to assign multiple definitions to a single dynamic block.

Actions

Actions drive the geometry in specific ways. For example, use Stretch to change the length of a bolt. Here's a tip: An exclamation point will display to indicate any parameters with no assigned action.

Customization

AutoCAD 2006 offers a wealth of options for customizing the user interface (Figure 15). In the April *AUGI HotNews*, I shared some techniques for customizing the interface to suit your needs.

Here are some tips for customization of fields—a feature introduced in AutoCAD 2005 that I like to highlight because it takes data input to the next level (think attributes on steroids!). Fields are like “smart text” that updates automatically.

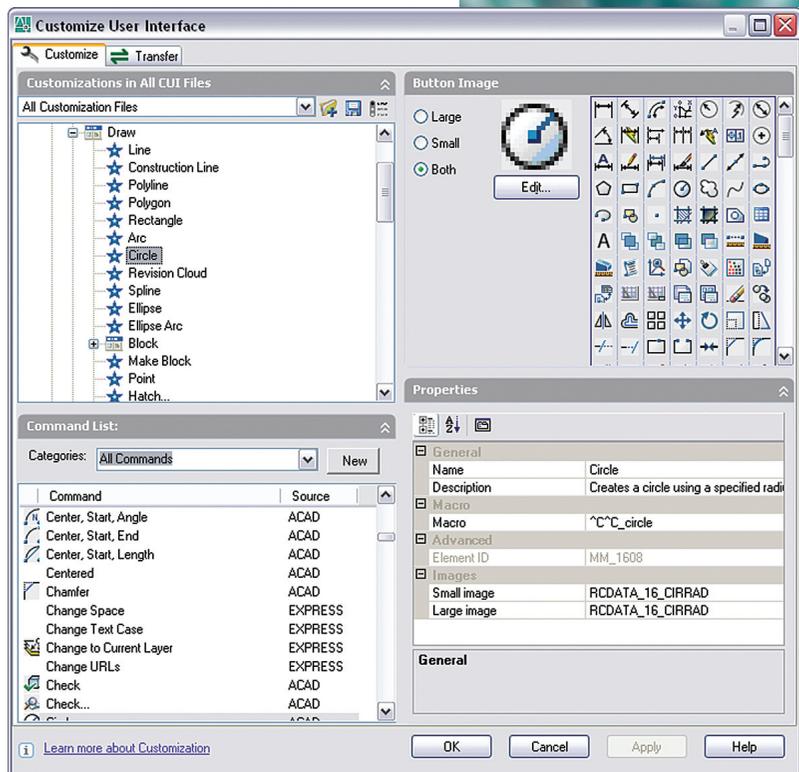


Figure 15. Choose from an extensive list of interface customization options for AutoCAD 2006.

You can use field data for such things as dates, sheet numbers, titles, and so on. Fields can work across sheet sets to update all instances (think “Change it once, change it everywhere”).



Lynn Allen, Autodesk technical evangelist, speaks to more than 30,000 AutoCAD users worldwide each year. For the past 12 years she has written a monthly column in *Cadalist* magazine called “Circles and Lines,” and has authored several books about AutoCAD. Lynn has been teaching at the corporate and college level for 14 years, and started using AutoCAD software with Release 1.4 more than 20 years ago. Visit Lynn Allen's blog at www.autodesk.com/lynnallen and access her *Cadalist* columns at www.cadalist.com.

To get a copy of Lynn Allen's new “Tips & Tricks for Using AutoCAD® 2006” booklet, sign up and attend the Autodesk Realize Your Ideas Tour in a city near you. You'll get an insider's look at the new AutoCAD 2006 software. You'll also get a free copy of Lynn's tips and tricks booklet (a \$14.95 value and only available at select Autodesk events including the Realize Your Ideas Tour). These events are free, but registration is limited. Visit www.autodesk.com/augi to register today.

SING AUTOCAD 2006

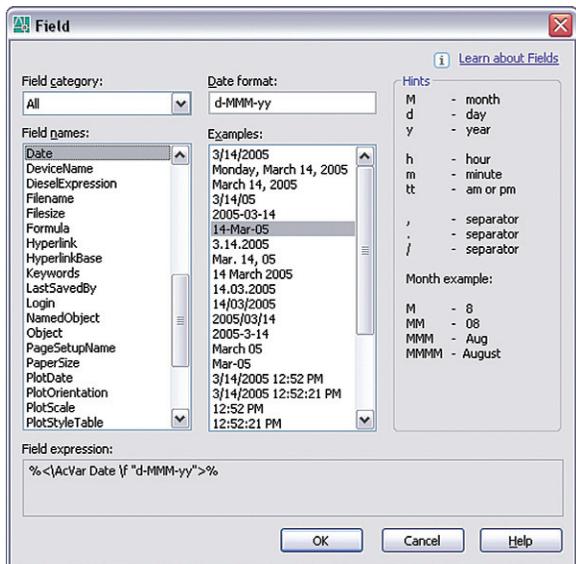


Figure 16. Choose fields to insert, from a pre-defined list.

Inserting Fields

Select fields from a list of predefined fields (Figure 16). These fields can be

background for field text, so that it's easy to recognize information that is Field text.

inserted into text objects, attributes, or table cells. There are three methods for inserting a field:

1. Select Insert Field in the shortcut menu when prompted for text in MTEXT, DTEXT, ATTDEF, and BATTMAN. Some of these commands also have an "insert field" button.
2. Press Ctrl+F.
3. Execute the Field command (this will place the field as MTEXT).

Whichever method you use, simply select the field you wish to add. The FIELD-DISPLAY system variable toggles the display of a gray

Editing Fields

Easily edit your fields by double-clicking. The appropriate editing command will appear—DDEDIT, EATTEDIT, and the like.

Updating Fields

By default, field values automatically update when you open, save, plot, eTransmit, or regenerate a drawing. Here are a couple more tips:

You can suppress this automatic evaluation by setting the FIELDVAL system variable.

You can also use the UPDATEFIELD command to manually update.

Drafting, Transformed

This is only a snapshot of some of the exciting new features in AutoCAD 2006. From dynamic input capabilities to a host of step-savers and productivity enhancements, AutoCAD 2006 takes drafting to a new level of productivity. Best of all, migration is easy, deployment is simple, and intuitive new features make learning quick. Say goodbye to your painstaking workarounds, and hello to a more efficient way to work.

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Four Simple Rules

» *“Where are the vla-functions documented?” A commonly asked question, with an answer you might not expect. Here are four rules for solving the mystery*

Thinking about objects, properties, and methods is foreign to the traditional AutoLISP® programmer. The lack of straightforward documentation on the vla- functions compels many to avoid the ActiveX interface. So where is the documentation? Before I tell you, there are some basics to cover first.

Objects, properties, and methods

An object is just what it sounds like, “something mental or physical toward which... action is directed” (Merriam-Webster Dictionary). In AutoCAD®, an object can be a visible thing such as lines, arcs, and text. It can also be a non-visible thing such as a layer, text style, or data storage dictionary. Another related term, collection, simply refers to a container of similar objects, e.g., the Layers collection.

Obviously, an object will have properties. For instance, you know intuitively

that a line has a starting point and an ending point. It also has other properties in AutoCAD such as its layer. Some properties are modifiable (TrueColor) and others are read-only (an object’s handle).

Objects also have methods. Methods are actions that the object supports. For example, a line supports move, copy, and rotate methods. Different objects support different methods.

Now you have an understanding of the basic terms used in the ActiveX interface. It is time to reveal the vla- function documentation and my 4 Simple Rules.

The 4 Simple Rules

The documentation for the vla- functions has always been located in the ActiveX and VBA Reference. The problem is that it was written for the VBA programmer, not the Visual LISP programmer. However, Autodesk knew that the “translation” of the VBA documentation to Visual LISP was so easy that there was no real need to duplicate the documentation unnecessarily.

Take a moment to look at the Object Model map in the reference (see Figure 1). This map is a great way to start a session in the documentation. You can click on any of the objects or collections to go directly to that object/collection’s documentation. It will also give you an understanding of the hierarchy of the objects in the ActiveX interface. Notice how the Application object is the start for most of the object model.

Once you have the Object Model map displayed, select the Line object. This screen lists all the methods and properties (also events; we won’t be discussing those) available to the object (see Figure 2). It tells you what objects may contain a line. You will see nothing about Visual LISP here. However, translating the “VBA-speak” into a vla- function only requires these following four rules:

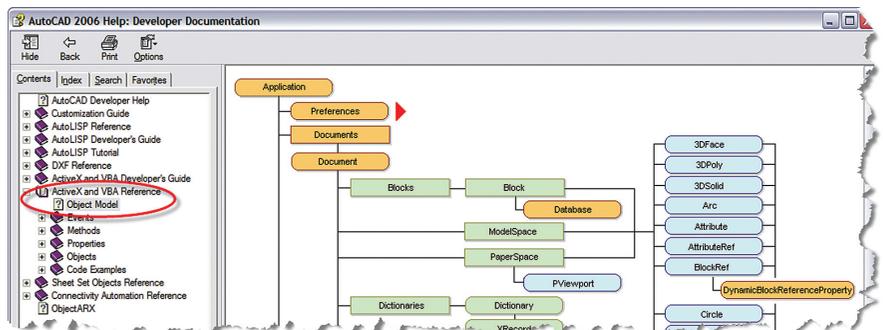


Figure 1

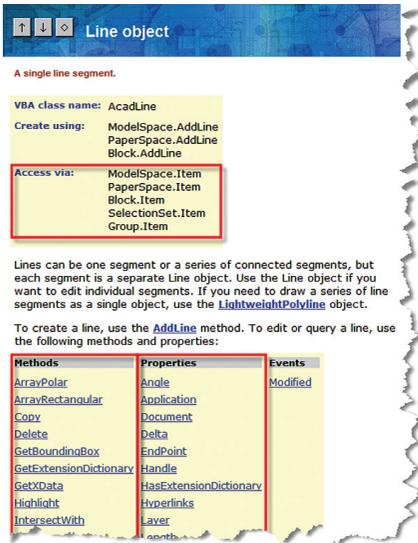


Figure 2

1. You always provide the object as the first argument

Remember, properties and methods wouldn't exist without an object. So every vla- function will require an object. Therefore, the first argument of every vla-function will be the object itself.

2. You query a property's value by prefixing the property name with vla-Get-

All the properties are retrievable in Visual LISP by prefixing the property name with vla-Get-, e.g., vla-Get-Layer. Because you are only querying the property, there is only one argument—the object.

3. You modify a property's value by prefixing the property name with vla-Put-

All the properties that are not read-only are modifiable in Visual LISP by prefixing the property name with vla-Put-, e.g., vla-Put-Layer. Because you are modifying the property, you need two arguments: the object, and the data to change the property. The type of data needed to change the property is documented in the Signature section.

4. You execute a method by prefixing the method name with vla-

All the methods are available in Visual LISP by prefixing the method name with vla-, e.g., vla-Delete. The number of arguments will vary from method to method and will include the object as the first argument along with the documented arguments. The type(s) of data needed by the method's arguments are documented in the Signature section.

Querying a property

As you can see, up to this point the effort to “translate” the ActiveX Reference is trivial. Now let us look at some examples.

First, draw a single line in AutoCAD. We will use this line for the code samples that follow. Save the line object to an AutoLISP variable. (vl-Load-Com loads the ActiveX interface and vlax-ENAME->vla-Object translates an entity name into an object.)

Command: (vl-Load-Com)

Command: (setq myLine (vlax-ENAME->vla-Object (entlast)))

#<VLA-OBJECT IAcadLine 05e8d054>

Look at what the ActiveX and VBA Reference tells us we can do with our object. Select the Line object in the Object Model map. That should jump you to the documentation for the line object. Let's say that we want to find out what layer the line is on using ActiveX. Select the Layer property in the documentation.

Now let's apply the four rules.

Are we working with a method or a property? It is a property, so rule 2 states we need to prefix the property name Layer with vla-Get-, i.e., vla-Get-Layer is the function. Since we are only querying the

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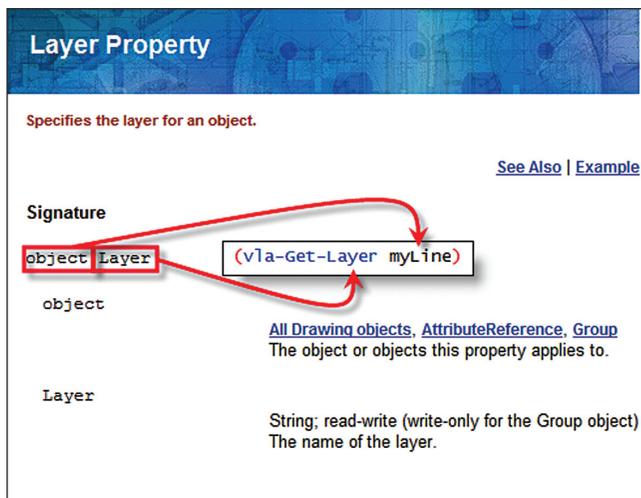


Figure 3

layer property we can skip rules 3 (modifying a property) and 4 (a method's prefix). Rule 1 tells us that the first argument is always the object itself. So the AutoLISP statement that will return the line's layer is:

Command: (vla-Get-Layer myLine)
"0"

What type of data do we expect when we query the property?

Notice that the documentation in the Signature section states that the layer property is a string. Different properties will have different types of data. For instance, go back to the line object's documentation. Now select the StartPoint property. Notice that this property's data is a variant that holds an array of numbers. In Visual LISP an array is called a SafeArray.

Command: (vla-Get-StartPoint myLine)
#<variant 8197 ...>

However, a variant that holds a SafeArray needs to be "drilled into" to get the actual numbers in which you are interested. The two functions used to get at the data stored in a variant and SafeArray are: vlax-Variant-Value and vlax-SafeArray->List. Many Visual LISP programmers find it easier to combine those two functions into one and use that single function instead.

```
(defun i:Array->List (variant)
  (vl-Load-Com)
  (vlax-SafeArray->List (vlax-Variant-Value variant)))
```

Now you can use that function to return the list of numbers that you expect:

Command: (i:Array->List (vla-Get-StartPoint myLine))
(15.4375 16.1814 0.0)

Modifying a property

This next example will change a property, the layer on which the line resides. Now we will apply rules 1 and 3. Rule 3 states we need to prefix the property name Layer with vla-Put-, i.e., vla-Put-Layer is the function. There will be two arguments. Don't forget rule 1! The first argument is always the object itself. Since we are modifying a property, it makes sense that the second argument would be the value itself. The documentation tells us that the layer property is a string, so we must provide a

Figure 4

string. Before you attempt the following code, please create a layer named "Test" that is visually distinct from the current layer of the line.

Command: (vla-Put-Layer myLine "Test")
nil

Notice that when the statement succeeds, nil is returned. If the statement fails (say the layer doesn't exist) then an "Automation Error" is returned.

Strings and plain numbers such as integers and doubles (reals in AutoLISP terms) are easy to provide. Some properties will require a variant, usually points for objects such as lines or lightweight polylines. Therefore, many Visual LISP programmers have written a function similar to the following to quickly convert a list of reals into a SafeArray.

```
(defun i:Points (ptList)
  (vl-Load-Com)
  (vlax-SafeArray-Fill
   (vlax-Make-SafeArray vlax-vbDouble
    (cons 0 (1- (length ptList))))
   ptList))
```

You could use the above function to change the line's StartPoint property to 0,0,0.

Command: (vla-Put-StartPoint myLine (i:Points '(0.0 0.0 0.0)))
nil

Executing an object's method

Finally, let's use the move method to move the line. Go to the Move method in the ActiveX and VBA Reference. Note that it documents two arguments, Point1 and Point2. Don't forget rule 1, however. The object itself is always the first argument. So the move method in Visual LISP will have three arguments: the object, the first point, and the second point. Rule 4 states that the

Figure 5

method's name will be prefixed by vla-, i.e., vla-Move is the function.

So use what you have learned so far to move the line:

Command: (vla-Move myLine (i:Points '(0.0 0.0 0.0)) (i:Points '(12.0 0.0 0.0)))
nil

Once again, if the statement succeeds, nil is returned.

Conclusion

As you have seen, it is easy to translate the ActiveX and VBA Reference into the functions needed for Visual LISP. The documentation was there all along—all you needed were the four simple rules.

However, there are some items for further discussion such as: output-only variables, vlax-Dump-Object, and the undocumented functions vlax-Get, vlax-Put, and vlax-Invoke. So now that you know where the vla- functions are documented and how to translate them, don't be shy about jumping into the ActiveX interface! Join us online in the AUGI forums, found at www.AUGI.com, if you have any further questions.



R. Robert Bell works for MW Consulting Engineers in Spokane WA US. He is their Network Administrator/Programmer. He has used AutoCAD since v2.18 (AutoLISP). He is on the AUGI Board of Directors and can be reached at robert.bell@augi.com.

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Creating Block-based MVParts with Autodesk Building Systems

What is an MVPart?

Imagine if you could have one part to insert into Autodesk® Building Systems that will display different in model view, top view, section view, and in a single-line drawing. Imagine no longer... you can! It's called a Multi View Part (aka MVPart) and Autodesk Building Systems has them! MVParts are a collection of blocks gathered into one object. You can create a solid model and let Building Systems create the alternate views for you, or you may create blocks for the views and specify them yourself.

5.5 steps to create an MVPart

Step 1 - Create and name a new part

In order for Building Systems to support automatic generation of blocks, all block-based MVParts need to be created from AutoCAD solid models. Creating MVParts from mass elements and third-party 3D conversions into AutoCAD solids may cause display errors in the process. Below are the 5.5 steps to follow for creating an MVPart. For this example I will be using a block for a centrifugal pump.

Before you can create a block-based MVPart, you must save your solid model as a block. It is recommended that your block be saved as its own file (use Wblock if you are working in a project drawing). I recommend saving in a well-thought-out file

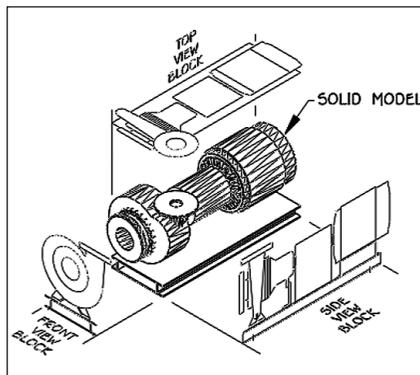


Figure 1

structure for future use that will accommodate later releases and possible part updates.

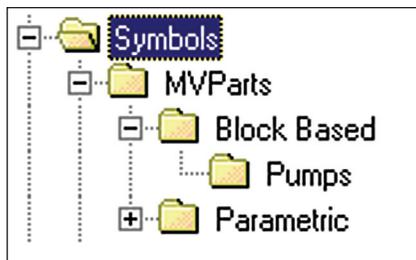


Figure 2

Your block needs to be as true to scale as you can make it (meaning overall length, width, height, and connection points need to be drawn accurately). Remember, it's important that your block represents the

important details required to properly locate and connect a service. If you choose to have separate detailed views for other orientations, leave the solid model with minimal detail to conserve file size. Your block should be drawn on Layer 0 and assigned byblock for color and bylayer for linewidth to ensure display control. I choose to name the block with the name of the company first followed by the model number of the part. Example for this block is: B&G 1510 4BC 254T. Save your block drawing in the plan (top) view before creating an MVPart.

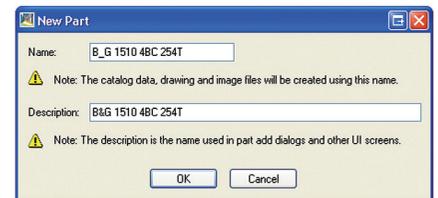


Figure 3

Open the drawing file that contains the newly created block. From the MEP Common pull-down menu, select Content Builder. Locate the directory MVPart >Mechanical >User defined folder. Select the new block part icon. You will get the new part dialog box. Enter a name. Remember, you can't use any specialty symbols within the name of the MVPart (% , / , & , and so on). The Description will automatically fill with the same information entered in the name. You may leave the description alone or you may specify any information you feel necessary. Click OK.

Step 2 - Specifying a category and part behavior

Building Systems will ask you to specify the category (type) of the part. Categories are predefined by Building Systems. You will also have to select a layer key and a subtype. If the part you've created is not on the subtype list you may add it by typing into the dialog box. You also have an option to select the behavior of this part. You can select whether the part should break into existing runs or if the part will be anchored to the side of a run. For this example the pump will have neither selected.

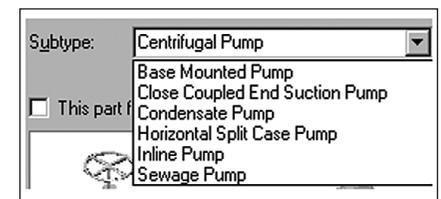


Figure 4

Step 3 - Designating blocks

The next dialog will prompt you for what blocks you wish to use to represent your part. You have the option to have Building Systems generate view blocks from your 3D model, or you may create each view block yourself and designate them for the drawing block representation. There are seven standard Building System block views available: top, bottom, left, right, front, and back. You may also assign a 2D symbol block.

Select your block from the list.

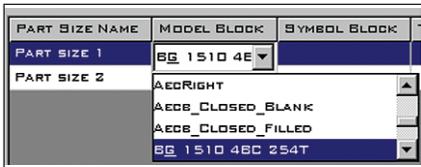


Figure 5

Notice that you have the option for multiple part sizes. This sets up the possibility of designating another pump size or a possible option to the part you create. You may wish to place multiple B&G pump model blocks into one drawing and use part sizes to select different pump models. Next we need to have AutoCAD® generate any of the block view representations we haven't created and specified already. Select the generate blocks button. Then click OK. Notice that Building Systems has filled in the missing block views. These blocks are generated and based on the standard display representation.

Step 4 - Generating an image for the MVPPart.

Select Generate an Image based on a model block from the SW Isometric View Option. To have Building Systems generate a preview image for you, select the block you've created to represent the model for MVPPart view representation. Select generate. Then select next. If you'd rather use a picture of the actual part or an image created elsewhere, you may do so by using the "select a previously drawn image" option. Previously drawn images are to be in .bmp format.

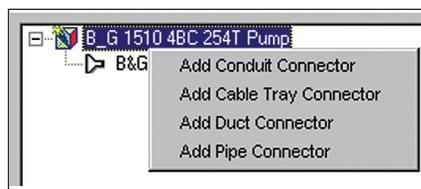


Figure 6

Step 5 - Adding connectors

Next we add connectors. Defining connectors allows the part to be connected intelligently to other Building Systems objects in a drawing. This helps assure that the connecting service a user will choose to connect to the pump matches any predefined conditions that you set. To add connectors, right click on the model name and add the appropriate connector type, name, and flow direction. After you add a new connection, a connection icon will appear in a file tree format.

Right click the new connection type you've just created and select "Edit Connector." You will be taken to a screen that enables you to place connectors on the part. The connection icon arrow helps you visualize what direction your flow connection takes.

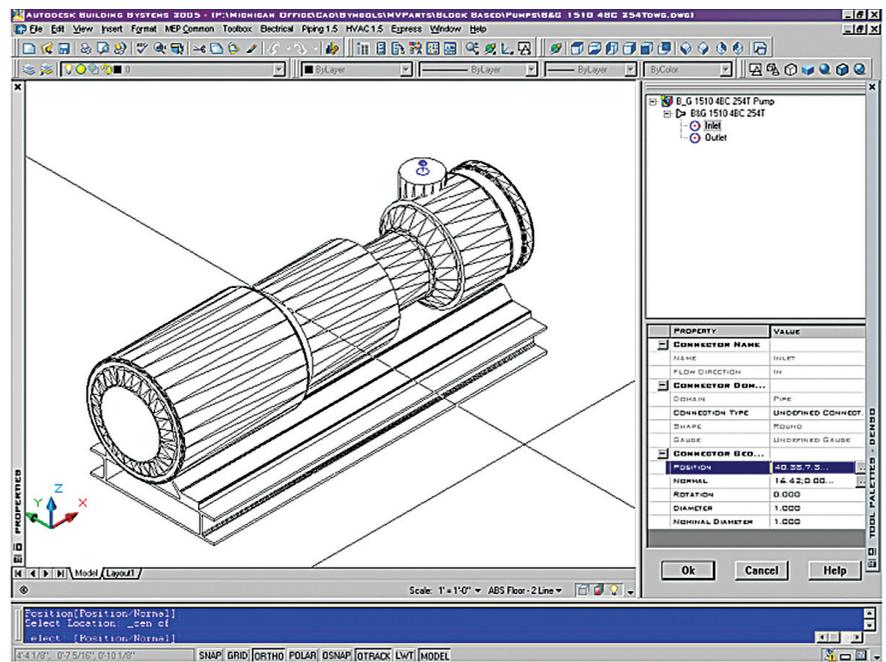


Figure 7

When locating the connectors there are a few options you need to know about. The first is the position option. This is what you select to locate the connection. Note in Figure 7 the location of the outlet connection point.

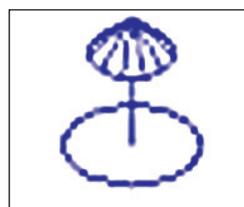


Figure 8

The next option is called normal. This allows you to rotate the connection arrow to actually represent which way the flow is

going. You can also designate the diameter of the connection size. Before connection is complete, you must define a connection type. When finished, click OK and then Next.

Step 5.5 - Adding property set information

If you wish to add more properties to your new MVPPart, you may do so through property set information. Property set information is optional and contains the ability to change the data storage, data type, units, visibility, and description. When finished with property set information, click finish.

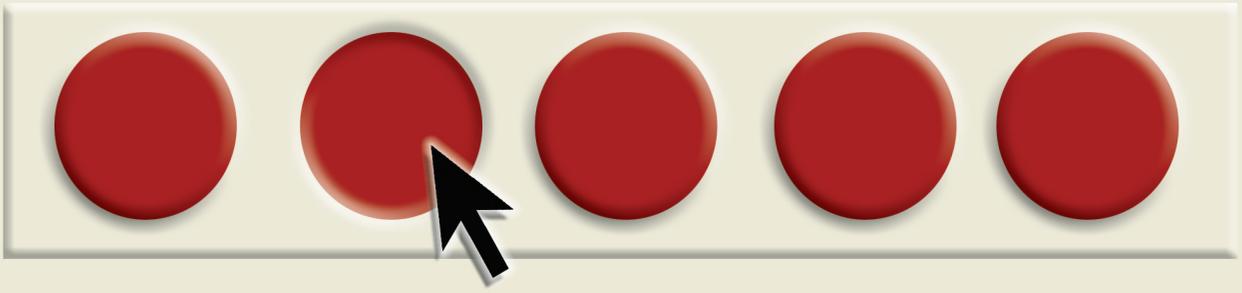
You're done! Piece of cake. You've now successfully learned how to create a block through MVPPart. By creating MVPParts through Content Builder, setting up a

smart 3D library has never been easier. You've learned how to create views, add smart connectors, and assign properties to MVPParts. The sky's the limit and the power is in your hands.



Mitchell D. McComb is a mechanical and electrical designer for ThermalTech Engineering, Michigan Office. ThermalTech specializes in innovative engineering, construction, and operations solutions for facility and utility customers.

Buttons for Scripts: An Easy Customization



In my last article I mentioned that buttons can be made to automatically run your favorite scripts. In this article I will delve deeper into this subject by taking the script we previously wrote and creating a button that will run it.

Here, we will learn a little bit about the customize dialog box, user-defined commands, and menu macros. We will also briefly go over script formatting and file extensions.

Getting started

First, if you do not have a script I have attached one below. If you have one that you use quite often, it would be best to use it.

```
;;MY FIRST SCRIPT
```

```
SETVAR DIMASZENTER.1875
```

```
;;dimension and leader arrowhead size
```

```
SETVAR DIMEXE .0625
```

```
SETVAR DIMEXO .125
```

```
SETVAR DIMTXT .1875
```

```
SETVAR DIMCEN -.093
```

```
SETVAR DIMDLE .0625
```

```
SETVAR DIMDLI .375
```

```
SETVAR PDSIZE .0625
```

```
SETVAR DIMGAP .0625
```

```
SETVAR DIMSCALE 24
```

```
SETVAR TEXTSIZE 4.5
```

```
LTSCALE 12
```

If this is your first time writing a script, retype this script exactly as it is shown here. I use Microsoft Notepad, but any program that can save to an ASCII format will be fine. When saving a script it needs to have a .scr file extension to be recognized by AutoCAD®.

Now that we all have the script, we will test run it to make sure it works. Open a new AutoCAD drawing and from the tools menu select Run Script. From the dialog box find this script and choose open. You should see your command line run through the entire script. If this doesn't happen, press the F2 button. If your script hasn't been run, open it up and check your typing for errors. Unlike AutoLISP, scripts are not equipped with their own troubleshooting mechanisms, so errors are easy to make, but sometimes hard to find.

Creating a button

Now that we have successfully run our script, we can start on the "button making" process. First, open a new drawing. Right click on one of your tool bars and click customize. This brings up the Customize dia-

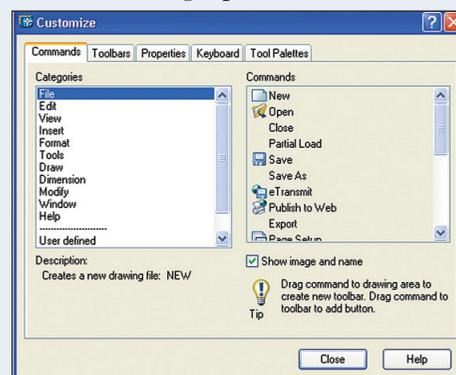


Figure 1 displays categories containing all commands.

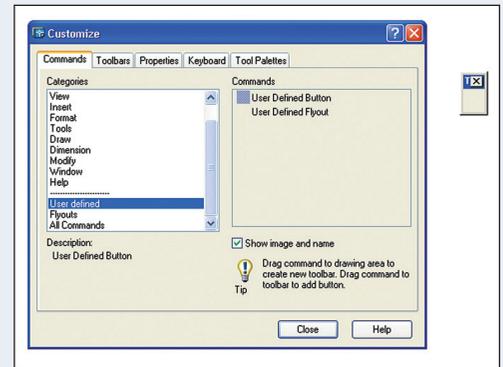


Figure 2: After dragging the button onto your screen it is displayed as a blank button.

log box. In this box there is a tab labeled Commands; click on this.

There is a list of commands categorized by file, edit, view, and so on. You are able to click on a category to bring up all of the associated commands on the right side of the pane. Clicking on any one of the commands will provide a description of what it does.

Click on the User Defined category. In the right pane there are two options: User Defined Button and User Defined Flyout. Click on the user defined button and drag it onto your screen (see Figure 2).

The button is blank, but don't worry, we will customize the look of it soon. Now that you have a button for your script, all there is left to do is customize it and add a macro.

Click on your new button; it brings you to the Button Properties tab. This is where all the magic happens. First off,



Figure 3: Names and descriptions can be applied and changed in these windows.

give your new button a name. I generally try to keep mine as technical as possible because you never know who may come behind you and see it. Next fill in a description of what your button will do (see Figure 3).

This makes it easier if you plan to share this knowledge with your co-workers or if you have a new employee that has never seen these buttons before and doesn't know what they do. It can save a lot of time explaining to the new guy the function of each of your buttons.

Creating a macro

Under the description, there is a section labeled Macro associated with this button. As the title explains, this is where we are going to type in a macro that AutoCAD will recognize and allow our button to run a script.

Already supplied is the command ^C^C, (see figure 4); this command is the equivalent of pressing esc twice. Usually pressing esc once is enough to exit a command, but when dimensioning there is a need to press esc twice to return to the command prompt. That being said, it is good practice to include this before all of your macros.

Following the ^C^C, type in the word script, then click apply, and close. Click your button and the Run Script dialog box should be displayed. Congratulations, you have created a user-defined button! Now, right click on one of your toolbars and click customize, then click on your button to show the properties associated with it. In the macro section type “_script” (minus the quotes). The “_” automatically translates the command that follows into other languages. For most of us, this isn't a must, but when you come across it, you will recognize its purpose. Also, by typing “_script” it will allow this macro to be used transparently. Again for this lesson it isn't

necessary, but for other macros it comes in quite handy.

At your own discretion, there can be a Tag added to your macro by using “[]” you can insert comments that will not be run by AutoCAD. This is optional, but I find that it helps out here and there.

We now have everything in place to finish up the button macro. Keep in mind that a macro is just like writing a script or entering a command on the command line. We have ^C^C_script, which is pushing esc twice, then the command script. Now we must either push Enter or the space bar, so push the space bar. A “;” or a space



Figure 4: Typing in the file path will allow AutoCAD to find your script

acts as if you were pushing Enter. After adding the space, type in the file path to find your script using “/” instead of “\” (see Figure 4).

This “\” is used as a pause for user input. If you create a macro using a circle you would use “\” to pause for the user to pick the center of the circle. Also, there are no spaces in the file path name as this would be interpreted as Enter and would send an “unknown command” error message. Just a hint: when you save your script file, type it in as one word. This can also send an error message of “cannot find file.”

Customizing the button image

Now, we will customize the look of your button. This is probably the most fun part of the whole exercise and maybe the best looking part of your screen. In the right pane of the Button Properties dialog box, there is a section called button image and under that is your button as it appears now—totally blank. Click on the edit but-

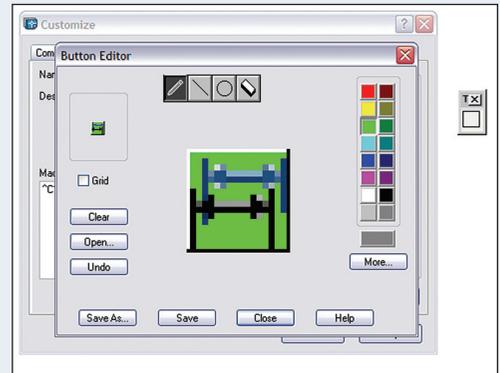


Figure 5: Existing dimstyle button slightly modified

ton, it brings up the button editor dialog. You have a choice of all the colors as well as an eraser, grid, line, pencil, and circle (see Figure 5).

If you want, you can start drawing something on the button to be displayed. I, on the other hand, pick an existing button from the window under the edit button and modify it a bit, taking less time, especially since I'm not an artist.

When you are happy with your button's appearance, click Save and save it in the My Pictures folder. (This is usually the easiest place in which to save it.) Then click close and you're finished! Move your new button wherever you want it and you are ready to go.

All buttons can be modified to suit your personality and most companies don't mind you changing a few small things. I would, however, recommend that in the beginning you experiment cautiously. You never know how hard it is to undo something until you have to do it. I've learned this from past experience; some of it not too pleasant!

Wrapping up

Everything is done and ready to go. After you've placed your button where you want it, click it and watch it go!

Menu macros are a great part of AutoCAD and can be customized in many ways. Users in every discipline have a need for custom toolbars and buttons. From dimstyles to multiple copies, it can all be customized to fit your needs.

Tommy Huckabee is a design draftsman for Oil States Industries, Houma, Louisiana, USA. He has extensive experience with AutoCAD 2005, AutoCAD Mechanical, the Autodesk Inventor Series, and has been an active volunteer at his local college. He can be reached at tommy.huckabee@oilstates.com

The Project Navigator Part One: Building a Project

*First in a two-part series on Project Navigator,
an Autodesk Architectural Desktop tool*



In this two-part series we will dig into the process of building a project in the Project Navigator (PN), explain the critical elements key to a sheet set, and explain how to manage a large multi-division building.

The PN is a document management tool built into the Autodesk® Architectural Desktop tool set. The PN tracks and modifies folder structure, drawing file names, and external referencing using XML code behind the scenes. The intent of the PN is to manage the related project drawings while streamlining the document management process. Redundant tasks such as plotting, transmitting electronic files, and archiving projects are simplified to single-click procedures.

In Autodesk Architectural Desktop 2005, the power of the PN was extended even further through the use of fields. Fields are an extremely useful feature that functions much like those found in the Microsoft product family. Fields take text entities and attributes to the next level by allowing them to become dynamic. Text is no longer simple geometry displaying inputs from a user, but now packs the power to report values on virtually any given AutoCAD® entity. For example, take a simple line. The line has specific beginning and endpoints, an angle, line-type, color, etc. Fields can report any or all of these aspects in a string of text, m-text, or even attributes. To take it a step further, if the information being reported in the field changes, then the resulting text is changed.

The possibilities are virtually endless; the user now has the ability to set up automatic schedules, title blocks, and reports. If you can imagine it, it can likely be reported through the use of a field. Fields are updated automatically any time a draw-

ing is saved, plotted, or opened. Fields are easily distinguished from regular text and attributes by having a shaded background as shown in Figure 1.

Figure 1

The PN builds with every project four basic folders: Constructs, Elements, Views, and Sheets. The project files are automatically x-referenced into the views and sheets simply by dragging them into the drawing window.

The Constructs folder is used to house files that are major aspects of the project. For example, a typical construct may be the exterior skin of a building. These components may be used multiple places throughout the drawing set, but are typically represent a non-repeating item in the building.

The Elements folder contains items that may be used over and over in a building. Let's use, as an example, a high-rise hotel. The rooms in the core of the building floor plan have a tendency to change to accommodate storage rooms, stairs, and so on, but the rooms on the outskirts often repeat themselves. An element could be used to represent the typical hotel room. For the exterior portions of the floor plan, the element may be used 100 times. When changes are made to the typical room element, the changes are reflected across all 100 of those rooms.

The views folder is self explanatory in that it houses views. Look at the floor plan shown in Figure 2, the building is broken up into four quadrants. For HVAC draftsmen, it could be a real headache to coordi-

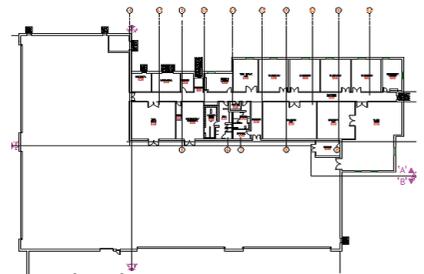


Figure 2

nate the beginning and endpoints of ductwork and piping as they pass across the match lines of the building. You can imagine how easy it would be to accidentally refer to the wrong sheet for continuation. With views, you have the ability to draw the ductwork for the entire building all in one file. You then create model space views of the various areas that will be shown on the different sheets. If a building component that continues from one sheet to another changes location, then the new location is updated on all affected sheets. The views part of PN also allows for easy enlarged plan and detail plan coordination in that you're simply creating larger scale saved views of the same file.

The sheets folder contains the plotting sheets of a project. They are the conglomeration of all the different aspects of the project tied together into construction documents. This folder contains all the files in a sheet set such as Architectural, Structural, Plumbing, and so on.

Building a new project

You will need to create a template from which all of the new sheets will be created. In a new drawing I recommend x-referencing the title block for the job into the file. This eliminates the risk of parts of the

border accidentally being erased throughout the life of the sheet and it allows any part of the border to be updated globally across the project. Various aspects of the border can be completed automatically through the use of fields such as the project title, sheet title, and sheet number.

In the template file you may need to build custom properties for filling out the border. For instance, Project Navigator by default creates the “Checked By,” “Drawn By,” and “Sheet Index” fields for the sheet custom properties and the “Total Sheets” for the sheet set custom properties. You may ask yourself what is the difference between sheet and sheet set custom properties. Sheet custom properties are fields that typically change from sheet to sheet such as who drew the sheet. In contrast, Sheet Set custom properties are fields that hold true across the entire sheet set such as the construction contract number.

In the government arena, there are some fields that are not typically needed in the private sector. For instance, many military installations assign a drawing number to each sheet. This number is separate from the sheet number, such as A-1, in that it may be an eight-digit number linked to a database so it can easily be located later.

To create a custom field, start a text or m-text window. Inside the text window, right click and select insert field. In the field dialog box with all showing in the field category, select the appropriate field name paying close attention to whether this field is being created for the sheet or the sheet set. For this example we are going to step through the process of creating a custom field for the current sheet, so select the field name titled *currentsheetcustom*. In the format window on the right, select the format for the field. At the bottom of this dialog box specify a descriptive name in the custom property name window.

A special note: In order for a field to work, the field name must exist in the sheet as well as in the custom properties of the project. To avoid a short circuit, I recommend highlighting the new field name, right clicking, and selecting copy. Press OK and back out of this dialog box and then OK out of the text window. You'll now have a blank field in the new string of text. Go to your Project Navigator tool palette, right click over the main project title, and select properties (see Figure 3).

In the sheet set properties dialog box, press the edit custom properties button. The custom properties dialog box will come up; here you need to add the new field. Add the new field name by right

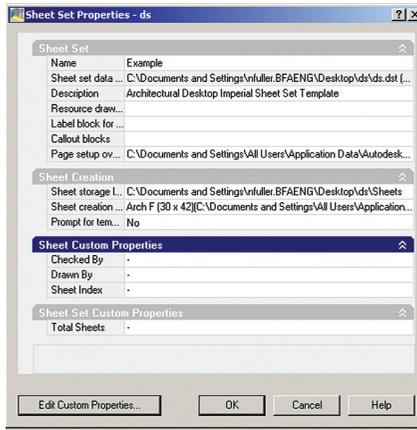


Figure 3

clicking and selecting paste in the name window. Be careful to select the correct owner for the new field. Remember that we are creating a custom field for the current sheet, so be sure that sheet is selected before hitting OK. Once this is done, back out of the dialog boxes back to the main AutoCAD window. Repeat the process for any additional fields you may need, then save the file as a .dwt file. Back in the Project Navigator palette, right click on the various subsets such as Architectural, Structural, etc., and select properties. In the bottom window of the subset properties dialog box you need to specify the new template you created. This ensures that all new sheets are created from the new template. You may need to make several templates for the different disciplines in your firm.

Now you can begin building the floor plans. Imagine a very large building in need of being split into several sections. PN has the ability to create divisions, which are used to represent the different areas of the building. In the sample building, there are five different areas—areas A through E (see Figure 4).

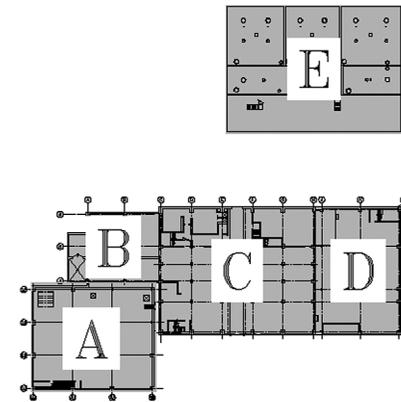


Figure 4

To create divisions in AutoCAD, switch to the Project tab on the PN palette. At the bottom of this tab you will find divisions.

Click on the Edit Divisions tab to create and manage the divisions within a project. I recommend using letters to represent the various areas. For the example shown, I took the overall floor plan and broke it up into five manageable floor plans. When the pieces are all brought together as shown in Figure 4, they fit together in their respective places. This is done by having all the pieces use the same insertion point. When editing any of the five files you will be looking at only that portion of the building, thus allowing other users in the firm to be working on another part of the building.

Next time I'll discuss bringing it all together by creating views and taking those views to build the final plotting sheets. We'll dive into streamlining the mundane processes of plotting sheets and finally archiving the project.



Nick Fuller is a mechanical designer with Bowman Foster & Associates P.C. in Norfolk, Virginia. Nick has been using AutoCAD since release 12 for DOS and has been known to dabble in

AutoLISP and customizations. For comments and questions, please feel free to contact Nick at somecadguy@verizon.net.

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Plotting with Style

Take a close look at your standards

Have you thought about your plotting standard recently? I mean really thought about it? How was it developed? Where did it come from? Does it really serve your needs? Do you have to constantly perform a workaround within your own standard in order to output exactly what you want?

Historical perspective

I'm an old-time hand drafter. I started my career with what seemed like stone knives and bear skins by today's standards. In the seventies I was drafting with pencil lead on vellum. During the eighties I used ink on mylar. Finally, in the early nineties, I started using computer-aided drafting (AutoCAD® 10) for production work.

That old CAD technology forced my firm to map AutoCAD colors to one of six pens in a pen plotter carousel. (We had to make sure we placed the pens in the correct carousel slots based on pen thickness.)

As plotters improved, we were able to define and store desired pen weights in a configuration file (.pcp, .pc2, .ctb). Armed with a color pen weight chart (usually posted on your cubicle wall), plotting became a matter of knowing which of 255 AutoCAD colors equaled which line weight. Ultimately, my firm ended up utilizing only 36 colors in our standard and that worked fine most of the time. I was always fond of calling the pen weight chart "our Secret

Decoder Ring" because without the chart, you really couldn't draft and you sure couldn't plot.

Sound familiar? Still using that system today? Many firms do. For me (and the engineering firm I work for), generating lineweights based on layer colors just wasn't getting the job done.

Color-based problems

Eighty-five percent of the drawings my firm produces are black and grayscale. The remaining percentage is black and grayscale with color enhancements and, additionally, some full-color drawings. Most of these are exhibits used either to obtain project approval from a Planning Commission/City Council or to market ourselves to a prospective client. In either case, the feedback always seems to be the same: "We would like to see friendlier colors." Worse: a "friendly" color for "City A" is never "friendly" enough for "City B." Go figure.

We kept cranking out .pcp, .pc2, and .ctb files and ended up with a boatload. Someone once said, "The nice thing about standards is that there are so many from which to choose." That was our problem with color-based plotting in a nutshell; too many standards. We needed a change and the AutoCAD 2000 plotting enhancements were just the ticket.

When most firms migrated from

AutoCAD r14 to 2000/2000i/2002, they converted their .pc2 files into .ctb files and just kept going "the way we've always done it." I can understand that thinking. Changing standards is difficult and sometimes costly. But if a standard isn't working for you, you owe it to yourself to examine alternatives.

With the advent of the .ctb (color table) file, Autodesk also gave us an alternative: the .stb (style table) file. For my firm, Named-Style based plotting turned out to be the solution. We have successfully employed a single standard .stb file now for the last four years, and it works! Sure, we have to employ agency or client standards on projects when they require them, but for the rest, one file does it all.

Rethinking the standard

For maximum flexibility we need a standard that allows us to assign a plotted lineweight as a separate setting apart from print color (black, grayscale, or color). We also want the ability to easily assign "friendly" colors on the fly and to make them even "friendlier" when we have to.

Rather than define lineweights in a plot style, we use the "Lineweight" column in the Layer Manager. Not only is the lineweight now "divorced" from the plot style, but you'll also find that there are 23 different lineweights (plus 0.00) from

Someone once said, "The nice thing about standards is that there are so many from which to choose."

which to choose. (What? More than you need? More than you want? Probably, but you've gotta love the additional power this gives you.)

To plot something in color we need a style that plots color. Once again we want maximum flexibility. Rather than assign a color to a style, we want a style that uses the layer color as the actual plotted color. That way when the layer color is assigned using the Layer Manager, the plot color is also assigned. Oh, and by the way, you do know that you have more than 16 million different colors to choose from, don't you? When selecting a layer color try the "True Color" tab (See Figure 1). There's got to be a "friendly" color in there somewhere! (Gee, and you thought 23 linetypes were too many.)

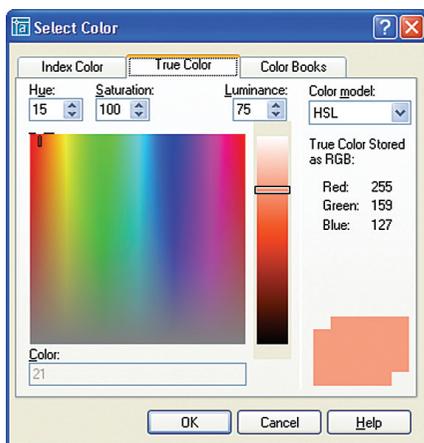


Figure 1

Finally, to plot either black or grayscale, we need plot styles that are assigned either black or grayscale. That way when a black (or grayscale) plot style is assigned to a layer, that layer will plot black (or grayscale) regardless of the color assigned to the layer.

In our standard, we now have a choice of more than 16 million colors and each one has 23 available lineweights. We can now change a group of plotted colors from gray to black (or other color) and maintain the existing lineweight of each layer. We can now select different defined grayscale percentages so that grayscale hatches no longer obliterate underlying grayscale lines. We can now take a black and grayscale construction document, add

some new layers, and turn it into an exhibit with added color for use in that important Planning Commission meeting.

Obviously, this standard works for my firm, but may not work for you. The best advice I can give is that you must examine your plotting needs and then see if you can engineer your own solution to solve any shortcomings in your current standard. Before you can do that though, you need to know some of the basics.

A drawing's "state"

The most basic thing to know is that, once created, a .dwg file can be only named-style based (.stb) or color-based (ctb), but not both. Style-based .dwgs cannot access .ctb files and color-based .dwgs cannot access .stb files. The CONVERT PSTYLES command may be used to

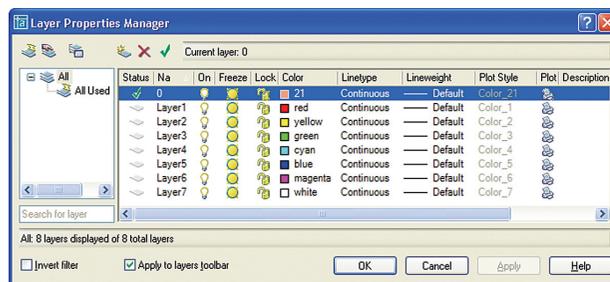


Figure 2

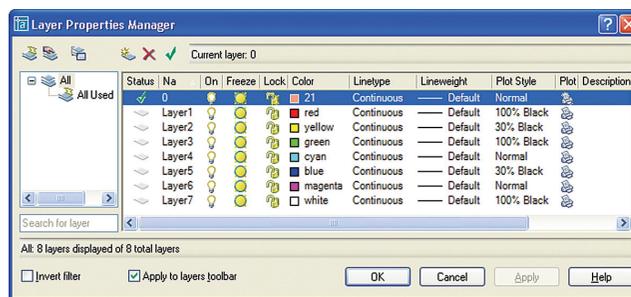


Figure 3

change a .dwg from its current style "state" to the other. But, in my opinion, you should consider a drawing's "state" to be a permanent setting. Drawing conversion should only be done when bringing a drawing in from an outside source or when pushing a drawing out to a client.

To identify the "state" of any drawing, simply open the layer manager and take a look at the "Plot Style" column. If the column is not selectable and is grayed-out, the file is color-based (See Figure 2).

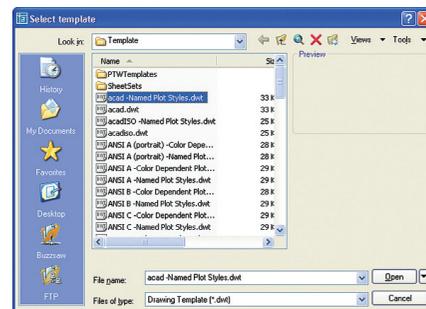


Figure 4

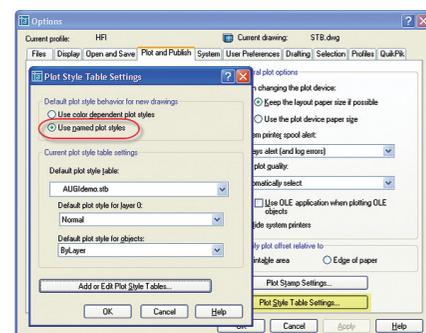


Figure 5

If the column is selectable and is not grayed-out, the file is named-style based (See Figure 3).

One interesting thing to note is that when you examine the "Plot Style" column in a color-based drawing, the grayed-out column contains plot style names! That's right, color-based drawings are style-based; you can't just change the pre-named style like you can in a named-based drawing. In a color-based drawing the plot styles are permanently linked to the

"Color" column. Change a layer's "Color" and the "Plot Style" will automatically change accordingly.

To create a new named-style based drawing from scratch, simply select one of the named plot style templates (See Figure 4) using the NEW command or, if you don't use startup templates, change your AutoCAD OPTIONS settings to use named plot style (See Figure 5). Should you prefer to use QNEW, be sure you are preconfigured to use a named plot style.

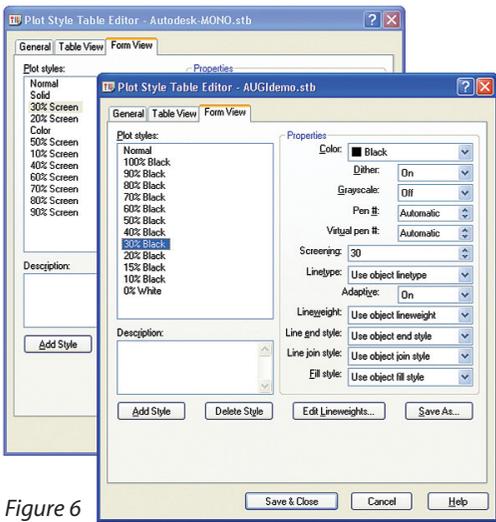


Figure 6

Plot style table settings

The next thing that you need is the .stb. You can use one that ships with AutoCAD or create one very easily. Let's see how to create a file named "AUGIdemo.stb" that will plot per the standard we've been discussing.

Access the "Plot Style Manager" from the "File" pull-down menu. Locate and open the Autodesk furnished file named "monochrome.stb." Note that even though our file "AUGIdemo.stb" is very similar to a file that comes with AutoCAD named "Autodesk-MONO.stb" (See Figure 6), for this exercise it's easier to use the more basic "monochrome.stb" file as a template.

Once "monochrome.stb" is open, switch to the "Table View" tab and save the Plot Style Table as "AUGIdemo.stb" using the "SaveAs" button. Next, click on the "Style 1" column header to highlight the entire column. With the column highlighted, click the "Add Style" button 11 (eleven) times and then switch to the

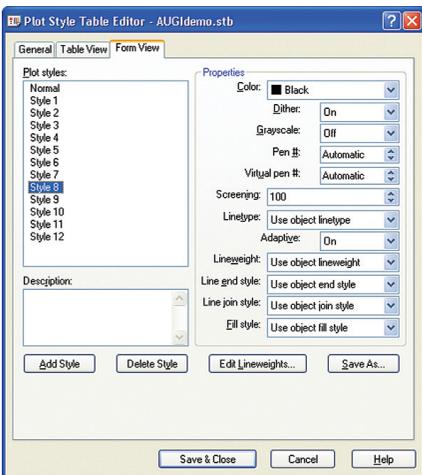


Figure 7

"Form View" tab. You should now see 13 plot styles named "Normal" and "Style 1" through "Style 12" (See Figure 7).

Starting with "Style 1", right-click on each style and rename the styles as "100% Black", "90% Black", "80% Black", "70% Black", "60% Black", "50% Black", "40% Black", "30% Black", "20% Black", "15% Black", "10% Black", and "0% White" respectively (See Figure 6).

Now highlight each named plot style, make sure the "Color" is set to black and then change "Screening" to match the name of the style. In other words, set "100% Black" to 100, "70% Black" to 70, "30% Black" to 30, and so on. Don't worry about

you'll see that Color is set to "Use object color" meaning that the plotted output will be the color assigned to the layer. If the name of the plot style Normal is not intuitive enough for you, you could always create another style with the same properties named Color. Either way, by assigning that plot style to a layer, the color of the layer is what is plotted.

Assigning the style

To assign a named plot style, open the Layer Manager. Using standard selection techniques, highlight the desired layers and then click in the Plot Style column. The "Select Plot Style" dialog box will open. Make sure the desired .stb file is set current in the "Active Plot Style Table" drop-down list and then select the desired named style from the Plot styles list (See

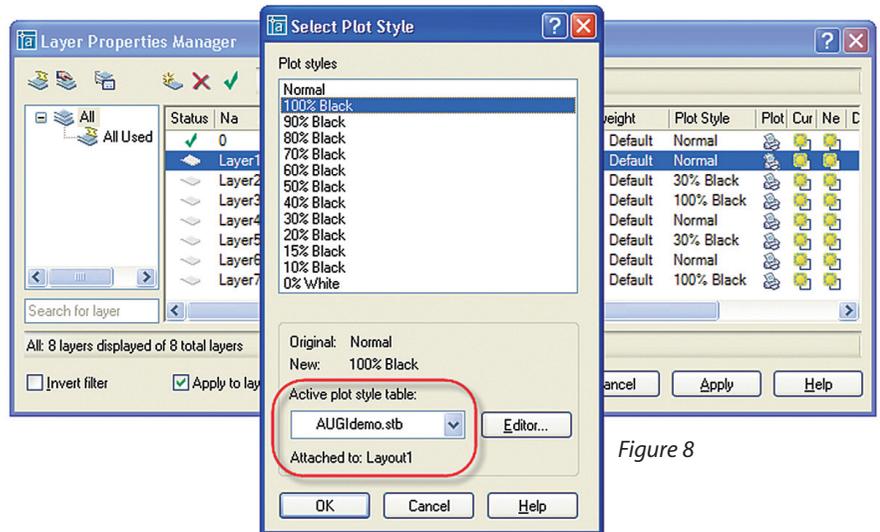


Figure 8

the "Normal" style. It cannot be changed in any way, nor can it be deleted.

The change we just made to each style determines how a layer will plot. When the style "100% Black" is assigned to a layer, that layer will plot black with a screening of 100 regardless of the layer color. The same occurs for each of the other named styles except that the black will be screened, or grayscale, in increasingly lighter shades as the "Screening" value approaches 0. The style "0% White" will result in a white line (if plotted on white paper) when the "Lines Overwrite" setting is used in your .pc3 file. This is useful when you want white text on hatched areas or dark background photos. Just remember that if your .pc3 file is set to "Lines Merge," this will not work.

Finally, for plotting any of the 16 million colors, use the plot style named "Normal." Check the properties of that style and

Figure 8). Note that you have the ability to select different .stb files for different layout tabs, should you desire to do so.

Rethink it

Are you sure "the way we've always done it" is working for you? Or is it time for a change? Take a few minutes to rethink this fundamental AutoCAD process. You just may be able to resolve issues you've been struggling with for years.



An Autodesk user since 1990, Mike Partenheimer is the corporate CAD Manager of Hall & Foreman, Inc (HFI) a southern California-based Civil Engineering firm. Mike is a member of the Civil CAD Consortium (C3) and long-time AUGI member. He can be reached at mikepartenheimer@hotmail.com.

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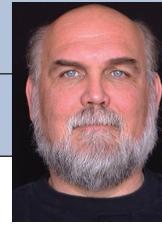
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On The Back Page



David Kingsley



Things to Come

I've had two passions throughout my life. One is Italian cooking (and eating), and the other is renewable energy. Oddly enough, there is now a connection between the two, at least in my mind.

This ain't fast food

In Italy, there are many styles of cooking based on small regions. These geographic regions are defined by areas like river basins, or by the high country between two rivers. A regional style is generally based on the ingredients available locally. Certain herbs, creatures, and vegetables thrive in certain regions, so that's what they eat. True native Italian cooks take pride in presenting a meal made of the most local ingredients.

In the U.S., the lettuce in your salad can travel 3,000 miles, and the beef in your burger even further. Succulent salmon is farmed in Norway. Wonderful apples come from New Zealand. Most foods are highly perishable and must be shipped fast and kept cold. An incredible infrastructure using hundreds of freighter jets, thousands of train cars, and hundreds of thousands of 18-wheelers deliver these foods fresh to our tables. It is estimated that for every calorie we eat, 10 calories of energy have already been consumed to deliver it.

Energy past

I'm sure many *AUGIWorld* readers remember the 1973 Oil Embargo and Three Mile Island (the closest thing to Chernobyl on U.S. soil). I was a young adult then, and those two events shaped my future. I became a firm believer in renewable energy. In reaction to the oil embargo, President Carter established the Solar Energy Research Institute in Golden Colorado in 1976. SERI has since been renamed to NREL, the National Renewable Energy Laboratory (www.nrel.gov). I moved to Colorado in 1978 to work for a company that designed and built a lot of hardware for SERI (www.barber-nichols.com). We developed a working prototype of a solar powered air

conditioning system. It required 400 square feet of collectors (a two-car garage), and the power unit was the size of a refrigerator. In 1980, when President Reagan was elected, funding for SERI was soon drastically cut (no politics, just facts) and all of that work was shelved, until now...

Energy future

In November 2004, Colorado voters soundly adopted Amendment 37, and we became the 18th state to enact energy standards. By 2015, 10 percent of Colorado's electricity will be produced by renewable



sources. The goal is to have 50,000 Colorado homes with photovoltaic cells on the roof producing 2,000 watts per home.

Like any major institution, the power companies serving Colorado change very slowly. I've been donating my time to the Colorado Energy Science Center (www.energyscience.org) that works toward speeding up that change. They focus on Colorado legislation, but the message of our energy future is universal. I developed the organization's website at www.solarcolorado.org. Feel free to visit.

And now for the bad news...

Unless you have been in a cave for the last few months, you've noticed that gas prices have spiked. By the time you read this, they might be at national average of \$2.50 for regular. Prior to the 1973 oil embargo, gas was \$0.36 per gallon. By 1978 it passed \$1.00. If you believe that gas prices will drop, like interest rates did

a few years ago, then you should head on back to your cave.

I have become aware of a disturbing event called "Peak Oil." World oil production has been on the increase for 75 years. The growth of the world economy is tied to this growth. Oil companies are envisioning a time when they cannot sustain an increase and oil production will "peak." This does not mean we will run out all of a sudden, it means that we are approaching the top of the bell shaped curve, and at some point the production of oil will begin to decrease. This all stands to reason, but here's the shocker. Most believe that this peak will occur between 2010 and 2020. What is disturbing is the level of players supporting these numbers. They are not crackpots, they are some of the most influential people on the planet. Check out these sites: www.peakoil.net www.odac-info.org and especially www.simmonsco-intl.com. Matthew Simmons advises the President regularly; just read his home page and you'll see why.

I need to say a word here about hydrogen. It is clean, but unfortunately, is not the silver bullet. Producing hydrogen requires energy. Ultimately there is a net loss of energy. In the long run we'd be better off using the energy elsewhere. Producing hydrogen with solar electricity is considered a "silver bb."

So what does this have to do with Italian cooking?

We will be living very differently in 50 years. The 3,000-mile salad will be a thing of the past. We will eat food that is grown locally. We will lead a smaller, simpler life. Instead of dreaming about a 400 hp SUV, we will be eyeballing that nice fat pig in the field and lusting after those beautiful apples in the orchard. I like to think that this could be a good thing.

David Kingsley served from 1999-2004 on the AUGI Board of Directors and is the Director of Electrons at CADPlayer Web Courseware. He can be reached at djkingsley@cad-tv.com.



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