

Bonus Chapter 6: AutoCAD Utilities

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This chapter focuses on some of the utilities that AutoCAD offers to make you a more efficient drafter. Some of these utilities will help you precisely select what you want to modify or remove from a drawing. I also show you how to use the AutoCAD calculator for math calculations and unit conversions, and even to calculate points in a drawing.

Before you complete a design it's a good idea to clean up after yourself. You can use the PURGE command to remove named objects such as blocks and text styles from a drawing if they're not being used. This makes it easier to find the named objects you are using and can help to decrease the file size on disk for your drawing files. From time to time you might encounter a drawing that might not want to open or AutoCAD might crash because of a bad drawing from a contractor, a defect in the program, or even a Windows crash. In these cases, you may need to call upon a few utilities that allow you to recover and audit a drawing file that might contain errors.

Filtering Objects during Selection

In Chapter 2 of Book II, I talk about how to select objects by using single- and multiple-object selection methods. Those options are great when you know the exact location of the objects that you want to modify, but what if you want to select, for example, all the circle objects on a specific layer? You can zoom and pan around the drawing to select each one and then modify the objects, but that can take some time. AutoCAD provides two commands, QSELECT (Quick Select) and FILTER, to locate and select objects with specific property values.

Quick Select



Quick Select was introduced with AutoCAD 2000 and has seen only minor revisions since then. When you use QSELECT, the Quick Select dialog box appears (see Figure BC6-1). From this dialog box, you can specify the objects for which you want to apply the filter, the type of object you want to find, and the property and value you want to use for the filter. You can't run QSELECT transparently while another command is active.

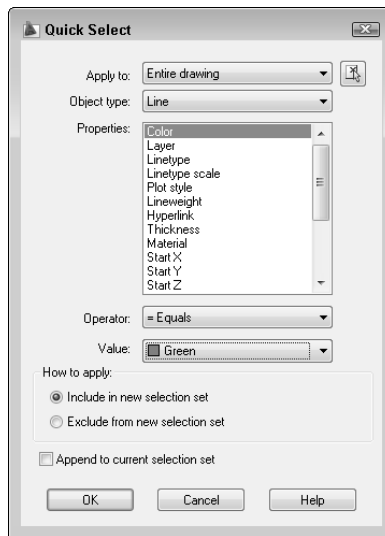


Figure BC6-1: Quick Select allows you to filter objects by a single property.

To start the QSELECT command and display the Quick Select dialog box, do one of the following:

- ◆ On the ribbon, click Home tab⇨Utilities panel⇨Quick Select.
- ◆ On the menu browser or menu bar, click Tools menu⇨Quick Select.
- ◆ At the command prompt, type **QSELECT** and press Enter.
- ◆ With no command active, right-click the drawing window and choose Quick Select from the shortcut menu.
- ◆ In the Properties palette, click the Quick Select button.

To select all the blocks with a specific name, follow these steps:

- 1. Start the QSELECT command by using one of the methods in the preceding list.**

The Quick Select dialog box is displayed.

- 2. In the Apply To drop-down list, select Entire Drawing or click the Select Objects button.**

If you click the Select Objects button, AutoCAD returns you to the drawing window where you are prompted to select the objects in the drawing to which you want to apply the filter. When you finish selecting objects, press Enter to return to the Quick Select dialog box.



You can select objects before you start the QSELECT command. If you do so, the value in the Apply To drop-down list is Current Selection. This saves the step of needing to click the Select Objects button.

- 3. In the Object Type drop-down list, select an object type.**

For example, selecting Block Reference selects blocks from the drawing. Only the object types that are contained in the drawing appear in the Object Type drop-down list. To search for all objects in the drawing, select Multiple.

- 4. In the Properties list, select a property, such as Name.**

All the available properties for the object type that you selected are displayed in the list. If you selected Multiple, only the properties common to all object types are displayed.

- 5. In the Operator drop-down list, select a comparison operator, such as = Equals.**

The = Equals operator looks for a perfect match between the value and property of the selected object type. The other operators allow you to perform not equal to, greater than, or less than comparisons. Select All selects all the objects in the drawing that have the object type you selected in Step 3.

- 6. In the Value field, enter the value to look for or select one in the predefined list. Click the down arrow in the Value field, and select one of the block names in the drawing.**

The Value field changes between a text box and a drop-down list depending on the property you select. For example, for the Name property, the values are pulled from the named objects stored in the drawing and are presented as a list of choices to choose from. If you had selected the Rotation property, you would have been presented with a text box in which to enter a value.

7. Select the Include in New Selection Set option or the Exclude in New Selection Set option.

Include in New Selection Set is the commonly used option because you want to work with the objects that result from the filter. If you want everything except what you specified, select the Exclude in New Selection Set option.

8. Select the Append to Current Selection check box to append the results of the filter to the objects that are currently selected.

This option allows you to use multiple filters to select objects in the drawing.

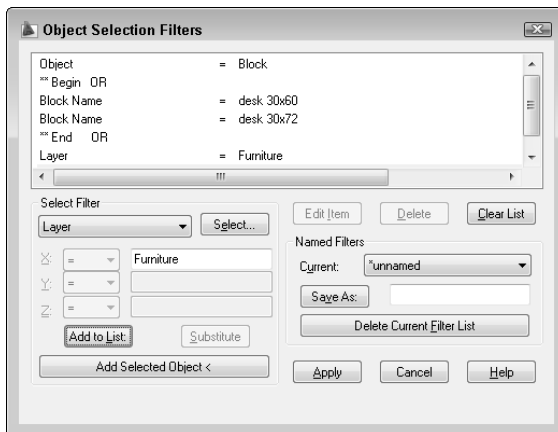
9. Click OK.

The objects in the drawing or current selection that match the filter criteria are selected, and their grips should be displayed. Now you can run one of the modify commands on the objects, such as erase or move.

Filter

Filter is the original command for selecting objects in the drawing based on their property values. Filter is a little more cumbersome than Quick Select, but it is much more powerful because it allows you to create complex filters. Complex filters can select more than one object type and can include more than one property at a time. Filter one-ups Quick Select by allowing you to run the command transparently, which means you can use the FILTER command at the *Select objects:* prompt. To use the FILTER command transparently, you add an apostrophe (') in front of the command's name. When you use the FILTER command, the Object Selection Filters dialog box appears (see Figure BC6-2).

Figure BC6-2: Filter is more complex, but with complexity comes more control.



The following steps create a filter that selects two different blocks based on their name and the layer they are on:

1. At the command prompt, type FILTER and press Enter.

The Object Selection Filters dialog box appears (refer to Figure 6-2).

2. In the Select Filter area of the dialog box, select Block from the drop-down list.

The drop-down list contains many common object types, properties, and selection conditionals. You can add other properties to a filter by clicking the Add Selected Object button and then clicking the Delete button to remove the properties you're not interested in using with the filter.

3. Click Add to List.

The select object type, property, or conditional is added to the list box at the top of the Filter dialog box.

4. In the Select Filter area, select ** Begin Or from the drop-down list to start a conditional statement and then click Add to List.

The items that begin with ** are selection conditionals, which allow you to build complex conditional statements that let you look for more than one object type or property at a time.

5. In the Selection Filter area, select Block Name from the drop-down list and then click Select.

The Select Block Name(s) dialog box appears.

6. Hold down the Ctrl key and select the blocks that you want to filter on in the drawing and then click OK. In the Filter dialog box, click Add to List.

The selected block names are added to the list box at the top of the dialog box.

7. In the Select Filter area, select ** End Or from the drop-down list and then click Add to List.

For each conditional ** *Begin* you have, you must have a ** *End*.

8. In the Selection Filter area, select Layer from the drop-down list and then click Select.

The Select Layer(s) dialog box appears.

9. Select the layer on which you want to isolate the blocks in the filter and then click OK. In the Filter dialog box, click Add to List.

10. Enter a name in the text box to the right of the Save As button to save the filter for future use and click Save As.

The filter is saved to a file named Filter.nfl in the folder C:\Documents and Settings*<user name>*\Application Data\Autodesk\AutoCAD 2009 (or AutoCAD LT 2009) \R17.2 (or R14.0)\enu\Support by default.

11. Click Apply to use the filter you just created.

You are returned to the drawing window where you can select objects to search for ones that match the criteria of the filter. If some objects are found, they are selected. To use the found objects with a modify command such as ERASE, start the command, type **P** at the *Select objects:* prompt (for the Previous selection option) to use the objects selected with the filter, and press Enter.

AutoCAD Calculator



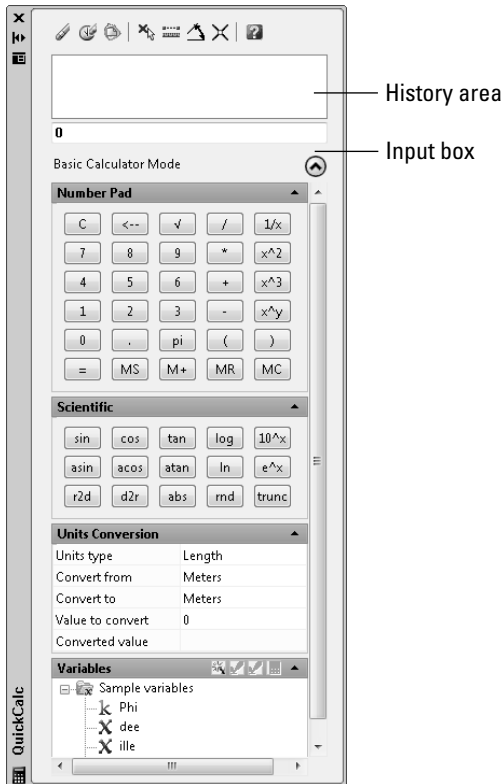
AutoCAD has two calculator commands: one is command line based while the other is a palette. The command line-based calculator is the older calculator, and you must remember all the functions that can be used, instead of having a graphical interface in which you simply click buttons to perform sets of functions on values. Due to the calculator being command line based prior to AutoCAD 2006, many users turned to third-party applications or even the calculator that comes with Windows. The problem with these other calculators is that they were not integrated into AutoCAD's workflow. The command line-based calculator that comes with AutoCAD is not available in AutoCAD LT.

This all changed with the introduction of the QUICKCALC command in AutoCAD 2006. The QUICKCALC command could do everything that the command-line based calculator could do but made it accessible through a graphical user interface that is a palette. Along with the change to a graphical interface, the calculator was integrated into AutoCAD's workflow by incorporating it into the Properties palette. The QUICKCALC command displays as either the QuickCalc palette (see Figure BC6-3) or a dialog box. When a command is running in AutoCAD, QuickCalc is displayed as a dialog box instead of as a palette.

Following is an overview of the types of expressions that QuickCalc offers.

- ◆ **Basic expressions:** The Number Pad area offers most of the basic functionality for inputting basic math expressions such as adding, subtracting, and multiplying. The area contains numbers, parentheses for grouping expressions, and buttons for storing results in memory.
- ◆ **Scientific expressions:** The Scientific area offers many common expressions related to scientific and engineering applications. These functions range from being able to calculate sine (sin) and cosine (cos) to converting radians to degrees (r2d) and degrees to radians (d2r). A total of 15 expressions are available.

Figure BC6-3:
The QuickCalc palette makes using calculations and expressions in commands easier.



- ◆ **Unit conversion expressions:** The Units Conversion area offers the ability to convert between different units of measurement. QuickCalc allows you to convert units of measurement based on length, area, volume, and angular values.
- ◆ **Geometry and variable expressions:** The Variables area and some of the tools along the top of the QuickCalc palette (or dialog box) allow you to calculate values based on existing geometry in the current drawing. You can calculate the midpoint between two endpoints (mee) or the radius of a selected circle or arc (rad). You can also create custom variables to store values that you use often.



The legacy calculator is available only in AutoCAD and can be started by using the CAL command. This command is still great when you want to use calculator expressions in a command macro.

To start the QUICKCALC command and display the QuickCalc palette or dialog box, do one of the following:

- ◆ On the ribbon, click View tab↔Palettes panel↔QuickCalc.
- ◆ On the menu browser or menu bar, click Tools menu↔Palettes↔QuickCalc.
- ◆ On the Standard or Standard Annotation toolbar, click the QuickCalc button.
- ◆ At the command prompt, type **QUICKCALC** or **QC** and press Enter.
- ◆ Press Ctrl+8.
- ◆ In the Properties palette, select a property and click the Calculator icon on the right side of the field.

Using QuickCalc with the Properties palette

The following steps demonstrate how to access the QuickCalc dialog box from the Properties palette to reduce the radius of a circle by a factor of .75.



1. On the ribbon, click View tab↔Palettes panel↔Properties.

The Properties palette appears.

2. In the drawing window, select the circle in which you want to reduce the radius by a factor of .75. Draw a circle first if one does not already exist in the drawing.

3. In the Properties palette, select the Radius property of the selected circle.

The Calculator icon is displayed on the right side of the text box for the Radius property.

4. Click the Calculator button.

The QuickCalc dialog box is displayed.

5. Click to the right of the radius value in the Input box. Type a space after the radius value and then type the expression $\cdot .75$ after the radius value (there is a space after \cdot in the expression). Press Enter.

The result of the radius multiplied by .75 is displayed in the Input box, and the full expression and result are added to the History area.

6. Click Apply.

The result of the expression is applied to the Radius property of the circle in the Properties palette and the circle is updated in the drawing window.



If the CALCINPUT system variable is set to 1, you can enter expressions directly into many of the text boxes that appear in dialog boxes and in palettes in AutoCAD. For example, if you enter $=1+2.75$ in the Thickness property of the Properties palette and press Alt+Enter, the expression is evaluated to the value of 3.75. If you're using AutoCAD and not AutoCAD LT, you can also use AutoLISP variables in your expressions.

Using QuickCalc with a command

The following steps use the QuickCalc dialog box at the *Specify radius of circle or [Diameter]* prompt of the Circle command to determine the radius of the circle that is being created.



- 1. On the ribbon, click Home tab → Draw panel → Circles flyout → Center, Radius.**

The CIRCLE command starts and displays the following prompt at the command line:

```
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
```

- 2. Specify the center point for the circle by selecting a point in the drawing window or by entering a coordinate value.**

The following prompt appears at the command line:

```
Specify radius of circle or [Diameter] <5.6964>:
```



- 3. On the ribbon, click View tab → Palettes panel → QuickCalc.**

The QuickCalc dialog box appears.

- 4. In the QuickCalc dialog box, enter the expression $0.675+1.285$ in the Input box and press Enter.**

The result of adding the two numbers is displayed in the Input box, and the full expression and result are added to the History area.

- 5. Click Apply.**

The result of the expression is placed at the command prompt.

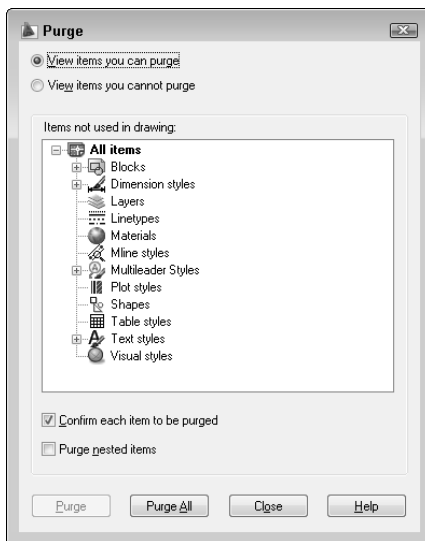
- 6. Press Enter to use the result for the radius of the circle.**

Purging Named Objects



AutoCAD drawings can fill up with styles that might be imported with blocks and drawings that get exploded or are created on-the-fly and then never used. AutoCAD doesn't know enough to remove styles that aren't being used, so you have to help AutoCAD do housekeeping at times. The PURGE command displays the Purge dialog box (see Figure BC6-4) and allows you to remove unused named objects from a drawing; it also can help you to see which styles are currently in use as well, but does not tell you where they are in use.

Figure BC6-4: The Purge dialog box helps you do spring-cleaning on unreferenced named objects in a drawing file.



With the PURGE command, you can remove the following named objects from a drawing:

- ◆ Blocks
- ◆ Dimension styles
- ◆ Layers
- ◆ Linetypes
- ◆ Materials
- ◆ Mline styles
- ◆ Multileader styles
- ◆ Plot styles
- ◆ Shapes

- ◆ Table styles
- ◆ Text styles
- ◆ Visual styles

To start the PURGE command and display the Purge dialog box, do one of the following:

- ◆ On the ribbon, click Tools tab⇨Drawing Utilities panel⇨Purge.
- ◆ On the menu browser or menu bar, click Files menu⇨Drawing Utilities⇨Purge.
- ◆ At the command prompt, type **PURGE** and press Enter.

To purge all unused blocks from a drawing, follow these steps:

- 1. Start the PURGE command by using one of the methods in the preceding list.**

The Purge dialog box is displayed.

- 3. Double-click the All Items node if it's not already expanded, and select the Blocks node.**
- 4. Click Purge.**

By default the Confirm Purge dialog box is displayed.

- 5. In the Confirm Purge dialog box, click Yes.**

The block listed in the Confirm Purge dialog box is removed from the drawing. Deselect the check mark next to Confirm Each Item to Be Purged to purge all the blocks.



Blocks can contain nested named styles and blocks. You can purge all nested items from one of the selected nodes by placing a check mark in the Purge Nested Items option. You can also use the Purge All button to purge all the currently unused named objects from the drawing. If a named object in your drawing has a nested named object, you will need to click Purge All a few times unless the Purge Nested Items is checked.

- 6. Click Close to exit the Purge dialog box.**

Auditing and Recovering Drawings

Although computers crash less frequently these days because operating systems and software are more stable, the occasional crash still occurs, and it usually happens at the least convenient of times. A number of things can cause AutoCAD to crash, such as lack of memory for the drawing file that is being opened or a corrupted drawing file.

The most common reason why you may need to jump into disaster recovery mode is a corrupted drawing file. A corrupted drawing could be due to a file that was exported out of a CAD package that can write DWG files other than AutoCAD or AutoCAD LT. AutoCAD can write bad DWG files, but this is a rare problem. In these cases, you can use the AUDIT and RECOVER commands to try and open the bad drawing file.

Auditing a drawing



Auditing a drawing is a manual process unless AutoCAD detects that a drawing needs to be recovered. Audit allows you to check the integrity of the current drawing for any potential errors in the objects contained in the drawing. Errors might be in the form of bad data formatting or even missing information that AutoCAD expected to be there. To audit a drawing, you use the AUDIT command (on the menu browser or menu bar, click File menu→Drawing Utilities→Audit; on the ribbon, click Tools tab→Drawing Utilities panel→Audit). The prompt *Fix any errors detected? [Yes/No] <N>*: appears at the command line. Specify Yes or No to automatically fix any errors that AutoCAD detects in the drawing file. Any errors that are fixed are displayed in the text window; press F2 to display the text window and review the corrections that AutoCAD made. Following is an example of output in the text window:

```
Auditing Header
```

```
Auditing Tables
```

```
Auditing Entities Pass 1  
Pass 1 600      objects audited
```

```
Auditing Entities Pass 2
```

```
AcDbDimStyleTableRecord: "14-ROMANS-115"  
                                Not in Table
```

```
Added  
Pass 2 600      objects audited
```

```
Auditing Blocks  
9      Blocks audited  
Total errors found 1 fixed 1
```

Recovering a drawing



If AutoCAD can't successfully open a drawing in AutoCAD due to a problem with the file, it will suggest performing a recovery on the drawing file. The RECOVER command is used to select a drawing file that needs to be recovered; during this process AutoCAD audits the drawing file in memory for any defects and attempts to fix them. In most cases the recovery of a drawing file is successful, but if all the errors can't be fixed, the drawing file can't be opened.

To recover a drawing, you use the RECOVER command (on the menu browser or menu bar, click File menu⇨Drawing Utilities⇨Recover; on the ribbon, click Tools tab⇨Drawing Utilities panel⇨Recover flyout⇨Recover). The Select File dialog box appears. This dialog box allows you to select the drawing you want to recover. AutoCAD audits the drawing, and if everything is fixed, a message box appears informing you that the recovery and audit of the drawing file completed successfully. Any errors that are fixed are displayed in the text window. Press F2 to display the text window and review the corrections that AutoCAD made.



As an alternative to the RECOVER command, you can use the RECOVERALL command (on the menu browser or menu bar, click File menu⇨Drawing Utilities⇨Recover Drawing and Xrefs; on the ribbon, click Tools tab⇨Drawing Utilities panel⇨Recover flyout⇨Recover with Xrefs). The difference between the two commands is that RECOVER audits only the selected drawing, while RECOVERALL audits the selected drawing and all xref files attached to the selected drawing.

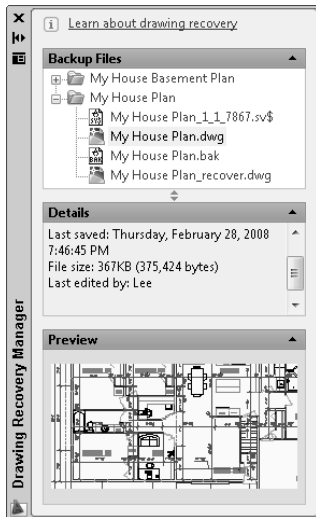
Using Drawing Recovery Manager



If AutoCAD happens to crash, you usually get the option to save any open files before AutoCAD completely closes. When you open AutoCAD the next time, the Drawing Recovery Manager palette (see Figure BC6-5) is displayed automatically. The Drawing Recovery Manager palette can be displayed also with the DRAWINGRECOVERY command (on the menu browser or menu bar, click File menu⇨Drawing Utilities⇨Drawing Recovery Manager; on the ribbon, click Tools tab⇨Drawing Utilities panel's title bar⇨Drawing Recovery Manager).

The Drawing Recovery Manager palette, which was introduced in AutoCAD 2006, lists the available files you can recover after a crash. The files that AutoCAD usually displays in the palette are the drawing files that you saved on the way out of AutoCAD during the crash, the last saved version of the drawing files, the last autosave files associated with the drawing files, and the last backup files of the drawing files that were open.

**Figure
BC6-5:**
The
Drawing
Recovery
Manager
palette.



Each drawing that was open during the crash is represented by a blue folder. Expand the folder by clicking the plus sign next to the drawing that contains the files that can be recovered. When the folder is expanded, you can select a file to see its details and view a preview of the drawing, or double-click the file to open it in a drawing window. After the file is opened, save it with the same name as the original drawing file that was open when AutoCAD crashed (to replace the corrupted file) or save the file with a different name (to create a copy of the original drawing). If one of the backup or autosave files overwrites the original file that was open during the crash, all the files that are related to that drawing are removed from the Drawing Recovery Manager palette.