Command and System Variable Changes

When you upgrade to a new release of AutoCAD, one way to quickly identify what's new is to look at a list of new commands. Also, you need to know which commands have been discontinued or significantly changed — you may have to redo some script files, AutoLISP routines, and customized menus, or simply find new ways of accomplishing old tasks. This appendix provides the information you need to smooth the upgrade learning curve.

At one time, all system variables were command-line affairs, but now most are set in dialog boxes. When a system variable is set in a dialog box, you hardly need to know about it (unless you are customizing AutoCAD). However, some system variables still need to be typed on the command line and are therefore useful to know about. This appendix lists these system variables. I also list new and changed system variables.

AutoCAD's Help contains a Command Reference that includes every command and system variable. The Readme file (Help ⇨ Help ⇨ Contents tab ⇨ Getting Information ⇨ AutoCAD 2002 Readme) also has information on command and system variable changes that didn't make it into the regular Help documentation. Another source of information is a summary of new features that you can find by choosing Help ⇨ What's New.
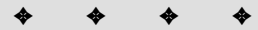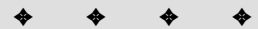
Commands

Table E-1 lists the commands that are new to AutoCAD 2000i and 2002.

Table E-1	
New Commands in AutoCAD 2000i and 2002	
Command	*Description*
3DORBITCTR	Specifies the center of the 3D Orbit display.
ASSIST	Opens the Active Assistance help window.
ATTSYNC	Prompts you to select a block and then updates all instances of that block with the current attributes defined for the block.
CHECKSTANDARDS	Opens the Check Standards dialog box so you can check a drawing against its standards file.
CLOSEALL	Closes all open drawings.
CONVERTCTB	Converts a color-dependent plot style table (CTB) to named plot style tables (STB), one for each color with unique plot properties, another for each group of colors that have the same plot properties, and a default named plot style called NORMAL.
CONVERTPSTYLES	Converts the current drawing to either named or color-dependent plot styles, whichever type the drawing is not currently using.
CUSTOMIZE	Opens the Customize dialog box in which you can customize toolbars, buttons, and keyboard shortcuts.
DBLCLKEDIT	Determines whether a dialog box opens when you double-click an object so you can edit the object. A value of ON means that a dialog box opens; OFF means that no dialog box opens.
DIMDISSOCIATE	Disassociates dimensions with objects and converts the dimensions into nonassociated dimensions.
DIMREASSOCIATE	Associates dimensions with objects.
DIMREGEN	Updates the locations of dimensions relative to their objects. For example, if you use a wheel mouse to pan or zoom with model space active in a layout, the dimensions are no longer in their proper locations. DIMREGEN fixes this.
EATTEDIT	Opens the Enhanced Attribute Editor in which you can change attribute properties and values.
EATTEXT	Starts the Attribute Extraction Wizard so you can extract the data from block attributes.
ENDTODAY	Closes the Today window.

Command	Description
ETRANSMIT	Opens the Create Transmittal dialog box, so you can e-mail a drawing and its related files.
JUSTIFYTEXT	Changes the justification of text without changing its location.
LAYERP	Equivalent to clicking the new Layer Previous button. Not only switches to the last current layer but restores previous layer settings, such as color or linetype and states, such as off/on or frozen/thawed.
LAYERPMODE	When on, AutoCAD tracks layer changes so you can use the LAYERP command or Layer Previous button. When off, AutoCAD does not track layer changes.
LAYERTRANS	Opens the Layer Translator in which you can specify translation (mapping) from one layer to another.
MEETNOW	Opens the Meet Now feature (Microsoft's Net Meeting within AutoCAD) so that you can share a session of AutoCAD and communicate with other people via the Internet.
PLOTSTAMP	Adds a plot stamp on a plotted drawing, that is, a label containing some or all of the following: drawing name, layout name, date and time, Windows login name, plotting device, paper size, and plot scale.
PUBLISHTOWEB	Starts the Publish to Web Wizard, so you can publish drawings to a Web page.
RMLIN	Inserts markups from an RML file, created in Autodesk's Volo View, into a drawing.
SCALETEXT	Scales text without changing its location.
SPACETRANS	Converts length values between model and paper space, usually for the purpose of displaying text heights the proper size in a scaled layout.
STANDARDS	Displays the Configure Standards dialog box so that you can associate a standards file with the drawing.
TODAY	Opens the Today window.

Discontinued commands

Table E-2 shows the commands that have been discontinued in AutoCAD 2002, along with any available alternatives. Some of these discontinued commands still work because they have been included as an alias in *acad.pgp,* but they actually execute the alternative command.

<table>
<tr><td colspan="2" align="center">Table E-2
Discontinued Commands in AutoCAD 2002</td></tr>
</table>

Command	Alternative
PSDRAG	PostScript files are now configured in a PC3 file using the Plotter Manager.
PSFILL	PostScript files are now configured in a PC3 file using the Plotter Manager.
PSIN	PostScript files are now configured in a PC3 file using the Plotter Manager.
PSPROLOG	PostScript files are now configured in a PC3 file using the Plotter Manager.
PSQUALITY	PostScript files are now configured in a PC3 file using the Plotter Manager.

Changed commands

The commands in Table E-3 now open a dialog box.

<table>
<tr><td colspan="2" align="center">Table E-3
Changed Commands in AutoCAD 2000i and 2002</td></tr>
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Command	Changes
ARRAY	Now opens a dialog box. Use **-array** to create arrays on the command line.
PURGE	Now opens a dialog box. Use **-purge** on the command line.

System Variables

For more information on new and changed system variables, see the AutoCAD Command Reference.

New system variables

Table E-4 shows the new system variables.

Table E-4
New System Variables in AutoCAD 2002

System Variable	Description
DIMASSOC	Replaces the DIMSASO system variable (which still exists). Sets the associativity for dimensions. 0 = exploded dimensions; 1 = non-associative dimensions; 2 = associative dimensions.
HALOGAP	Specifies the percentage of an inch to shorten a haloed line—a line in a 3D model that would otherwise be hidden using the HIDE command. Haloed lines are shortened to set them off from unhidden lines and display only when you use the HIDE command or the Hide option of SHADEMODE. Does not apply to the lines that create the back (hidden) sides of solids or surfaces. Also see OBSCUREDCOLOR and OBSCUREDLTYPE.
HIDETEXT	Use the On setting, the default, if you want text to be hidden behind other objects and to hide other objects when you use the HIDE command. If you use an Off setting, text will not hide other objects, nor be hidden, unless it has a thickness.
LAYOUTREGENCTL	Determines regeneration when you switch to another layout/model tab. A value of 0 regenerates the drawing each time you switch tabs. A value of 1 suppresses regenerations (by saving the display list to memory) between the Model tab and the last layout you used. A value of 2 (the default) regenerates the drawing the first time you switch to each tab only but suppresses regenerations afterward.
OBSCUREDCOLOR	Determines the color of obscured lines, specified by using one of the standard AutoCAD color index numbers (from 0 to 256). An obscured line is a line that would otherwise be hidden when you use the HIDE command and appears only when you turn on OBSCUREDLTYPE by giving it a value other than 0 and when you use the HIDE command or the Hide option of SHADEMODE. Also see HALOGAP.
OBSCUREDLTYPE	Determines the linetype of obscured lines. Also turns on obscured lines. Also see OBSCUREDCOLOR and HALOGAP. 1 creates a solid line, 2 creates a dashed line, 3 creates a dotted line. For more options, see the AutoCAD Command Reference in Help.
PROXYWEBSEARCH	Specifies how AutoCAD checks for Object Enablers, which display custom objects in AutoCAD drawings even when the ObjectARX application that created them is unavailable. Object Enablers are checked via your Internet connection and Point A. You can also set this system variable by choosing Tools ⇨ Options, clicking the System tab, and going to the Live Enabler Options.

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Table E-4 *(continued)*	
System Variable	**Description**
REMEMBERFOLDERS	Controls in which folder file selection dialog boxes, such as the Select File dialog box, open by default. A value of 1, the default, remembers the last used folder and opens subsequent file selection dialog boxes in the same folder. A value of 0 uses the path specified by an AutoCAD icon, if that's how you start AutoCAD. See Appendix A for information on creating Start In paths for an icon.
STARTUPTODAY	A value of 1, the default, displays the Today window when you start AutoCAD or a new drawing. A value of 0 displays the traditional Startup dialog box. Because displaying the Today window takes a great deal of time, you may want to turn it off at the beginning of script files or AutoLISP routines that run through many drawings. You can turn it on again at the end of the script or routine.
WHIPTHREAD	Determines whether AutoCAD uses an additional processor, if one exists on your computer. Settings range from 0 (no additional processor is used) to 3 (the default—additional processor is used for regens and redraws).
WMFFOREGND	Works in tandem with the WMFBKGND system variable. Controls the foreground (line) color of AutoCAD objects when the objects are brought into another application using the WMFOUT command, copied and pasted as a Windows metafile, or dragged and dropped. WMFFOREGND is applicable only when you set WMFBKGND to 0, which makes the background color transparent. A value of 0, the default, swaps foreground and background colors if necessary to make the foreground color, that is, the objects, darker than the background color. A value of 1 does the opposite, so that the foreground color is lighter than the background color.

Discontinued system variables

Table E-5 shows system variables that have been discontinued along with available alternatives.

Table E-5 Discontinued System Variable in AutoCAD 2002	
System Variable	**Alternative**
DIMASO	Obsolete; replaced by DIMASSOC.

Changed system variables

Table E-6 lists some changed system variables.

Table E-6	
Changed System Variables in AutoCAD 2002	
System Variable	*Change*
DIMASSOC	Supports AutoCAD 2002's new associative dimensions.

System variables often used on the command line

If you write AutoLISP routines, you need to work with all system variables. However, for most drawing tasks, you only need to know the system variables that you usually do not set in a dialog box. These are the system variables that you might want to type on the command line, and thus you should know them by name. Tables E-7 through E-14 list system variables by function so you can find them even if you don't know their names.

Table E-7	
System Variables for 3D Drawing	
System Variable	*Description*
COMPASS	Turns the compass, a 3D orbit aid, on and off in the current floating viewport.
DISPSILH	Turns on and off the display of silhouette curves of 3D objects in wireframe display.
FACETRES	Sets the smoothness of 3D shaded and rendered objects and objects with hidden lines removed.
HALOGAP	Specifies the percentage of an inch to shorten a haloed line — a line in a 3D model that would otherwise be hidden using the HIDE command. Haloed lines are shortened to set them off from unhidden lines and display only when you use the HIDE command or the Hide option of SHADEMODE. Does not apply to the lines that create the back (hidden) sides of solids or surfaces. Also see OBSCUREDCOLOR and OBSCUREDLTYPE.

Continued

Table E-7 *(continued)*	
System Variable	**Description**
HIDETEXT	Use the On setting, the default, if you want text to be hidden behind other objects and to hide other objects when you use the HIDE command. If you use an Off setting, text will not hide other objects, nor be hidden, unless is has a thickness.
ISOLINES	Sets the number of lines used to represent 3D objects in wireframe. The default is 4 but values can range from 0 to 2,047. A value of 8 is a good balance between display and speed.
OBSCUREDCOLOR	Determines the color of obscured lines, specified by using one of the standard AutoCAD color index numbers (from 1 to 255). An obscured line is a line that would otherwise be hidden when you use the HIDE command and appears only when you turn on OBSCUREDLTYPE by giving it a value other than 0 and when you use the HIDE command or the Hide option of SHADEMODE. Also see HALOGAP.
OBSCUREDLTYPE	Determines the linetype of obscured lines. Also turns on obscured lines. Also see OBSCUREDCOLOR and HALOGAP. 1 creates a solid line, 2 creates a dashed line, 3 creates a dotted line. For more options, see the AutoCAD Command Reference in Help.
SHADEDGE	Controls the way edges are shaded. The setting of 0 shades faces but does not highlight edges; 1 shades faces and highlights edges with the background color; 2 doesn't shade faces, but hides hidden lines, and shows edges using the object color; 3 highlights edges only, using the background color. The default is 3 although 0 or 1 are probably more useful settings.
SHADEDIF	Sets the percent of diffuse reflective light to ambient light. The default is 70. Values can range from 0 to 100.
SURFTAB1	The number of tabulations used in the RULESURF and TABSURF commands. For the REVSURF and EDGESURF commands, sets the M direction.
SURFTAB2	Sets the N direction for the REVSURF and EDGESURF commands.
SURFU	The surface density in the M direction for polyface meshes.
SURFV	The surface density in the N direction for polyface meshes.
UCSFOLLOW	Determines whether AutoCAD returns you to plan view when you change the UCS. Choose 1 to return to plan view. The default is 0.
UCSORTHO	Sets whether the related UCS is automatically restored when you changed to an orthographic view such as Top, Bottom, Left, and so on.

Table E-8
System Variables for Attributes

System Variable	Description
ATTDIA	Determines whether you get a dialog box when you use the INSERT command to insert a block that contains attributes. A setting of 0 turns off the dialog box.
ATTMODE	Determines whether attributes are visible or invisible.
ATTREQ	If you set this to 0, AutoCAD uses default attribute values when you insert a block with attributes. If you set it to 1, the default, AutoCAD prompts you for values.

Table E-9
System Variables for Dimensioning

System Variable	Description
DIMASSOC	Replaces the DIMSASO system variable (which still exists). Sets the associativity for dimensions. 0 = exploded dimensions; 1 = non-associative dimensions; 2 = associative dimensions.
DIMSHO	When on, the default, AutoCAD continually recomputes associative dimensions as you drag an object.

Table E-10
System Variables for Drawing Aids

System Variable	Description
BLIPMODE	Turns blips on and off.
EXPERT	Controls whether certain prompts are issued, according to the following values:
	0 Issues all prompts normally.
	1 Suppresses "About to regen, proceed?" and "Really want to turn the current layer off?"
	2 Suppresses the prompts for 1 plus "Block already defined. Redefine it?" and "A drawing with this name already exists. Overwrite it?"

Continued

Table E-10 *(continued)*

System Variable	Description
	3 Suppresses the prompts for 1 and 2 plus those displayed when you try to load a linetype that's already loaded or create a new linetype in a file that already defines it.
	4 Suppresses the prompts for 1, 2, and 3 plus those displayed by UCS Save and VPORTS Save if the name you supply already exists.
	5 Suppresses the prompts for 1, 2, 3, and 4 plus those issued by the DIMSTYLE Save option and DIMOVERRIDE if the dimension style name you supply already exists.
	When EXPERT suppresses a prompt, AutoCAD acts as if you entered **y** at the prompt. Warning: EXPERT's value affects scripts, menu macros, and AutoLISP routines.
MAXACTVP	The maximum number of viewports that can be regenerated at one time. The default is 64.
MBUTTONPAN	Lets you use the third button or wheel on your mouse or digitizing puck to pan, overriding the menu definition.
UNITMODE	Determines how fractional, feet-and-inches, and surveyor's angle units are displayed on the status line. Set it to 1 to display them in input format, as in 3'2-1/2". A value of 0, the default, displays the same number as 3'-2 1/2".
VISRETAIN	A setting of 0 means that xrefs take on the layer settings (on/off, thawed/frozen, and so on) in the current drawing. A setting of 1 means that the layers settings in the xref drawing take precedence, meaning that these settings persist each time you open a drawing with the xref. The default is 1.
WHIPARC	Sets whether arcs and circles are displayed as a series of vectors (the default) or as true, smooth curves.
WORLDVIEW	Determines whether the UCS changes to the WCS when you use DVIEW or VPOINT. If you set it to 0, the UCS remains unchanged. If you set it to 1, the default, the UCS switches to the WCS during the DVIEW or VPOINT command.
ZOOMFACTOR	Controls the incremental change in zoom created by one IntelliMouse wheel action.

Table E-11
System Variables for Editing

System Variable	Description
EXPLMODE	The default, 1; lets you explode nonuniformly scaled blocks (the X and Y scales are different). Set to 0 to disallow exploding nonuniformly scaled blocks.
MIRRTEXT	By default, the MIRROR command mirrors selected text. Set to 0 to keep the text reading from left to right.
PICKSTYLE	Determines the way groups and hatches can be selected, according to the following values: 0 No group selection. Hatches are selected without their boundaries. 1 Group selection. Hatches are selected without their boundaries. 2 No group selection. Hatches are selected along with their boundaries. 3 Group selection. Hatches are selected along with their boundaries.
WMFBKGND	Lets you control the background of Windows metafile objects, whether created using WMFOUT, copied to the clipboard and pasted as a metafile, or dragged and dropped as a metafile. You can set the background to transparent or have it take on the background color of your drawing screen (the default).
WMFFOREGND	Works in tandem with the WMFBKGND system variable. Controls the foreground (line) color of AutoCAD objects when the objects are brought into another application using the WMFOUT command, copied and pasted as a Windows metafile, or dragged and dropped. WMFFOREGND is applicable only when you set WMFBKGND to 0, which makes the background color transparent. A value of 0, the default, swaps foreground and background colors if necessary to make the foreground color, that is, the objects, darker than the background color. A value of 1 does the opposite, so that the foreground color is lighter than the background color.

Table E-12
System Variables for Information and Customization*

System Variable	Description
ACADPREFIX	The path of the ACAD environment.
ACADVER	The AutoCAD version number.
AREA	The most recent area calculated by AREA, LIST, or DBLIST.

Continued

Table E-12 *(continued)*

System Variable	Description
AUDITCTL	Set to 1 to create an audit report file, or to 0 if you don't want a report.
CDATE	Stores the date and time.
CMDACTIVE	Stores whether a regular command, transparent command, script, or dialog box is active.
CMDNAMES	The name of the active regular and transparent commands.
DATE	The date and time in Julian format.
DBMOD	Saves the types of modifications done in the drawing. Its value is the sum of the following: 1 Objects have been modified. 4 System variables have been modified. 8 A window has been modified. 16 The view has been modified.
DCTCUST	The custom spelling dictionary file.
DCTMAIN	The main spelling dictionary file.
DELOBJ	Determines whether objects used to create other objects are maintained in the drawing database — for example, when using the EXTRUDE command to create a 3D object from a 2D object. The default, 1, deleted these objects. A value of 0 retains them.
DISTANCE	The most recent distance calculated by the DIST command.
DWGNAME	The drawing name.
DWGPREFIX	The path for the drawing.
DWGTITLED	0 if your drawing has not been named; 1 if it has.
EXTMAX	The upper-right corner of the drawing extents.
EXTMIN	The lower-left corner of the drawing extents.
FILEDIA	Turns on and off the display of dialog boxes that deal with files, such as the Select File dialog box. Set to 0 to turn off file dialog boxes.
INSNAME	Stores a default block name for INSERT.
LASTANGLE	The end angle of the arc you drew most recently.
LASTPOINT	The most recent point entered.
LASTPROMPT	Stores the most recent text string that appears on the command line, including user input.
LIMMAX	The upper-right corner of the drawing limits.

System Variable	Description
LIMMIN	The lower-left corner of the drawing limits.
LOCALE	The ISO language code of the current AutoCAD version.
LOGINNAME	The user's name required in order to log in on a network system.
MENUECHO	Sets the display (echo) of menu items. By default (0), all items are echoed. Its value is the sum of these values: 1 Suppresses echo of menu items (^P in a menu item toggles echoing). 2 Suppresses display of system prompts during menu use. 4 Disables ^P menu toggle. 8 Displays input/output strings; used for debugging DIESEL macros.
MENUNAME	The current menu name.
OLEHIDE	Determines whether OLE objects are visible onscreen and printed/plotted. By default (value = 0), they are visible. The setting of 1 makes them visible in paper space only, 2 makes them visible in model space only, and 3 suppresses their visibility.
PERIMETER	The most recent perimeter calculated by AREA, LIST, or DBLIST.
PFACEVMAX	The maximum number of vertices per face.
SCREENSIZE	The size, in pixels, of the current viewport.
SHPNAME	The default shape name.
STARTUPTODAY	A value of 1, the default, displays the Today window when you start AutoCAD or a new drawing. A value of 0 displays the traditional Startup dialog box.
TDCREATE	The date and time the drawing was created.
VIEWCTR	The center of the view of the current viewport.
VSMAX	The upper-right corner of the virtual screen.
VSMIN	The lower-left corner of the virtual screen.
WORLDUCS	Stores whether the UCS is the same as the World Coordinate System. A value of 0 means that the UCS is different; a value of 1 means that the UCS and the WCS are the same.
XREFCTL	Set to 1 to create an xref log file. Set to 0 if you don't want to create a log file.

* Many of these system variables are read-only, which means that they just provide information and can't be changed.

Table E-13
System Variables for Object Creation

System Variable	Description
CECOLOR	Sets the current color for new objects.
CELWEIGHT	Sets the current lineweight for new objects.
CELTSCALE	Sets the current linetype scale for new objects.
CELTYPE	Sets the current linetype for new objects.
PELLIPSE	A setting of 0, the default, creates a true ellipse. Set to 1 to create a polyline representation of an ellipse.
PLINEGEN	When set to 0, the default, linetypes start each vertex of a polyline with a dash. When set to 1, the linetype is generated in a continuous pattern regardless of the vertices.
POLYSIDES	The default number of sides for polygon.
SKPOLY	If set to 0, SKETCH creates lines. If set to 1, it creates polylines.
SPLFRAME	Sets spline-fit polyline display. If 0, the default, doesn't display the control polygon for splines and spline-fit polylines, displays the fit surface of a polygon mesh but not the defining mesh, doesn't display the invisible edges of 3D faces or polyface meshes. If 1, displays the control polygon for splines and spline-fit polylines, displays the defining mesh of a polygon mesh but not the fit surface, and shows invisible edges of 3D faces and polyface meshes.
SPLINESEGS	Sets the number of line segments that each spline generates. A higher number results in a curve that more precisely matches the frame.
SPLINETYPE	Determines the type of spline curves created by the Spline option of the PEDIT command. Use 5 for a quadratic B-spline and 6 (the default) for a cubic B-spline.
SURFTYPE	Sets the type of surface fitting used by the Smooth option of the PEDIT command. Set 5 for a quadratic B-spline, 6 (the default) for a cubic B-spline, and 8 for a Bézier surface.

Table E-14 **System Variables for Text**	
System Variable	*Description*
FONTALT	Sets an alternative font that AutoCAD will use if the specified font cannot be found.
FONTMAP	Specifies a fontmap file to use for substituting fonts.
MIRRTEXT	Specifies whether text is mirrored like other objects or retains its typical left-to-right orientation.
TEXTSIZE	Stores the default or most recent height for text styles without a fixed height.
TEXTSTYLE	The current text-style name.

❖   ❖   ❖