

AutoCAD Release 10  
README.DOC for XENIX

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This file highlights some important changes in AutoCAD Release 10 and includes last-minute information that missed the deadline for the printed manuals.

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## COMPATIBILITY WITH APPLICATION PROGRAMS - FLATLAND

- \* Some functions, including entity selection and object snap, have changed subtly in Release 10 to accommodate 3D, viewports, and perspective views. These changes have little effect when you're using AutoCAD interactively. However, programs that generate AutoCAD command sequences may get unexpected results. Therefore, the FLATLAND system variable is provided to let you maintain compatibility with programs that have not yet been updated to recognize and take advantage of Release 10's new capabilities.

When FLATLAND is zero, Release 10's 3D features are fully enabled. When FLATLAND is nonzero, compatibility with previous versions is maintained. The initial FLATLAND setting for a drawing is shown in the following table.

Condition	Initial setting
Editing an old drawing (produced by AutoCAD Release 9 or earlier)	FLATLAND = 1
Editing a drawing created or previously edited with AutoCAD Release 10	As saved with the drawing
Creating a new drawing (with "name=")	FLATLAND = 0
Creating a new drawing (without "=", or with "name=prototype")	Set by prototype drawing -- in the standard ACAD.DWG prototype, FLATLAND is zero.

If you're using a third-party application that has not yet been updated to take advantage of Release 10's new features, you may experience problems with entity selection or object snap if you set FLATLAND to zero. In such cases, we recommend that you keep FLATLAND set to 1 (set this in the prototype drawing, if necessary) and use only the World Coordinate System when editing with AutoCAD Release 10.

NOTE: The FLATLAND system variable is a temporary conversion aid, and will be removed in the next major update of AutoCAD. If you've written your own application and it fails to work properly with FLATLAND set to zero, run with FLATLAND set to 1 if necessary, but UPDATE YOUR APPLICATION AS SOON AS IS PRACTICAL, because future versions of AutoCAD will always operate as though FLATLAND was zero.

See the Release 10 Reference Manual (Appendix D) for further details.

- \* In previous versions, DXF files and the AutoLISP (entget) function supplied a 38 group containing an entity's elevation. Release 10 continues to supply this field if FLATLAND is set to 1 and the entity lies in the World Coordinate System's XY plane. Otherwise, the 38 group is omitted and the elevation is supplied as the Z coordinate of the object's 3D points.

This poses a small problem for Polylines, since the 38 group was supplied with the POLYLINE (header) entity, but that entity has no associated point. Therefore, if FLATLAND is zero or the Polyline doesn't lie in the XY plane of the WCS, a "dummy" 3D point (10 group) is supplied with the POLYLINE header entity, with its Z coordinate being the only item of interest.

## PERSPECTIVE VIEWS

- \* Although most commands are now permitted in perspective views, use of a pointing device to designate points is generally prohibited in such views. You can type in coordinates using the keyboard, but if you use a pointing device to enter points, AutoCAD displays the message:

Pointing in perspective view not allowed here.

Entity selection is the only activity that allows pointing device input in perspective views. Even then, certain commands (BREAK, FILLET, CHAMFER, TRIM, EXTEND, UCS Entity, and DIM by picking) use the pick point for calculations and don't permit the point to be supplied by means of a pointing device in a perspective view.

- \* If you use the PLOT/PRPLOT "Extents" option with a perspective view active, and the CAMERA position is within the drawing extents, AutoCAD displays the message:

PLOT and PRPLOT extents incalculable, using Display  
and proceeds as though you had used "PLOT/PRPLOT Display".

- \* When plotting a perspective view, an explicit plot scale makes little sense, so AutoCAD ignores it. If you specify a scale factor, AutoCAD displays the message:

\*\*Plot of perspective view has been scaled to fit available area.\*\*

and proceeds as though you had responded to the "Specify scale" prompt with "Fit". Use the "paper size" option in the plotting configuration dialogue of the PLOT and PRPLOT commands to enter a specific output size for your desired plot area.

## MISCELLANEOUS CHANGES AND CORRECTIONS

- \* The default setting for the WORLDVIEW system variable is now 1. This variable affects the VPOINT and DVIEW commands, as described in Chapter 6 of the AutoCAD Reference Manual.
- \* The AutoLISP (entsel) function returns a list that includes the point by which the entity was selected. This point is expressed in UCS coordinates.
- \* The AutoLISP Programmer's Reference warns application developers to avoid referencing the ATOMLIST symbol list. Performance-related issues in AutoLISP Release 10 make this more important than ever.
- \* It is now possible to OSNAP to the insertion point of an Attribute within a Block. If you use "INSERTion" object snap and point to an Attribute within a Block, the Attribute's insertion point is now chosen rather than that of the Block. Note that if a Block consists entirely of Attributes, there is now no way to object snap to the Block's insertion point.
- \* The FLIP SCREEN function key, all transparent commands, and several AutoLISP functions are now disabled during certain commands, since some displays must redraw when flipping from text mode to graphics mode, and a redraw occurring at critical times could result in an incorrect image.

The commands affected are VPOINT, DVIEW, and ZOOM Dynamic. During these commands, the FLIP SCREEN key and transparent commands are disabled, as are the following AutoLISP functions:

(entdel)	(getangle)	(getpoint)	(grtext)
(entmod)	(getcorner)	(graphscr)	(redraw)
(entsel)	(getdist)	(grclear)	(ssget)
(entupd)	(getorient)	(grdraw)	(textscr)

- \* In previous versions, the HIDE command had trouble with non-planar 3D Faces and entities that appeared as chevrons or bow-ties when viewed from particular points in space. Sometimes they were hidden incorrectly, and sometimes they were simply drawn as wire-frame images (not hidden). In Release 10, such objects are hidden in a more consistent manner.

HIDE now detects anything that looks like a chevron or a bow-tie from the current viewing point, and breaks it into two triangles (with an invisible joining line) for the purpose of hidden-line removal. Any other non-planar face is broken into two triangles along an arbitrarily chosen diagonal.

- \* Polar arrays are handled differently in Release 10. In previous versions, a "No" response to the "Rotate objects as they are copied?" prompt caused each object to be replicated using its own reference point. This resulted in the entities moving independently, rather than retaining their relation to each other as they would if you replied "Yes". (To visualize the effect, draw circles around the array's center point and passing through one endpoint of each selected line, through the center of each selected circle, and so on; then make copies of each entity along the circumference of its own circle.)

In Release 10, all selected entities are replicated using the same reference point, so they move as a rigid group, staying in the same relation in all the copies. The reference point for all the entities is that of the last entity in the selection-set. Therefore, if you pick the entities one by one, the last one you pick provides the reference point for all of them.

If you pick the entities by a Window or Crossing box, the last one in the selection-set is arbitrary. You can, however, remove one entity from the set and then add it back in, forcing it to be the last in the selection-set.

The method recommended in the Reference Manual still works: make a Block of the entities and create a polar array of the Block. This method gives greater flexibility, since the reference point for making the array is the Base (insertion) point of the Block, which can be any point you like. It does, however, create an array of Block References, which must then be exploded if you want the individual entities replicated.

- \* The following clarifies the purpose of the EXPERT system variable.

Some commands issue "are you sure?"-type (confirmation) prompts when specific circumstances make the operation questionable, such as when you use the LAYER command to turn off the current layer. When you write scripts, menu macros, or AutoLISP programs that issue command sequences, it is impractical to include responses to these prompts, since they only need to appear in situations where the user has an opportunity to answer the

question about the operation. Using the EXPERT system variable, you can suppress these confirmation prompts and let the script, macro, or AutoLISP program proceed with the operation unhindered.

If EXPERT is zero, all prompts are issued normally. Nonzero values suppress successively more confirmation prompts; as new prompts are added to AutoCAD, higher EXPERT values control those prompts as well as the older prompts. In Release 10, an EXPERT value of 4 is sufficient to suppress all special-situation confirmation prompts.

- \* In the "Maintaining Multiple AutoCAD Directories" section of the AutoCAD Reference Manual (Section B.2.1) it states

"If AutoCAD doesn't find the file acad.cfg in the current directory, the configuration directory or the system directories, then it performs an initial configuration..."

This is incorrect. AutoCAD does not look in the current directory, but instead, examines the directory where the AutoCAD program was executed.

- \* The description of the Arbitrary Axis Algorithm on page 410 of the AutoCAD Reference Manual has an error in the first line.

That line currently reads:

```
If (Nx < 1/64) and (Ny < 1/64) then
```

It should read:

```
If (abs (Nx) < 1/64) and (abs (Ny) < 1/64) then
```

Where "abs" is defined as an "absolute value function" in the user application code.

- \* Section 10.1.12.1 of the AutoCAD User Reference states "...definition points are not plotted unless you turn the 'DEFPOINTS' layer 'on'". This is incorrect. The defining points for associative dimensions are never plotted regardless of the on/off status of the DEFPOINTS layer.

If you wish to plot the defining points of associative dimensions, you must rename the DEFPOINTS layer. However, unless you rename it back to DEFPOINTS after you plot, any additional dimensions created will make a new DEFPOINTS layer. You then will have more than one layer with associative dimension defining point information.

- \* Page 93 of the AutoCAD Reference Manual indicates that when using a circle as a path curve in constructing a ruled surface with the RULESURF command, the start point of the vertices will be located at the 0 degree quadrant. This is only true for circles oriented in the WCS. UCS relative circles can be used for generating a ruled surface, but the start point is not predictable. If you need to control the start location of ruled surfaces of this type, you should use a polyline arc of 360 degrees instead of a circle. The start point of the polyline arc is predictably consistent and will allow you to have control when creating ruled surfaces.

## KNOWN PROBLEMS

- \* AutoCAD's dimensioning feature has the following known problems.
  - \* Changing the elevation of an associative dimension entity block will not effect the elevation of the defining points located on the DEFPOINTS layer. When adjusting dimension entities edited in this manner from a different viewpoint, you must be sure to select the defining points before you make the edit.
  - \* When setting DIMTOH to "OFF", DIAMeter dimensions fail to place the tick mark at the end of the extension line when the circle or arc are small enough to force the text outside and the angle of the extension line is greater than 15 degrees from horizontal.
  
- \* The following list identifies other known problems of a general nature in AutoCAD Release 10.
  - \* DONUTs drawn with a zero inside radius will plot incorrectly when using the "Adjust area fill boundaries for pen width" option at plot time. You can work around this problem by using a small undetectable inside donut radius (e.g., 0.0001).
  - \* The TEXT and DTEXT commands allow the entry of a text item when using the FIT option and the specified start and end points for the text are the same. This means that the text is trying to fit in a 0.0 length space. This results in the text making a vertical line the height of your text.
  - \* The HATCH command fails to recognize hatch patterns defined in an external ".PAT" file other than ACAD.PAT when using any of the I, O, or N (Ignore, Outermost, or Normal) pattern style modifiers. AutoCAD will return an "Unknown pattern name. \*Invalid\*" error message. You can add the pattern definition text to the ACAD.PAT file to avoid this problem.
  - \* DXFOUT does not include group code 30 information in the header variables \$EXTMAX and \$EXTMIN as is documented in the AutoCAD Reference Manual. This does not cause a problem for DXFIN since Release 9 DXFOUT did not produce a group code 30 for these variables and Release 10 can read a file with or without them.
  - \* When INSERTing a BLOCK using the XYZ scale option, the prompt for the Z scale factor is misleading if you enter a negative value for the X scale factor. It prompts <default=X> indicating that if a <cr> is entered, the scale factor for Z will be the negative value entered for the X scale factor. Instead, the Z scale factor is entered as the absolute value of the X scale factor making it positive.
  - \* Using PEDIT to Spline Fit a curve on a polyline will result in an unselectable entity if you have specified an invalid "0" value for the SPLINESEGS drawing variable. It does not make sense to fit a spline of 0 segments and this should not be attempted.

- \* The following list identifies other known problems specifically affecting the XENIX platform of AutoCAD Release 10.
  - \* Configuring for a XENIX System Mouse two times in succession for the same tty port without leaving AutoCAD will result in a "Device Busy" error message. To avoid this, exit AutoCAD completely before reconfiguring for that device.
  - \* In the "C" shell (csh), entering a "\" backslash character for the name of a drawing will result in the system appearing to hang. This is an incorrect name for a drawing file. The system is waiting for a control character to follow the backslash. Avoid using invalid names for drawing files.
  - \* Incorrectly configuring your hardware devices can result in difficult to diagnose system errors. Carefully configure AutoCAD for the exact peripherals you have installed. For example, configuring a XENIX System Mouse in AutoCAD as a Microsoft Mouse while having a Mouse Systems Mouse connected to the computer can result in the computer re-booting when the mouse is moved.
  - \* SCO XENIX 386 allows you to boot the machine as a DOS machine if you have an MS-DOS partition allocated on your hard disk. This can sometimes cause problems when entering the XENIX partition. For example, booting the computer as a DOS system with DOS system drivers loading from CONFIG.SYS and then entering XENIX from DOS may result in some SYSTEM devices failing to respond. To avoid this problem, enter XENIX first at boot time.
  - \* The external commands feature in AutoCAD using the ACAD.PGP file has some unique considerations and requirements. Using return codes 1 and 2 in the ACAD.PGP file allow the user to read in a DXB file or make a block out of a DXB file having a name of \$cmd.dxb (respectively). Under XENIX (and other UNIX systems), a file name preceded by a "\$" character results in that file name being passed to the system for metacharacter expansion. Using the external command feature above results in the error message "Error during metacharacter expansion". Also, entering a shell using the SHELL command immediately after a PLOT or a PRPLOT will fail, immediately returning to the AutoCAD editor.
- Since XENIX is a multi-tasking environment, the AutoCAD external command feature is not very useful because you can enter another shell to accomplish another process or task. You should rely on XENIX for external operations using return codes which conflict with metacharacter file names.
- \* Cancelling a PLOT or PRPLOT can result in an orphaned XENIX system process. The result can be slow or jerky digitizer tracking and/or response. The orphaned process must then be killed manually to relieve the system of the performance burden. If you experience this behavior, check for orphaned processes and remove them while logged in to the system as a Substitute User with root privileges.
  - \* Creating large 3DMESH entities can result in swap space limits problems. If you encounter an "INTERNAL ERROR: SCANDR" from AutoCAD, or an Out of Swap Space error from the system while working with large

polygon mesh entities, increase your swap space at the XENIX System level.

- \* SCO XENIX 386 has no provision for interfacing with pointing devices connected with the Compaq Mouse Port on Compaq 386 computers or the IBM Mouse Ports on IBM PS/2 computers. Specific XENIX System Mouse devices listed in your XENIX 386 documentation must be used.

#### SCO UNIX 3.2.x

- \* A new version of Santa Cruz Operations Operating System has been released as a true version of UNIX with a full AT&T license. This operating system was minimally tested with AutoCAD Release 10. Preliminary tests show that AutoCAD will run under this environment. However, extensive testing was executed using the XENIX 386 release from SCO. Problems running AutoCAD under SCO UNIX should be directed to SCO product support.
- \* SCO UNIX kernel release 3.2.0 has a problem running with AutoCAD. Communication IO privilege errors can occur. If you have this problem, you should contact SCO immediately for a patch or update to SCO UNIX kernel release 3.2.1. This version does not have the problem and the patch will provide a temporary work-around.

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