

DesignCAD 23.0 Release Notes

July 2013

Thank you for purchasing or upgrading to DesignCAD Version 23.0. This release of DesignCAD offers the following new features, improvements and fixes:

New Features:

DWG/DXF version update – DesignCAD 23 adds support for AutoCAD 2013 and 2014 DWG and DXF files. AutoCAD 2013 is now an option in the File/Export/Setup features for exporting DWG/DXF files. Also, you can use Type 7 to export AutoCAD 2013 from BasicCAD using DwgOut or DxfOut. (Type 7=2013, 6=2010, 5=2007, 4 = 2004, 3 = 2000, 2 = R14, 1 = R13, 0=R12). Export to AutoCAD 2013 is also now an option for Ole Automation.

Centered mouse-wheel zoom – Now when you use Ctrl+Mouse-wheel to zoom, the drawing shifts to center on the cursor location where you started the zoom, and subsequently zooms from the current cursor location, making it easier to zoom in quickly on a particular location. In all previous versions, Ctrl+Mouse-wheel always zoomed from the drawing center.

New options

Options/Layer Options/Right-click ->Set Highlighted Text Color – Lets you change the color of text on highlighted layers in the layer list. Note that this just changes the color of highlighted text in the dialog box – this does NOT change text colors in the drawing itself.

Command Enhancements

Stretch – Since time immemorial (well, at least 1995), the Stretch command paid no attention to whether or not the option "Enable Multi-layer Editing" was checked, but instead would always affect all editable layers. Now it restricts itself to the current layer if Enable Multi-layer Editing is not checked.

Bug Fixes:

Open – Fixed a problem which prevented the user from loading drawings from much-older versions of DesignCAD that contained no-longer-valid linetype values.

Rotate – When Drag Mode was checked, rotating around the Z axis was rotating the wrong way.

Wide Lines – If a wide line had two or more identical vertices, the line would not display completely at some zoom factors.

Wide Lines – When you close a wide-line shape, the closed corner should now miter correctly as you zoom in further.

Polygon Add – Occasionally using Polygon add two or more times in succession would crash DesignCad, depending on the order in which the polys were added.

Section Cutoff, Section Trim – Improved the accuracy of the Section Cutoff and Section Trim commands when cutting arcs and circles.

Block Define – When objects used to define a block contained hidden segments, all segments became visible in the newly-defined block. Now hidden segments remain hidden in the block.

Batch Convert – Previously, this command was only allowing the user to add a few files at a time to the conversion list.

LockX, LockY– These commands, if triggered by a shortcut key or the command line, would sometimes lock to an unexpected location.

Stretch – The Stretch command would sometimes stretch text that was not included in the original stretch region.

Weld – When welding multiple planes which did not all overlap to form a single shape, the resulted shapes were converted to vectors instead of planes.

DWG/DXF Export – Blocks whose insertion points were not at z=0 would not translate to the correct z-value in the exported drawing.

DWG/DXF Export – If the drawing was rotated before export, some blocks were not exported in the correct orientation.

DWG/DXF Import – Imported line widths were sometimes not the right value.

Layer Options – The Layer filter setting was not being correctly saved and restored after Save as Default.

Groups and Solids – Fixed a bug when saving and reading back very high solid ids that were also in a group.

User Notes – Sometimes user notes in the Drawing Info screen were being read back in with garbage data appended.

BasicCAD:

New Sys() functions:

Sys(171) – Print Black and White [0 = no, 1 = yes]

Sys(172) – Print Shadows [0 = no, 1 = yes]

Sys(490) – Read-only – acquires the index of an entity whose longid (64-bit id) was written to Sys\$(490)

Sys(1143) – Read-only – Last-used Solid ID

Sys(1144) – Read-only – Lowest unused Solid ID

Sys(1173) – Display CS [0 = no, 1 = yes]

Sys(1174) – Automatically pack Group IDs and Solid IDs on Save. [0 = no, 1 = yes]. This setting remains sticky between sessions.

IMPORTANT NOTE: Sys(1174), when set to 1, is currently conflicting with some save operations within BasicCAD. We suggest for now that if your macro is going to save a drawing, your macro code should set Sys(1174) to zero and use the >PackGroupIDs and >PackSolidIDs commands (if so desired) before saving a file.

New Sys\$() functions:

Sys\$(295) – Read-only – Retrieves the long id (permanent 64-bit id used by OLE Automation) of an entity selected by the Entity Statement as a string. An entity's index may change if other drawing entities are deleted, but so long as it remains unedited (no trimming or rearranging points) its 64-bit id should remain constant. This may prove handy for keeping long-term track of entities over the course of many drawing sessions.

Sys\$(240) – Write-only – Store the string representation of an entity's 64-bit id here, then read Sys(490) to get that entity's index. If the 64-bit ID is not found, Sys(490) will be set to zero.

>DwgOut and >DxfOut – added AutoCAD 2013 support to the <Type parameter.

<Type [7=2013, 6=2010, 5=2007, 4 = 2004, 3 = 2000, 2 = R14, 1 = R13, 0=R12]

New Query/Change Parameter:

Alias ENT_LONGID &H0x011B 'readonly string representing the long id of an entity selected by the Entity statement

Ole Automation:

The **PrintOption** class now has the **PrintBlackAndWhite** property added.

The **DcAcadVersions** enumerator has new value **dcAcadVer2013** for AutoCAD 2013 support.

The **EntArrow** class has new properties for **Curved** and **DoubleHeaded**.

The **Entity** class offers new **GetHiddenPoint**, **SetHiddenPoint**, **GetHiddenPoints**, and **SetHiddenPoints** methods to locate and/or define points that mark the beginning of a hidden segment. These work much like **GetPoint**, **GetPoints**, **SetPoint**, and **SetPoints**, except they use integers instead of floating point values.