

## DesignCAD 3D Max 23.1 Release Notes

March 2014

DesignCAD 3D Max 23.1 includes the following improvements and fixes:

### General fixes

Text-only dimensions from DesignCAD versions 17 and earlier were being interpreted as full dimensions, showing both arrows and extension lines.

Diameter dimensions from versions 19 and 20 were being displayed as Radius dimensions (i.e. the value shown was half the intended value).

The Password Options button in the new Layer Options dialog box was not asking for a password before allowing the user to alter locked layers in an already-protected drawing.

Some attributes were missing when certain .DWG files were imported into DesignCAD.

Opening .DC2 drawings that had the grid enabled were crashing DesignCAD 23 when opened.

Drag-select was not selecting some grouped items.

Save Image was not respecting the improved shadow map algorithm.

The Print command was not working properly when the Black and White option was combined with Hidden Line removal.

After de-selecting multiple items that included an image, the non-image entities sometimes still displayed in the selection color.

### RedSDK Mode fixes:

Lines with the CONTINUOUS linestyle (imported from .DWG files) were displaying as hatch fills when in RedSDK mode.

Loading an image into any view other than Front in RedSDK mode caused the image to display distorted, or not at all.

Rotating an image in RedSDK mode caused image distortion.

Deselecting by pressing Esc was only updating currently-active view in RedSDK mode. In other views the previously-selected items still appeared in the highlight color.

# DesignCAD 3D Max 23.0 Release Notes

July 2013

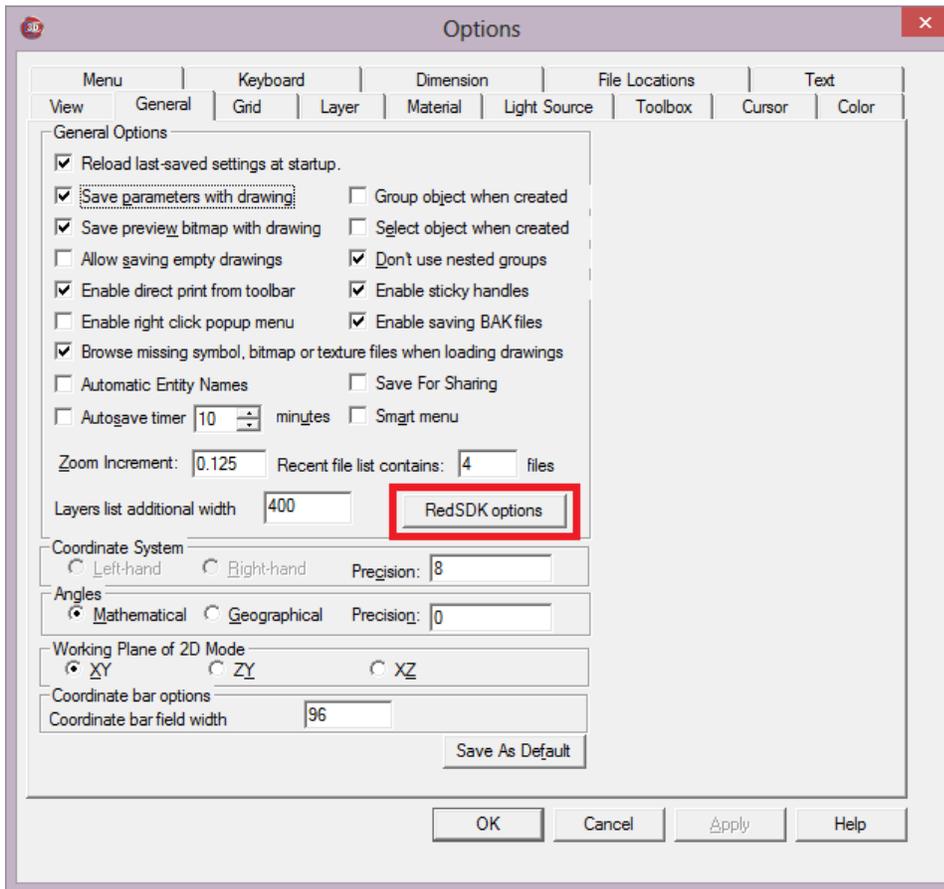
Thank you for purchasing or upgrading to DesignCAD 3D Max Version 23.0. This release of DesignCAD offers the following new features, improvements and fixes:

## New Features:

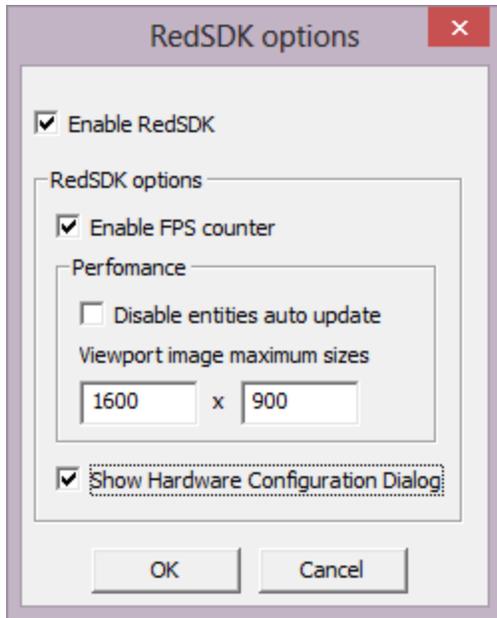
### **RedSDK Mode –**

DesignCAD 3D Max 23 is the first version of DesignCAD to incorporate RedSDK technology to leverage the power of modern graphics cards. With RedSDK Mode enabled, DesignCAD will be able to pass off some of the display-related tasks to your graphics card, potentially speeding up many display operations. RedSDK mode is currently aimed primarily at 2D operations; 3D operations are not yet fully supported in this release. More features and improvements will be added in the future.

RedSDK mode can be enabled or disabled via the "RedSDK Options" button in the General Options tab:



This opens the RedSDK Options dialog box:



**Enable RedSDK** – This checkbox turns RedSDK Mode off or on. Once you click OK after changing this setting, DesignCAD will have to restart to enter (or exit) RedSDK Mode.

**Enable FPS counter** – This checkbox will turn on or off the RedSDK performance data that can be shown in the upper right corner of each view. Most users will probably prefer to leave this off.

**Disable entities auto update** – RedSDK works with geometrical representation of entities. It means that before it can draw them, DesignCAD must vectorize them, and this procedure takes time. Also, there are many entities whose displayed geometry depends on the zoom factor (e.g. curves and circles). This way, DesignCAD should re-vectorize these entities each time when zoom factor is changed.

If this option is not checked, re-vectorization happens each time the zoom factor is. It slows down zooming commands.

If this option is enabled, zooming becomes faster (at least it should). However, on large zoom you can see defects (e.g. closed polygonal line instead of circle). But in this case you can use manual regeneration (CTRL+R or View->Regenerate command) and these defects should disappear.

**Viewport image maximum sizes** – RedSDK presents some entities as images (e.g. View Frames and Gradient Hatch patterns). However, images are strongly zoom factor dependent. This option limits maximum generated image sizes to prevent excessive memory consumption. The smaller the values here, the more restricted the maximum image size.

**Show Hardware Configuration Dialog** – If this option is checked, each time you start DesignCAD in RedSDK mode, DesignCAD will warn you if your graphics card is not supported, or if you are using an unsupported driver.

**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/Options/View -> Use accelerated pixel format** – This is another experiment to try and get more out of modern graphics cards when not using RedSDK. Results are likely to vary dramatically between different versions of Windows and different graphics cards – some may experience improved performance, others may see degradation in performance or no discernible change. Try it, and if it doesn't help, turn it back off.

**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

**Solid Add, Solid Subtract, Solid Intersect, Solid Segregate, Solid Exclusive Or** – These commands now all have an option to only work on selected items. This should help you with those pesky drawings that have lines, arcs, or other solids that share a face or vertex with what you're trying to Boolean. Pre-select those things you want to Boolean, run the command, check the "Selected only" box, and click on the objects you want to modify -- non-selected items should now be ignored and only the pre-selected items can be picked for the command.

**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.

**Display Grid** – Fixed a problem which prevented the display grid from appearing in all views if it was set to XZ or ZY plane.

**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, LockZ** – 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.

**Print** – When printing a perspective view with extreme perspective distortion (a very short view distance), saving the drawing afterward might corrupt the drawing geometry.

**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.

**Solid Add, Solid Subtract** – Sometimes these commands would place the results in unexpected layers or change the materials to Aluminum.

**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.

**OLE Automation** – Solid Intersect was not returning an entity id.

**Slice** – The Slice command would not always work on "improper" solids. For example, if you defined two 3D boxes that were separated by a gap between them as a single solid, attempting to slice away one of the boxes by slicing through the gap would have no effect. Now you can do that.

## **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 **Trim** class adds new **CrossSection(SelectedOnly, SliceType)** method.

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.