# What's New in DataCAD 14?

KnockOut

KnockOut can be enabled on various types of entities to mask out information behind the object. When KnockOut is enabled, other drawing information around the entity is masked by a solid fill which has the same color as the drawing area background, making it appear transparent. This is particularly useful, for example, when adding text or dimensions which you want to display over hatching.
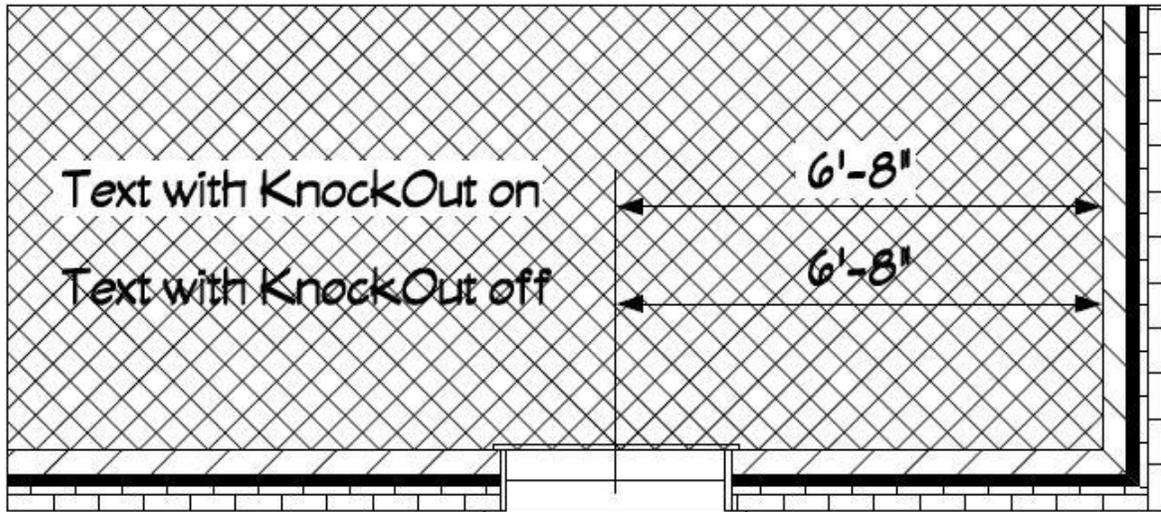
The KnockOut feature is supported by Text, including MText and PText, Associative Dimensions, 2D Arcs, 2D Circles, Ellipses, Polylines, Polygons and Slabs. Additionally, a sub-option can display or omit the KnockOut boundary, similar to the way in which hatching can be set to show the outline or not. The status of the KnockOut toggle in the Text menu and in the Dimension/Text Style menu is remembered in saved Text Styles and Dimension Styles respectively.

➔ To draw text with KnockOut enabled:

1. Click the [F8]KnockOut on in the Text menu.

2. Toggle "Yes" on to include KnockOut on new text.

   a. Toggle "Border" on if you wish to display the KnockOut border.

3. All new text will have KnockOut enabled.

➔ To enable KnockOut on an object already in the drawing:

1. [Ctrl]+[Right-click] on the entity and mouse over "KnockOut."

2. Select "Yes."

After enabling KnockOut on an object, you may [Ctrl]+[Right-click] on it again, and set "Border" to "Yes" to display the KnockOut border

You may also change the status of KnockOut on entities using the standard selection menu. Select "More" from the "Change" menu and then pick "KnockOut." Toggle "Yes" on to enable the option, or toggle "No" on to disable the option. When "Yes" is on, the "Border" sub-option may also be selected. Right-click twice to return to the "Change" menu, and then select the objects you wish to change.

The KnockOut feature will only mask entities that were drawn before it. These entities are described as being "behind" the KnockOut entity, both in terms of the draw order on the screen, and the position of the entities relative to the KnockOut entity in the drawing file database.

If you add an entity with KnockOut enabled, and then draw on top of it, the new information will display on top of the KnockOut. If you would like the KnockOut to mask information drawn after it, you can use the Move/To Front command to move the KnockOut entity on top of the other information. You could also use Move/In Front, and select the entities you wish to move the KnockOut entity in front of. Move/To Back and Move/In Back are also useful for defining the order in which the entities display.

Enabling KnockOut on an entity via the Change menu, the [Ctrl]+[right-click] menu or the Propertied Editor will also perform a "Move to Front" on the selected entity automatically. This makes the KnockOut entity the last entity on the layer, thus displaying it on top of other objects.

Related Settings

The [k/K] button on the SWOTHLUDFBK toolbar controls whether or not KnockOuts are displayed. When on, [K], KnockOuts are shown. When off, [k], no KnockOuts are displayed in the drawing. You'll find the same setting, labeled "KnockOut Display," in the "View" pull-down menu under the "KnockOuts…" sub-menu.

The "View" pull-down also has a "KnockOuts…" sub-menu with settings related to the display of KnockOuts. The option "KnockOut Display" controls the display and printing of all KnockOut entities. When toggled off, entities with a KnockOut property will display and print as normal entities without the KnockOut. The "Show all KnockOut Borders" setting forces DataCAD to display all KnockOut borders regardless of their individual border settings. This is a display only setting, and does not affect printing. The Compatibility Mode" option draws and displays KnockOuts in their database order (the order in which they were created), using the same methodology as AutoCAD. You can turn this option off to always display and print all KnockOuts last, or on top of other entities.

The "Show All KnockOut Borders" and "Compatibility Mode" options are only available when "KnockOut Display" is enabled.

AutoCAD compatibility

AutoCAD has a similar feature named Wipeout. When you export a file that contains KnockOuts, they are converted to AutoCAD Wipeout entities. The result is that the file looks the same in AutoCAD. When you import a file that contains Wipeout entities, DataCAD converts them to polygons with KnockOut enabled. The result is that the file looks the same in DataCAD.

DataCAD has the ability to display KnockOut borders on a per-entity basis, but AutoCAD does not. When you export a file to AutoCAD format, you must decide whether to display the KnockOut borders for all KnockOut entities or for none. After you select File/Export/DWG and pick either All Layers or On Layers, the "Enter filename to export" dialog appears. If you wish to display KnockOut borders for all KnockOut entities, place a check next to "Wipeout Frames On." If you do not wish to display the KnockOut borders, leave this option un-checked.

Similarly, when you import a file from AutoCAD, you must decide whether or not to display the KnockOut borders. After you select the file to import, the Import Entity Assignments dialog is displayed. If you wish to display the borders for all KnockOut entities, check the "KnockOut Borders On" box. Leave this un-checked if you do not want all KnockOut borders shown. Keep in mind that you can change them individually after you import the file.

Extended character codes

The following extended character codes can be used to change the status of the KnockOut option and the KnockOut Border sub-option. This toggle is found in the Text, MText, PText, Text Attributes and Dimension/Text Style menus, as well as in the 2D Curves menu and the 3D Polygons and Slabs menus:

  1056: KnockOut On

  1057: KnockOut Off

  1058: KnockOut Border On

  1059: KnockOut Border Off

The following extended character codes can be used to control the status of KnockOut display in the View/KnockOuts menu:

  1060: Display of KnockOuts On

  1061: Display of KnockOuts Off

Wall Type Manager

The Wall Type Manager has been updated to provide the ability to add hatching and/or solid fill to the wall cavities. As part of this enhancement, we have changed the layout of the Wall Type Manager. There are tabs named "Line Settings," which contains settings related to the wall lines in 2D and in 3D, "2D Cavity Settings," which has settings related to hatching and fill for 2D display of cavities, and "Cap Settings," which is where you define the way end caps, top and bottom caps, and opening caps render in the 3D Object Viewer.

Line Settings tab

The settings on this tab are the same as the previous version, except that the "Caps and Openings" section has been removed. On the Line Settings tab, you will define the basic properties of the lines that make up your wall type. Set the number of lines and select which is used as the Control Line, choose the distance that each line is offset from the previous line to define the width of the wall, and set the 2D line colors and 3D rendering materials for the individual wall lines.

2D Cavity Settings tab

It is on this tab that you may apply hatching and/or solid fill to the wall cavities. In the case of a 2-line wall, the hatching or fill covers the entire wall cavity. In the case of a 3-line wall, you may choose to hatch or fill the inside cavity, the outside cavity, or both. 4-Line walls support hatch and fill on the inside, center and outside cavities independently. These properties are type-dependent, so any changes you make affect all instances of the wall type throughout your drawing file.

➔ To apply hatching or solid fill to a wall type:

1. Open the Wall Type Manager, load a wall type and set it active.

2. Go to the "2D Cavity Settings" tab.

3. Click the "Cavity" drop-down to select which cavity to modify. For 2-line walls, "Inside" is the only option; for 3-line walls, you may select the "Inside" or "Outside" cavity; for 4-line walls, choose "Inside," "Center" or "Outside."

4. Check "Do Hatch" to apply hatching to the selected cavity.

   a. Click the "Pattern" drop-down to select which hatch pattern will be applied.

   b. Enter a value in the "Scale" field to define the scale of the hatching.

   c. When "Match Wall Angle" is unchecked, you define an absolute angle for the pattern in the "Angle" field. When it is checked, the angle value is relative to the angle of the wall. For example, if the angle is zero, the pattern matches the wall angle. If the angle is 45°, the pattern is displayed at a 45° angle relative to the angle of the wall segment.

   d. When "Origin at Zero" is checked, the hatch pattern origin is situated at absolute zero. When unchecked, the origin coincides with node 1 of the wall segment.

   e. You can match the color of the hatch to the wall line, or you can uncheck that option and click the color rectangle to open the Color Palette and select a different color.

5. Check "Do Fill" to apply a solid or pattern fill to the selected cavity.

   a. Under "Settings," you can either check the box to make the fill the same color as the line that borders the cavity, or you can uncheck the box and click the color rectangle to open the Color Palette and select a specific color.

   b. Click the "Pattern" drop-down if you wish to define a pattern fill.

   c. Check the box next to "Pattern Color" to make the pattern the same color as the line which borders the cavity, or uncheck it to define a specific color.

Wall cavities support both hatching and solid fill at the same time. As long as the hatch and fill are different colors, you will see the hatch drawn on top of the solid fill.

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6. Click OK when you have finished adjusting the settings.

The "Architect" menu contains toggles labeled "Hatch" and "Fill" that enable or disable the display of Smart wall hatching and Smart wall fill independent of other hatch and fill in the drawing. These settings affect both display and printing. The "Hatch Patterns" and "Fills" buttons in the "Display" menu and the corresponding [H] and [F] buttons on the SWOTHLUDFBK toolbar will suppress all hatching and fill, including those that are within Smart walls.

The following extended character codes are related to the display of Smart wall hatching and solid fill:

1062 – Turn display of Smart wall hatch on

1063 – Turn display of Smart wall hatch off

1064 – Turn display of Smart wall fill on

1065 – Turn display of Smart wall fill off

Cap Settings tab

Use the settings on this tab to define the rendering characteristics of the Top/Bottom Covers, the End Caps, and the Openings for Smart walls in the 3D Object Viewer. While the three categories of caps are defined independently, they all have the same settings available. Top/Bottom Caps refer to the horizontal plane on the top and bottom of each wall segment; End Caps refer to the vertical plane at the end of each wall segment; Openings refer to the 4 edges of a window or door void, which are only displayed if the layer that contains the window or door is turned off.

There are three mutually exclusive settings for the various caps, which you select by clicking the appropriate radio button. When "Inherit From Inside Line" is selected, the material assigned to the Inside line covers the cap from the inside line to the outside line. When "Inherit From Outside Line" is selected, the opposite is true; the rendering material assigned to the outside line covers the cap from the outside line to the inside line. Selecting "Assigned" gives you the ability to define the material on each wall cavity independently. Choose "Inside," "Outside" or "Center" from the "Cavity" drop-down, and then use the Material drop-down to set the material for that cavity. Refer to the preview window, which updates to reflect your changes in real-time, to see what your wall will look like.

Cleanup Routines

When you draw Smart walls with hatching or solid fill, DataCAD will continue to clean the intersections automatically. In order to display the hatching correctly, you will need to use the Cleanup menu commands to manually clean the intersections. There are new steps involved in the X-Intersect and L-Intersect routines, and a new result for T-Intersect cleanup that creates a more accurate representation of the intersection.

➔ To perform an X–Intersect cleanup:

1. Decide which two walls you would like to clean.

2. Choose "X Intersect" from the "Cleanup" menu.

3. Place an Area box around the intersection of the two walls. DataCAD prompts you to *Select Master wall."*

4. Click the Master wall. This wall will remain one continuous wall, while the other wall will be broken and cleaned to the inside and outside face of the Master wall.

If you are performing an L-Intersect cleanup and the two wall segments are the same wall type, the command works the same as always; the two walls are connected at the control line, and the intersection of the interior wall lines is mitered. If the two wall segments are of different wall types, and the angle between then is 90 degrees, you will need to define the master wall; the master wall gets its end capped, and the other wall will butt up against it. If the angle is not 90 degrees, the walls connect at the control line and have their interior lines mitered.

The manual T-Intersect cleanup produces a similar result. While the automatic cleanup continues to function as it always has, with the two walls meeting at the control line and the interior wall lines being mitered, the manual cleanup routine butts the perpendicular wall segment where it meets the primary wall.

Lock Size on associative dimensions

The "Lock Size" property that maintains text size regardless of print scale can also be applied to associative dimensions. DataCAD will automatically adjust the size and offset of dimension text, the offset, overlap, increment and overrun of dimension lines, and the size of the arrows, tick marks or dots.

➔ To enable Lock Size on associative dimensions before you draw them:

1. Select "Text Style" from the Dimension/Linear menu.

2. Toggle "Text Scale" on.

3. Turn "Lock Size" on.

4. You may press the [PgUp] and [PgDn] keys on your keyboard to set the current plot scale.

➔ To enable Lock Size on associative dimensions that you have already drawn:

1. [Ctrl]+[right-click] on the associative dimension.

2. Mouse over "Lock Size" in the context menu.

3. Select "Yes."

You may alternatively use the Change/More/Lock Size menu option if you wish to select multiple associative dimensions by Group or Area.

Lock Size on symbols

You can apply the "Lock Size" property to symbols that you insert into your drawing. This is particularly useful with notation symbols, which are typically printed at a fixed size regardless of the scale of the detail. It works in combination with the Symbol Scale feature, which is similar in concept to Text Scale for text and dimensions. When Symbol Scale is enabled, DataCAD enlarges the symbol by a factor that is determined by the current plot scale. The Lock Size property changes that factor each time you change the plot scale, resulting in symbols that print at the same size no matter what the print scale is set to.

When DataCAD changes the enlargement factor of the symbols in the drawing, the center of enlargement is defined by the symbol's insertion point. Keep this in mind when creating the library of symbols that you intend to use with this feature.

➔ To enable Lock Size on a symbol as you place it in your drawing:

1. Select the symbol that you wish to insert into the drawing by clicking on it in the Symbol Browser. DataCAD displays the *Ins Symbol* menu.

2. Toggle "Symbol Scale" on.

3. Toggle "Lock Size" on.

4. You may press the [PgUp] and [PgDn] keys on your keyboard to set the current plot scale.

➔ To enable Lock Size on a symbol you have already inserted into the drawing:

1. [Ctrl]+[right-click] on the symbol.

2. Mouse over "Lock Size."

3. Select "Yes."

You may alternatively use the Change/More/Lock Size menu option if you wish to select multiple symbols by Group or Area.

If you are creating your own library of symbols and you intend to use Lock Size in conjunction with Symbol Scale, you should not use Lock Size on any text that will be inside the symbols. Since Lock Size is being applied to the symbols directly, it is not necessary to apply it to text inside them.

Lock Size on hatching

To keep hatch scale at a fixed size regardless of print scale, turn the "Lock Size" property on. You might find this useful when drawing a ceiling grid using hatch. If you change the plot scale, DataCAD will automatically enlarge (or reduce) the scale of the hatch so that size of the pattern remains the same when you print your drawing.

➔ To enable Lock Size on hatching before you add it to the drawing:

1. Select "Hatch Type" from the "Hatching" menu.

2. Toggle "Lock Size" on.

➔ To enable Lock Size on hatching that is already in your drawing:

1. [Ctrl]+[right-click] on the hatching and choose "Polyline Properties."

2. Click the drop-down menu labeled "Lock Size" and select "Yes."

3. Click OK to dismiss the Properties editor.

3D Object enhancements

The maximum number of vertices on a 3D polygon or slab has been increased from 36 to 256. The maximum number of primary and secondary divisions for cylinders, cones, spheres, tori and revolved surfaces has been increased from 36 to 128. If you use the "Save As" command to save a DataCAD 14 file back to DataCAD 13 or earlier format, polygons and slabs with more than 36 vertices are exploded to a collection of three-sided polygons. Quadrics with greater than 36 primary or secondary divisions are exported with 36 divisions.

Clipping Boundaries

XREF Clipping boundaries (XClips) and Symbol Clipping boundaries (SClips) can be edited without having to redraw the entire boundary. [Ctrl]+[right-click] on the XREF or Symbol and choose "XClip" or "SClip" from the XREF Tools or Symbol Tools context menu. Click the "Edit" menu button to edit the boundary. You will use the standard polyline editing tools to edit the boundary.

Miscellaneous enhancements

GoTo Views and XREFs

The feature which enables GoTo Views in your drawing to remember the on/off status of layers in XREFs has been enhanced to provide greater control. If you define a highlight color and/or line type for individual layers in the XREF, those settings are now maintained when you use a GoTo View in the drawing to turn XREF layers on and off. In addition, the GoTo View will save and restore XClip boundaries.

Pen Sort

The Pen Sort toggle in the Print/Plot menu determines whether or not the draw order for information on the printed page is defined by the Pen Table or by layer order. When this option is on (default), DataCAD will continue to print objects based on their Pen Assignment order, as in previous versions. Colors assigned to Pen 1 are printed first, and then colors assigned to Pen 2 are printed next, continuing through the list of Pen Assignments. If you toggle Pen Sort off, then objects are drawn based on layer order and the order in which the objects were added to the drawing. For example, objects drawn on Layer001 are drawn first, and then objects on Layer002 are drawn next. If two objects on the same layer intersect, the one that was drawn first is on the bottom, and the one that was drawn second is on top. You can use the Move command to move objects in front of or behind each other.

SketchUp Export

DataCAD has the ability to export files in both SketchUp 7 and SketchUp 8 format. This can be helpful if you are working with someone that uses an earlier version of SketchUp. Select "SketchUp" from the File/Export menu. Click the "Save As Type" drop-down on the "Enter filename to export" dialog, and select the format in which to save. Supply a name and click "Save" to complete the process.

Multi-View Windows

There is a "Zoom %" option in the Multi-View Window pop-up menu. Right click in a Multi-View Window and select it to set the zoom percentage for all Multi-View Windows. The default is 100%. This feature may be helpful if you are unable to see the entire image in the Multi-View Windows, and need to reduce the image size.

New dcadwin.ini keys

[General]

LayerColorRemembersStatus=TRUE
  When FALSE, the Layer Color toggle in the Move/To Layer command will always default to off.

 [Hardlock]

Check For Prior Hardlocks=FALSE
 When TRUE, DataCAD will search for, and use, a previous version hardlock for license
 authentication.
Look For Network Hardlock=FALSE
 When TRUE, DataCAD will search the local network for a hardlock if one is not found on the local
 parallel or USB port. When FALSE, DataCAD only searches the local parallel and USB ports.