KUBOTEK®

Version 8.0
Release Notes

- **KeyCreator®**
- **Kubotek Spectrum™**

Table of Contents

ABOUT RELEASE NOTES AND OTHER DOCUMENTATION

Use release notes to find out about new functions or enhancements to existing functions for this release.

For more details about all functions, use the product's Help documentation (**Help>Help Topics**, or press **ALT+F1**).

For documentation that focuses on task-oriented needs, use the tutorials. These are available in the product's Help system (**Help>Tutorials**).

The web site has other product and company information (**www.kubotekusa.com**).

NOTE ON HELP SYSTEM: Starting with this release, KeyCreator Help is an HTML Help system rather than a WinHelp Help system. For details, see *HTML Help with This Release*, on page 36.

NEW FUNCTIONS

This release adds the new functions explained below.

OBJ Export

Location: File>Export>OBJ. With this release, you can export the model or selected entities as a `.obj` file.

Name Function Moved from Composite to Entities Menu

Location: Edit>Entities>Name. This release moves the Name function from the **Tools>Composite** menu to the **Edit>Entities** menu. The topic name in KeyCreator Help for this function has been changed from *Composite Name* to *Name*.

Modeless Change Attributes

Location: Edit>Entities>Attributes by Single Select. This function displays a modeless version of the **Edit>Entities>Attributes by Selection** dialog. This means that you do not have to select **OK** to close the dialog before selecting the entities to change. Rather, select the desired attributes in the dialog, and then click an entity in the viewport. The entity immediately changes to the selected attributes. The dialog stays open for the duration of the function so you can easily select different attributes for the next entity.

Phong Shaded Rendering

Location: View>Render>Phong Shaded. This function allows you to display a solid model with a shaded representation. It is more accurate and has fewer artifacts than the **View>Render>Gouraud Shaded** function. See the related *Smooth Shaded Now Named Gouraud Shaded* on page 14.

Shadow Map

Location: View>Render>Shadow Map. Causes shadows to be drawn on faces in the model. This function is available only when using the Direct3D driver.

Reflection Plane

Location: View>Render>Reflection Plane. Causes a reflection of the model to be drawn below the model.

Smart Trim Functions

Location: Modify>Trim>SmartSingle and **Modify>Trim>SmartBoth**. This release provides two "smart" trim functions. To understand their use, it is helpful to compare them with the **Modify>Trim>Both** function (which was available before this release and still is available). With the latter function, trim is accomplished by selecting explicitly the two desired entities involved in the trim. But with the smart functions, it is only necessary to select one entity. KeyCreator then automatically selects the second, intersecting, entity nearest to where you clicked on the first entity. So the location where you click on the first entity is crucial in determining which second entity KeyCreator selects automatically for the trim operation.

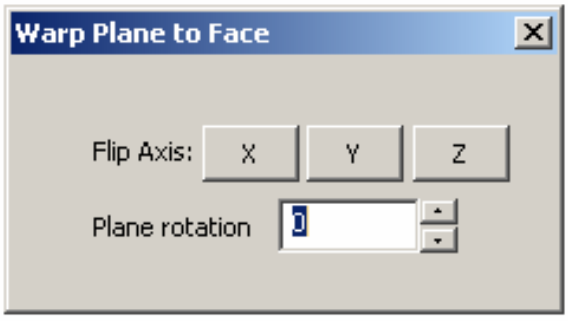
Here is the difference between SmartSingle and SmartBoth. SmartSingle allows you to select an entity, and the function automatically trims it to its intersection with the closest entity. Note this special case: When a closed entity is selected, the function looks for a second intersection with another (third) entity, and trims the selected entity against two intersecting positions (like Trim>Double). The function fails if any of the two intersections is not found.

SmartBoth does what SmartSingle does (trim the selected entity to the intersection of its closest neighbor) and additionally it trims the closest entity to that same intersection position as well. Note this special case: If both the selected entity and its closest neighbor are closed entities, the function defaults to a trim single of the selected (closed) entity, looking for and trimming against two distinct points of intersection. It ignores trimming the closest entity.

Plane to Surface Function Added to Warp Menu

Location: Modify>Warp>Plane to Surface
This function allows you to warp a set of bodies from a planar surface to a non-planar one. For example, you could use this function as you design a tire tread on a flat surface and then warp it to a revolved surface. Or, you are designing a shoe tread flat and then use this function to warp to the sole of a shoe. Still another example is to warp lettering around an object.

The **Warp Plane to Face** dialog, shown below, is available in this function. If you find that the orientation of the bodies is not what you had expected, you can use the dialog to flip the warped bodies about their X-, Y- and Z-axes. Also, you can rotate them about the Z-axis by entering angles in the **Plane rotation** edit field.

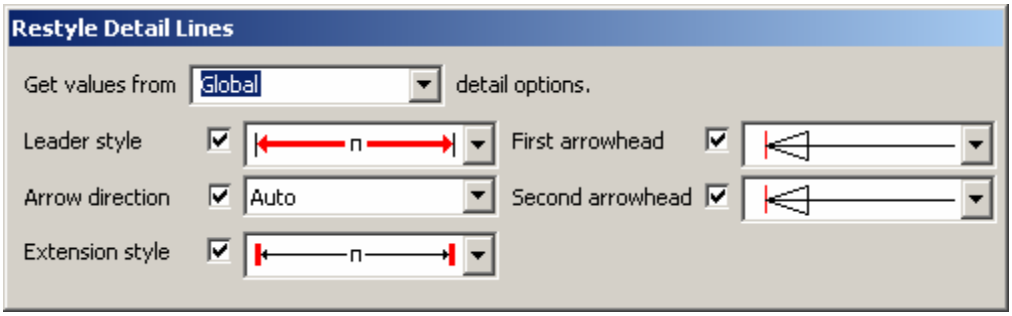


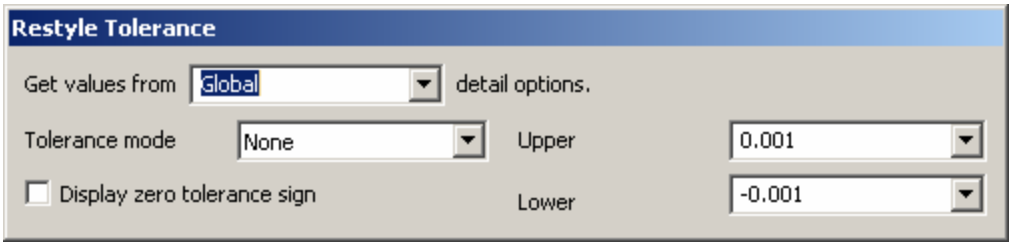
Hole Table to CSV

Location:Detail>Notes>Hole Table to CSV
This new function exports the information in a hole table to a .csv (Comma Separated Value) file. This capability also is made available through the Generic Edit for a hole table when you click the **CSV Export** button on the **Hole table formatting** pane of the **Hole Table** dialog. Note that the button appears on this pane only when you are editing a hole table.

Restyle Detail Entities

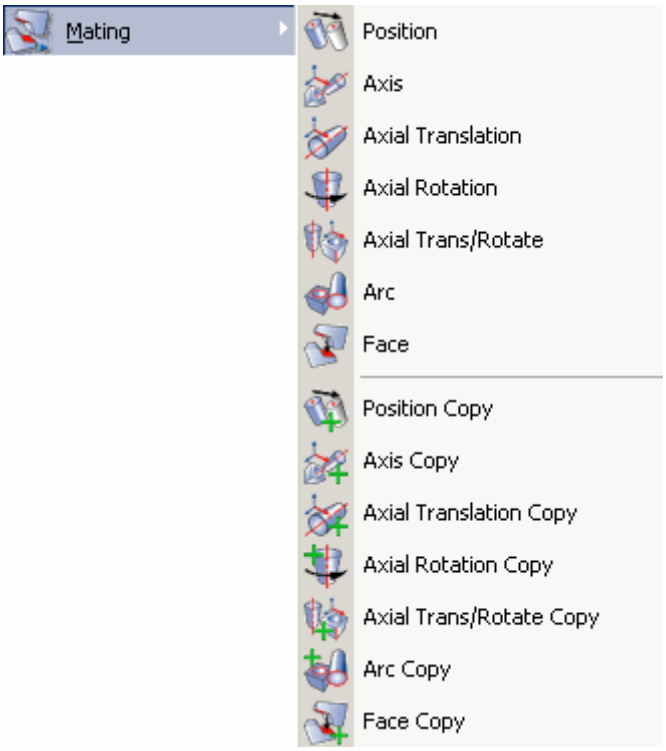
Location: Detail>Restyle Detail Ents>Lines and **Detail>Restyle Detail Ents>Tolerance**.
This release adds Restyle Detail Ents (that is, Restyle Detail Entities) to the Details menu. Restyle Detail Ents has two subfunctions: Lines and Tolerance. Their dialogs are shown below. As you can see in the dialogs, the Lines function allows you to modify how the detail lines of a dimension are displayed, and the Tolerance function allows you to modify the tolerance of a dimension.





Transform-Mate Moved to Transform Menu

Location: Transform>Mating
This release moves the Transform-Mate functions from the separate Qwins CDE (located on the Add-Ins menu) to the KeyCreator Transform menu. The functions are called "Mating" on the Transform menu, as shown below. These are useful for moving or copying existing geometry so that it is aligned with other geometry. The top seven on the menu (starting with "Position") are move functions, and the lower seven (starting with "Position Copy) are copy functions. See *Transform Mating* in Help for details.



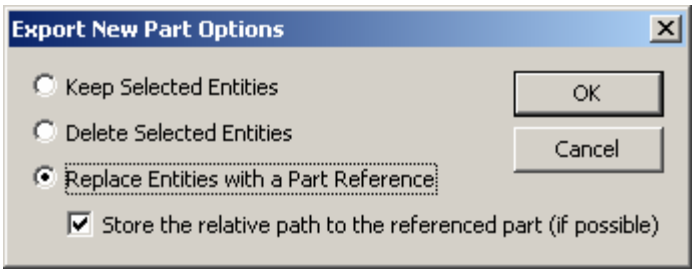
Projected Instance

Location: Layout>Projected Instance. This function allows you to create a projected instance of a drawing instance from a selected quadrant in the drawing instance. The function also allows you to create an isometric of a drawing instance, when you select from the top right in the drawing instance. In either case, the projected instance has all the same settings as the original instance. See *Projected Instance* in KeyCreator Help for more details.

Convert to Reference

Location: Assembly>Convert to Reference. This new addition to the **Assemblies** menu has the same functionality as the existing **File>Export>KeyCreator Part** function, when the **Replace Entities with a Part Reference** and **Store the relative path to the referenced part (if possible)** options are selected on the **Export New Part Options** dialog, shown below. (You can display that dialog by clicking **File>Export,** then following the prompts on **Conversation Bar**, and then clicking the **Options** button.)

The Convert to Reference function is a one-step alternative to the **File>Export>KeyCreator Part** function. Convert to Reference does not require opening the **Options** dialog and confirming that the correct options are selected.



Ability To Display Details through Part References Added

Location: Assemblies>Include Detail Entities. This release adds the Include Detail Entities function to the **Assemblies** menu, and the related

- **Include Detail Entities in Part Reference Display** check box to the **Options** pane of the **Creating a New Part Reference** dialog (Assemblies>Create Reference>select file>Options), and
- Display Details column to the **Part Reference** pane of the Parts Splitter

This allows a part reference to display details such as notes, labels, dimensions and crosshatches from the referenced part.

ENHANCED FUNCTIONS

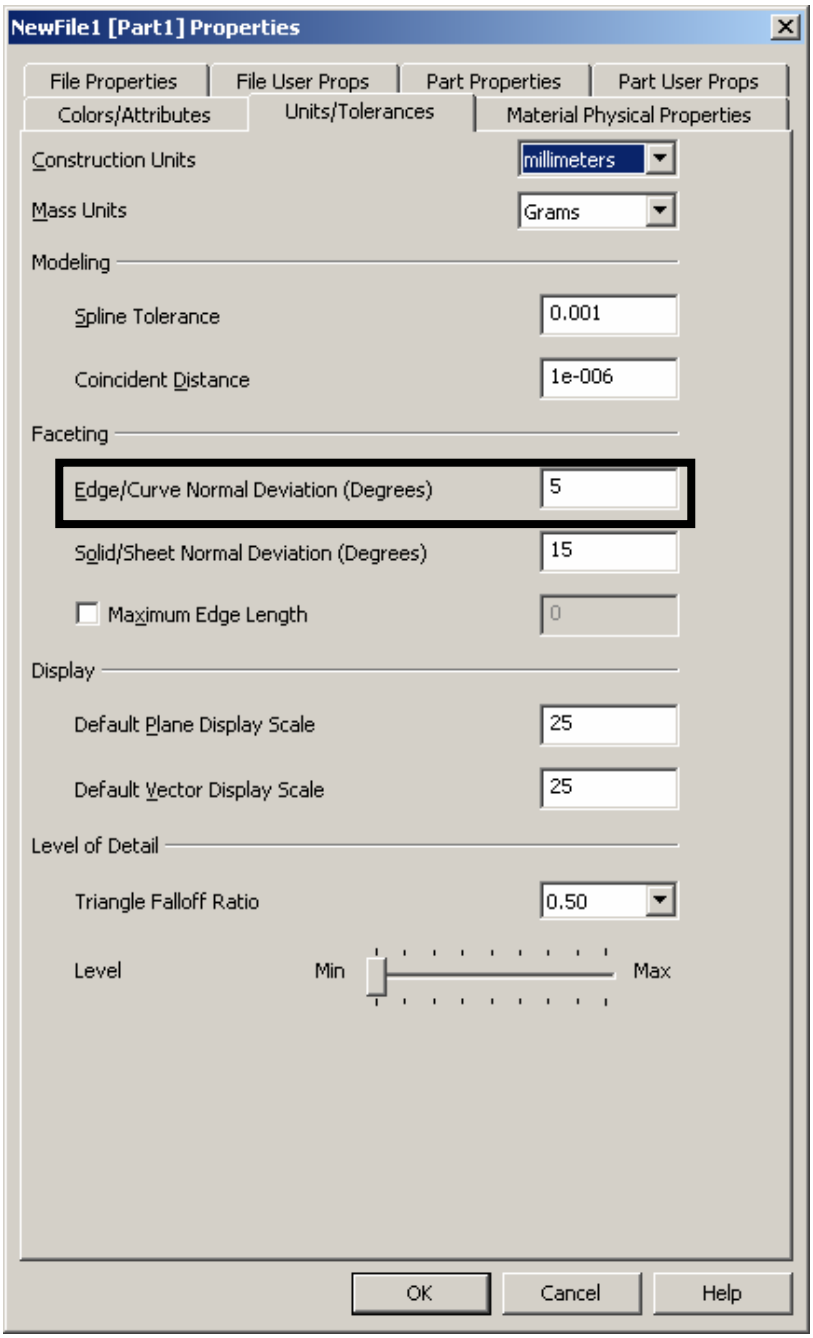
This release enhances several functions, as explained below.

Option Added to Units/Tolerances Pane of Properties Dialog

Location: File>Properties>Units/Tolerances. This release adds the following option to the **Units/Tolerances** pane as shown below:

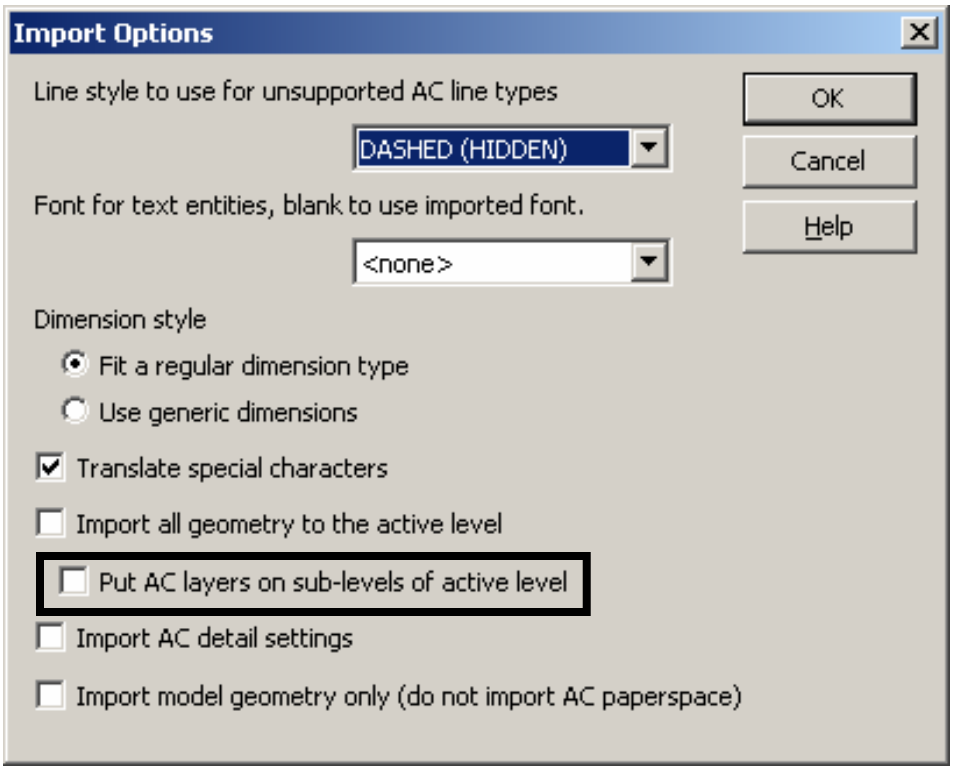
Edge/Curve Normal Deviation:
Allows you to define the tessellation for an arc, conic, spline and NURBS. It is the largest angle allowed between adjacent line segments in the tessellated representation.

Note that, in this release, the *name* of the **Normal Deviation (degrees)** option on this dialog was changed to **Solid/Sheet Normal Deviation (degrees)**: same function as previous release.


NewFile1 [Part1] Properties
File Properties
File User Props
Part Properties
Part User Props
Colors/Attributes
Units/Tolerances
Material Physical Properties
Construction Units
millimeters
Mass Units
Grams
Modeling
Spline Tolerance
0.001
Coincident Distance
1e-006
Faceting
Edge/Curve Normal Deviation (Degrees)
5
Solid/Sheet Normal Deviation (Degrees)
15
Maximum Edge Length
0
Display
Default Plane Display Scale
25
Default Vector Display Scale
25
Level of Detail
Triangle Falloff Ratio
0.50
Level
Min
Max
OK
Cancel
Help

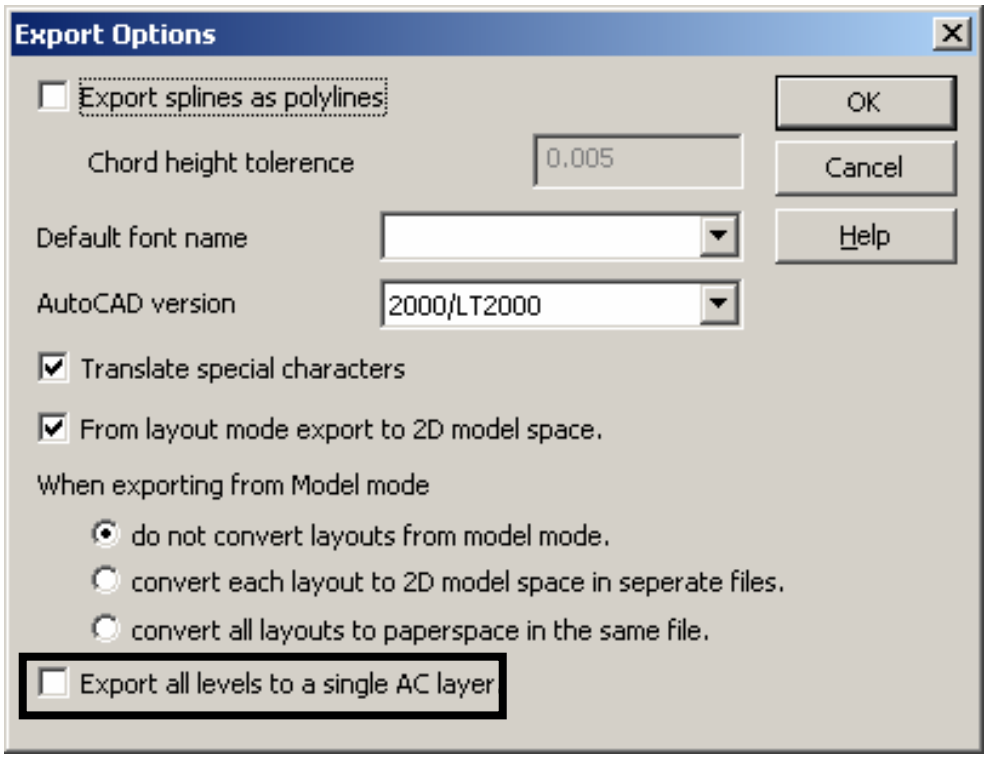
Option Added to DXF and DWG Import Dialog

Location: File>Import>AutoCAD DXF and **File>Import>AutoCAD DWF**. As shown below, this release adds the **Put AC layers on sub-levels of active level** check box on the **Import Options** dialog for importing an AutoCAD DXF or DWG file. ("AC" in the check box name stands for "AutoCAD.") When you select this option, the layer structure of the AutoCAD file remains intact. The layers are placed as sublevels under the currently active level in KeyCreator. (Levels are visible in the KeyCreator Part Splitter.)



Option Added to DXG and DWG Export Dialog

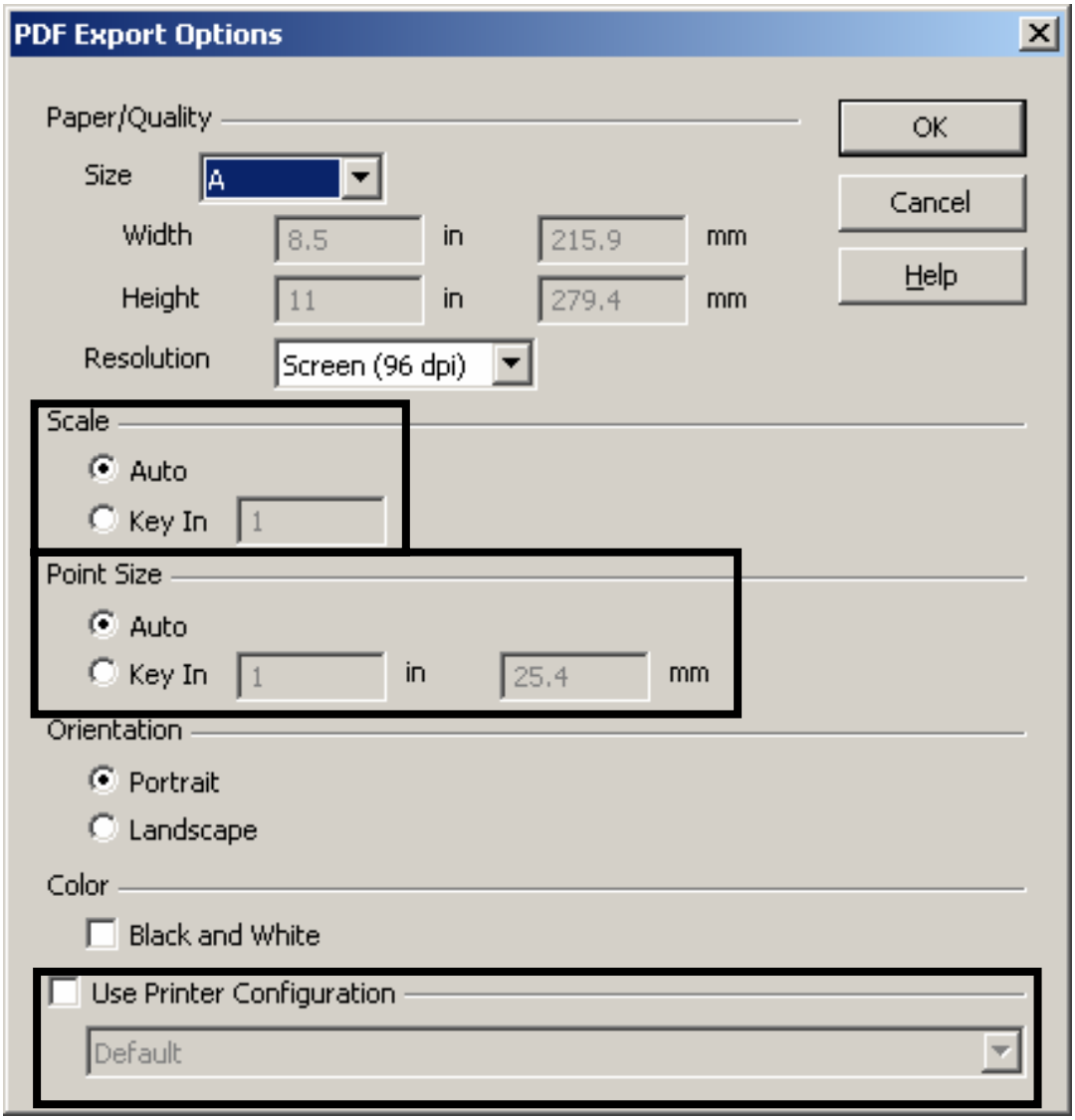
Location: **File>Export>AutoCAD DXF** then click the **Options** button, and **File>Export>AutoCAD DWG** then click the **Options** button. As shown below, this release adds the **Export all levels to a single AC layer** check box to the **Export Options** dialog. ("AC" in the name of this check box stands for "AutoCAD.") When you select this option, all of the geometry in the KeyCreator file is flattened to a single layer in the exported file. This is useful for exporting layouts.



Options Added to PDF Export

Location: File>Export>PDF. This release adds options to the **PDF Export Options** dialog, explained and shown below.

Scale determines the ratio of part size to paper size. **Point size** specifies the size of a printed point entity. Selecting a print configuration in the **Use printer configuration** field causes the applicable settings of the Print/Plot configuration (**Tools>Options>Print/Plot**) to be used for the export. For details on all options on the **PDF Export Options** dialog, see *PDF Export* in KeyCreator Help. For information on print configuration, see *Print/Plot* and related links in KeyCreator Help.



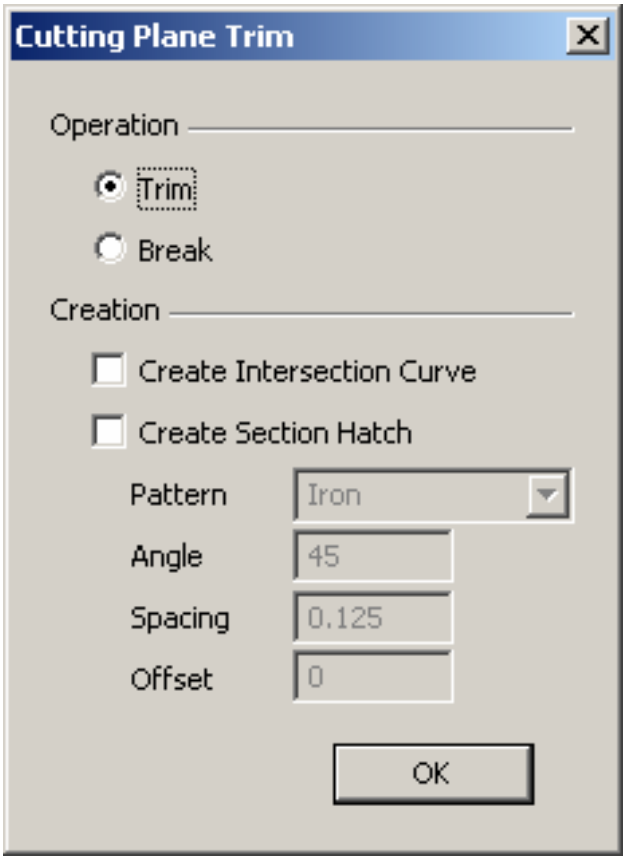
Smooth Shaded Now Named Gouraud Shaded

Location: View>Render>Gouraud Shaded. With this release, the function previously called Smooth Shaded, under V**iew>Render**, is now called Gouraud Shaded. The function remains the same: only the name has changed. See the related *Phong Shaded Rendering* on page 6.

New Dialog Added for Cutting Plane Trim Dialog

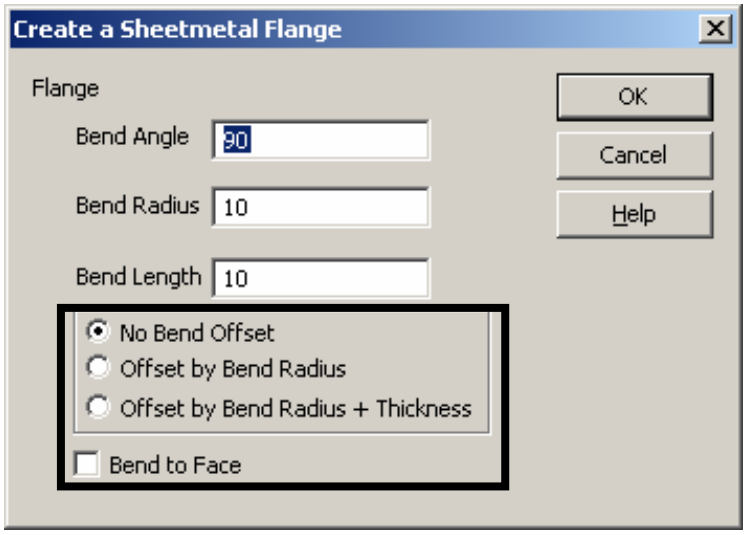
Location: View>Cutting Plane>choose plane**>Trim**, and **View>Cutting Section.** This release adds the **Cutting Plane Trim** dialog shown below. This dialog allows you to

- Specify either a trim or break operation
- Create lines and curves at the intersection of solids and cutting plane
- Create hatch at the intersection of solids and cutting plane. As the hatch, you can select pattern, angle, spacing or offset.

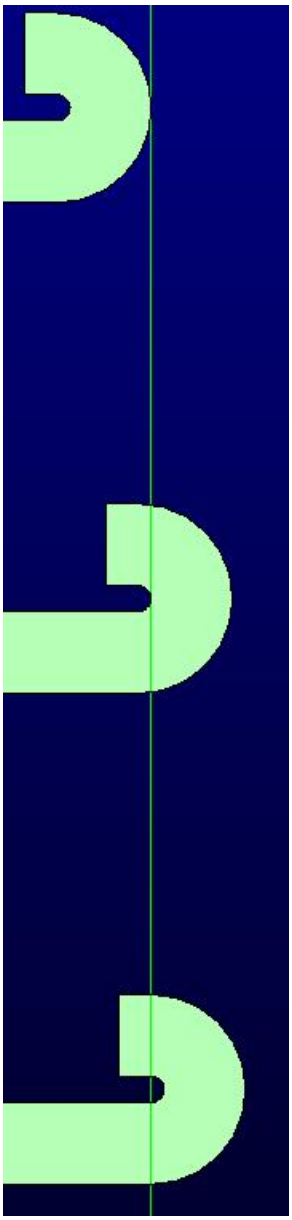


Options Added to the Create a Sheetmetal Flange Dialog

Location: Create>Solid Feature>Sheetmetal Flange. This release adds three radio buttons and a check box to the dialog, as shown below.

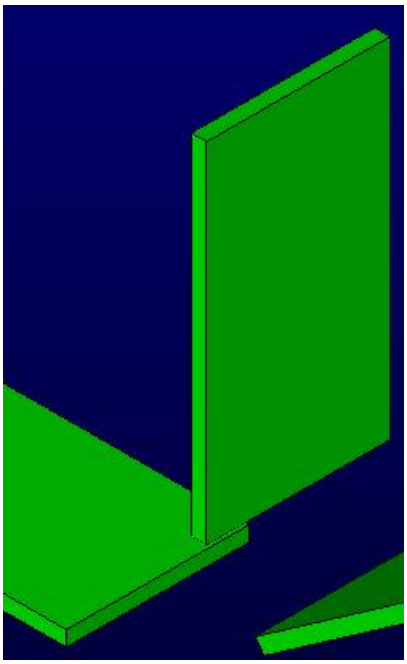


No Bend Offset, **Offset by Bend Radius**, **Offset by Bend Radius + Thickness** – These options allow the start of the bend to be offset backward. This allows for common sheetmetal operations where you want the sheetmetal part to maintain its dimensions (as opposed to building the part outward). The figure below illustrates each of these selections, respectively. The line in the figure corresponds to the face originally selected to create the flange. The **No Bend Offset** option matches the behavior of previous releases.

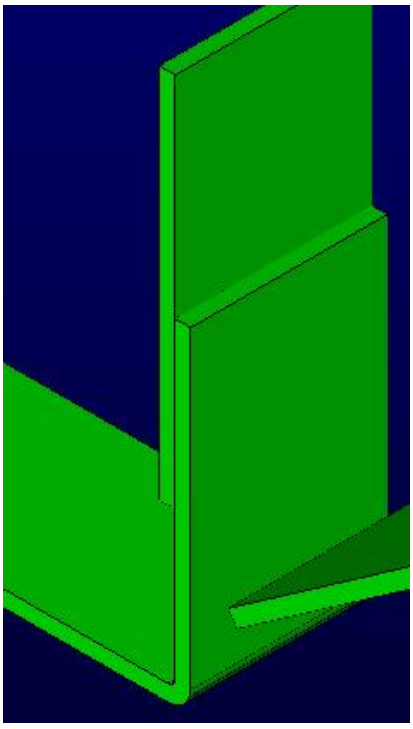


Bend to Face. See the before and after examples below. This option allows you to select another face while creating the flange. The flange angle and offset are automatically set to cause the face of the flange to be close and parallel to the selected face. A slight gap is left between the two faces so that the modeler will not attempt to merge the two.

Before:



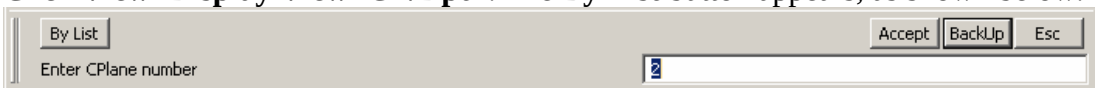
After:



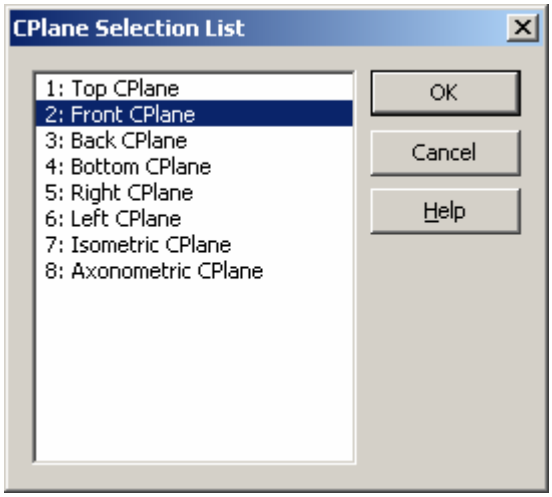
CPlane for "CP/Depth" (on Plane Definition Menu) Selectable from List

Location: Various, such as **View>Display View**. This release adds the **By List** button, shown below, on the **Conversation Bar** for the CP/Depth option. The By List button allows you to select the CPlane from the **CPlane Selection List** dialog, instead of having to key-in the CPlane number.

- Click **View>Display View>CP/Dpth**. The **By List** button appears, as shown below.



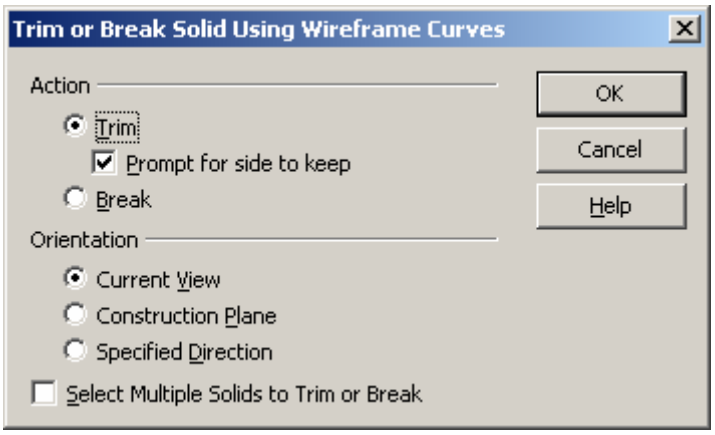
- Click the **By List** button. The **CPlane Selection List** dialog appears, as shown below.



- Select the desired CPlane from the list. The selection applies to the model.

Option Added to Solids Trim/Break Dialogs

Location: Modify>Solids Trim/Break>Curve, **Modify>Solids Trim/Break>Sheet Body/Faces** and **Modify>Solids Trim/Break>Plane**. This release adds the **Select Multiple Solids to Trim or Break** check box to the dialogs that appear for each of the three functions Curve, Sheet Body/Faces and FPlane. The example dialog below is for **Modify>Solids Trim/Break>Curve**.



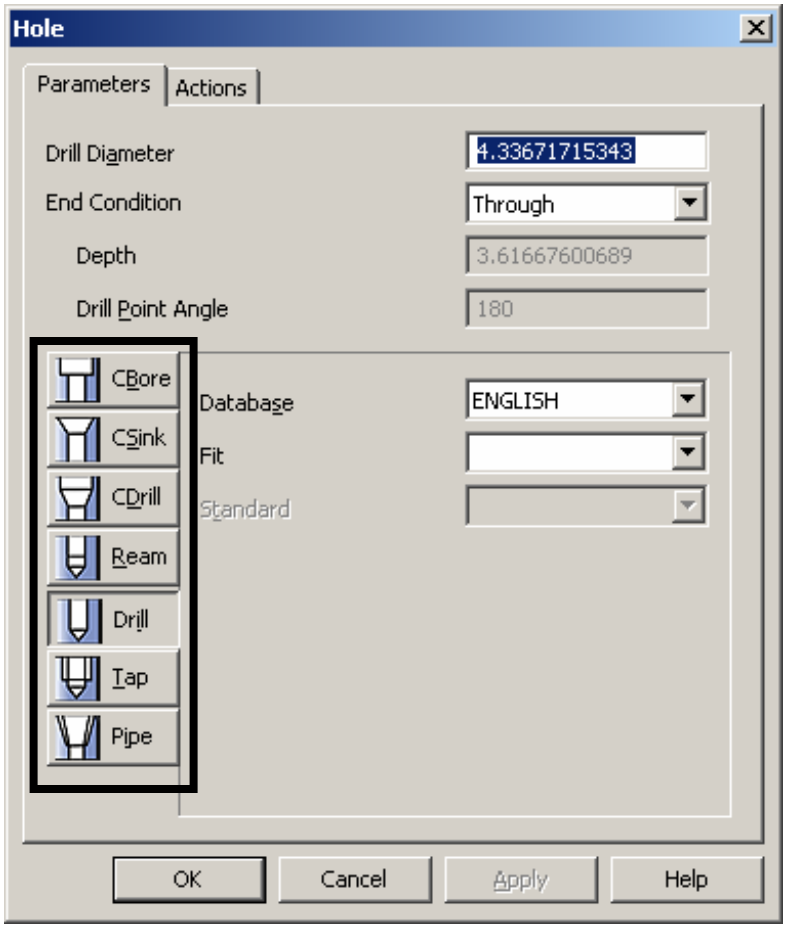
When selected, this option allows you to select multiple solids/sheets to trim. Once you click **OK**, you are prompted for the trimming entity/entities. The next step is to select one or more solids/sheets to trim.

When cleared (default), after you click **OK**, you are prompted for the trimming entities. The next step allows you to select one solid/sheet to trim/break. Once the operation is complete, you are prompted to select another solid/sheet to trim. If you are done trimming, selecting the ESC key exits the function.

Before this release, you were asked to select the solid/sheet to trim/break, followed by the selection of trimming entities, after which the function automatically exited.

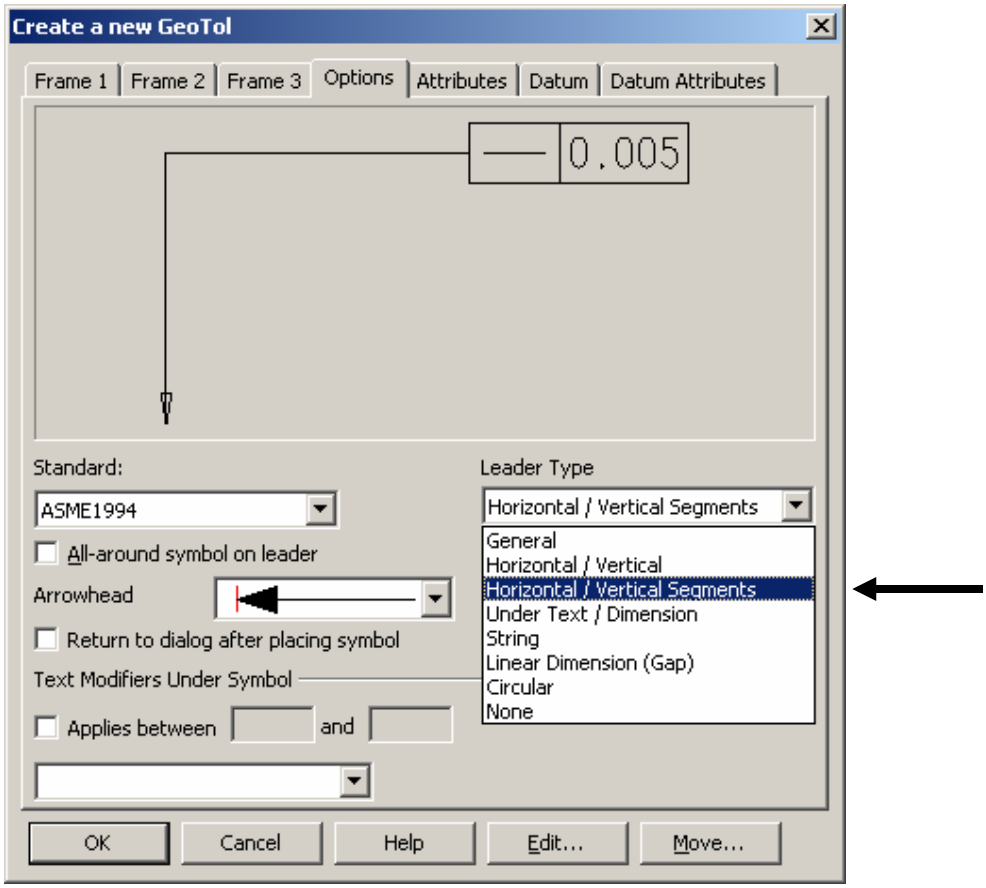
Icons Added To Buttons on Edit Dialog

Location: Modify>Solid Feature>Hole. This release changes the appearance of buttons located on the **Edit** dialog, as shown below. These icons are taken from the **Tools>Feature** menu. The change is aesthetic only: it has no affect on the functions of the buttons.

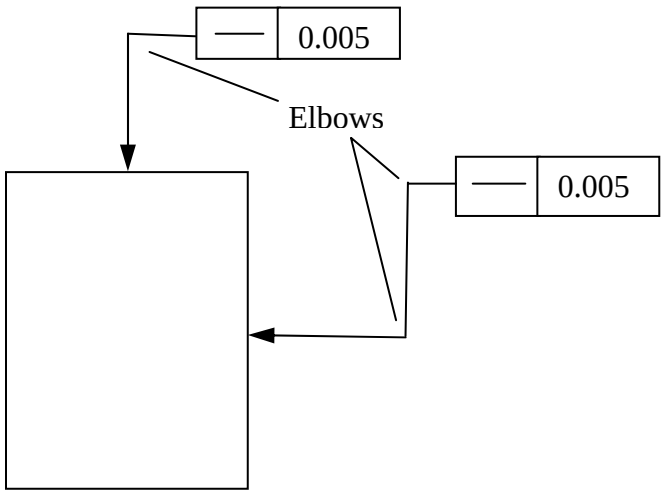


Selection Added to Allow Leader Elbows

Location: Detail>Symbol>Geometric Tolerance Symbol. This release adds the **Horizontal/Vertical Segments** selection to the **Leader Type** field list on the **Options** pane of the **Create a New GeoTol** dialog, as shown below.

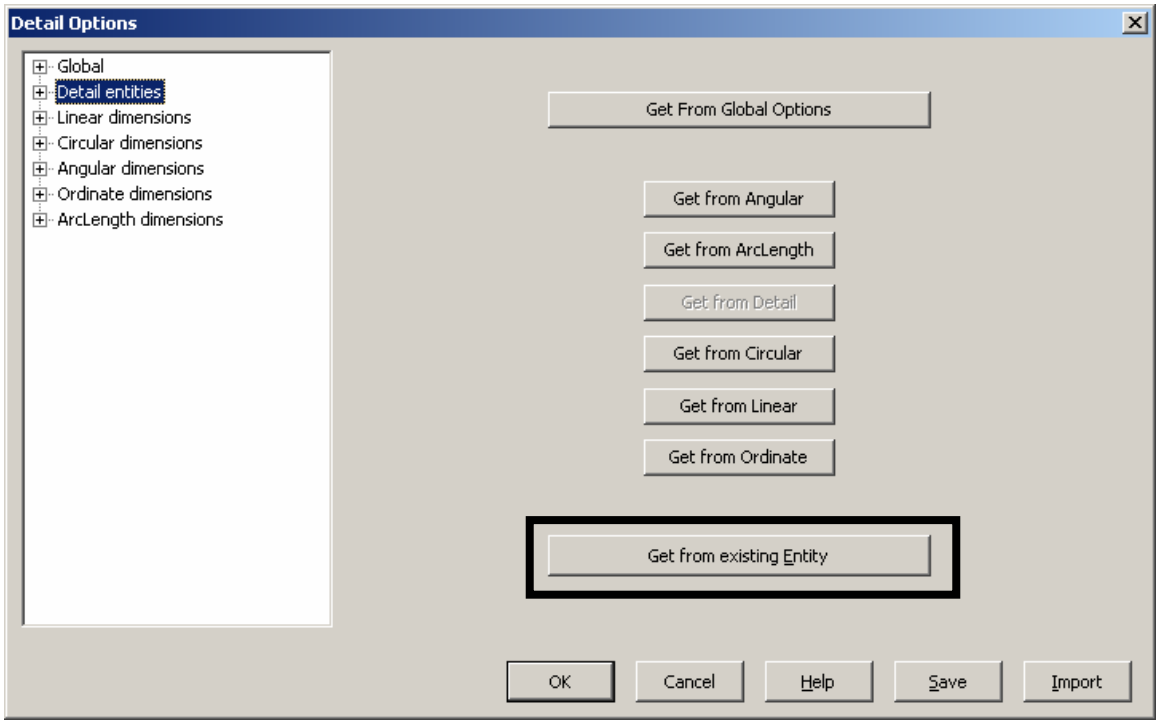


As illustrated below, selecting Horizontal/Vertical Segments allows the leader line to have one or two elbows, depending on where you place the leader in the KeyCreator viewport.



Get from Existing Entity Button Added to Detail Options Panes

Location: Detail>Settings>Settings. This release adds the **Get from existing entity** button to the pane of each top-level node of the **Detail Options** dialog (**Global**, **Detail entities**, **Linear dimensions**, **Circular dimensions**, **Angular dimensions**, **Ordinate dimensions** and **ArcLength dimensions**). The button is illustrated below. If you already have dimensions on the currently displayed part, this button allows you to use these to set default settings for that part.

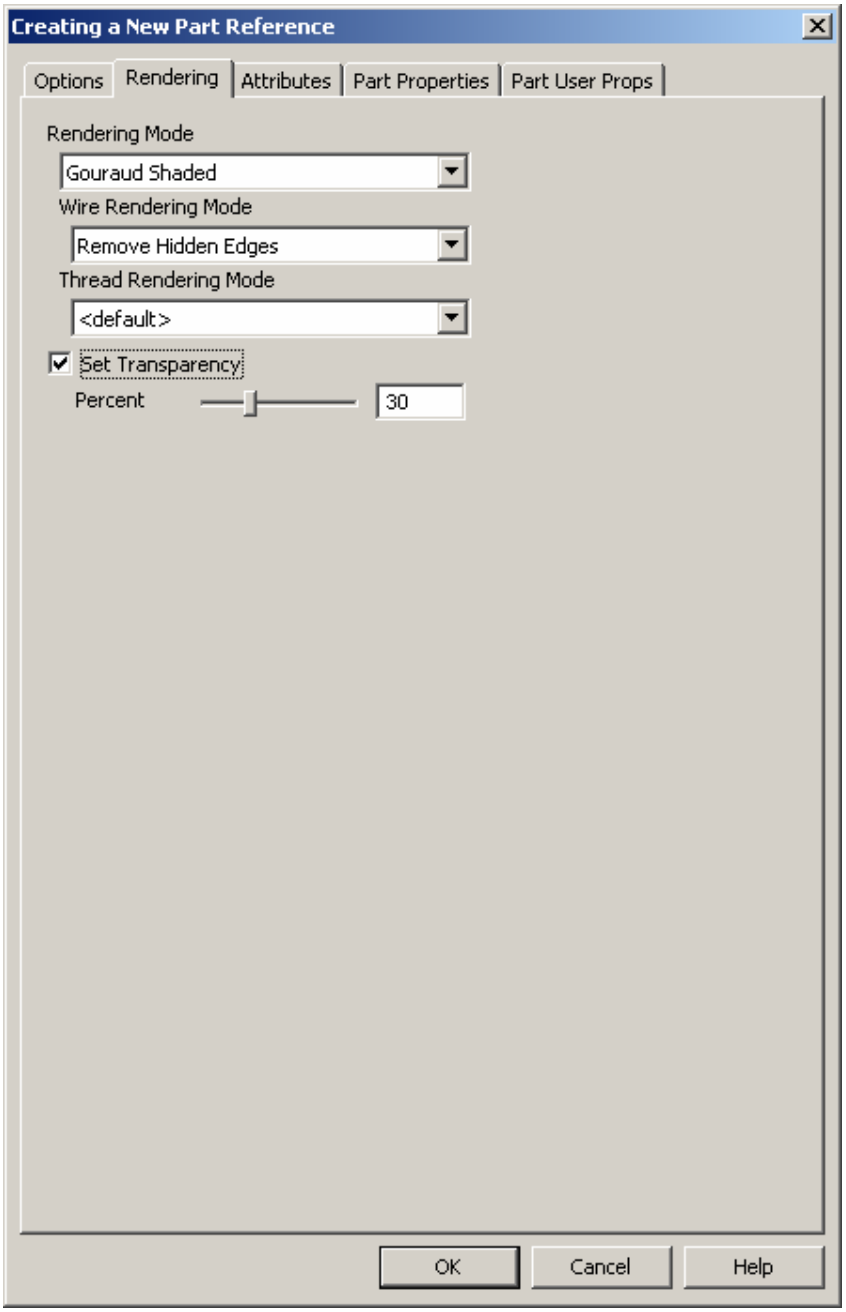


Active Instance Function Removed

Location: Layout>Active Instance. This release removes the Active Instance functionality. The presence of an active instance often resulted in confusion when users tried to use layout functions. It often resulted in unexpected results because there was no active instance, or it was set to a different instance than the one the user was trying to modify. The three functions that used Active Instance (namely, Detail>Crosshatch, Detail>Detail Lines>Centerline and Edit>Entities>Attributes by Selection) instead of explicitly asking the user to select an instance now have a '"select drawing instance" prompt on the Conversation Bar in layout mode. Because of the removal of Active Instance, the Active Instance Border term on the Color/Attributes pane of the Properties dialog (File>Properties>Color/Attributes) is now named Layout Border. The Inactive Instance Border term on the same pane is now named Instance Border.

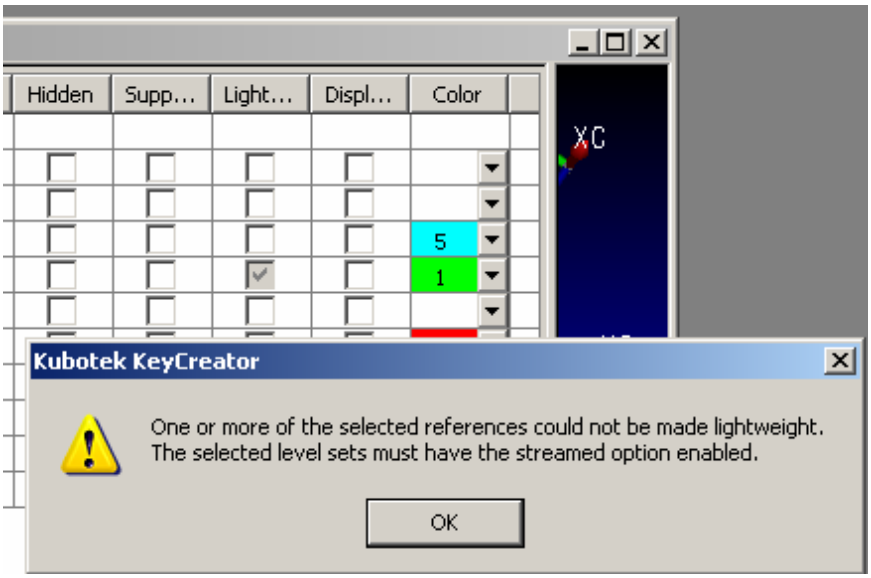
Rendering Pane Added for Part Reference

Location: Assemblies>Create Reference>Rendering and **Edit>Entities>Edit>Rendering**. This release adds the **Rendering** pane to the **Create a New Part Reference** dialog, shown below. (For a part reference, the **Rendering** pane also is added to the **Editing Part Reference** dialog.) This pane allows you to select the desired mode for rendering the part reference or to select the **Set Transparency** check box and set the desired degree of transparency (0 to 100). When creating a part reference, you can select the desired rendering using **Assemblies>Create Reference>Rendering**. When editing a part reference, use **Edit>Entities>Rendering**.



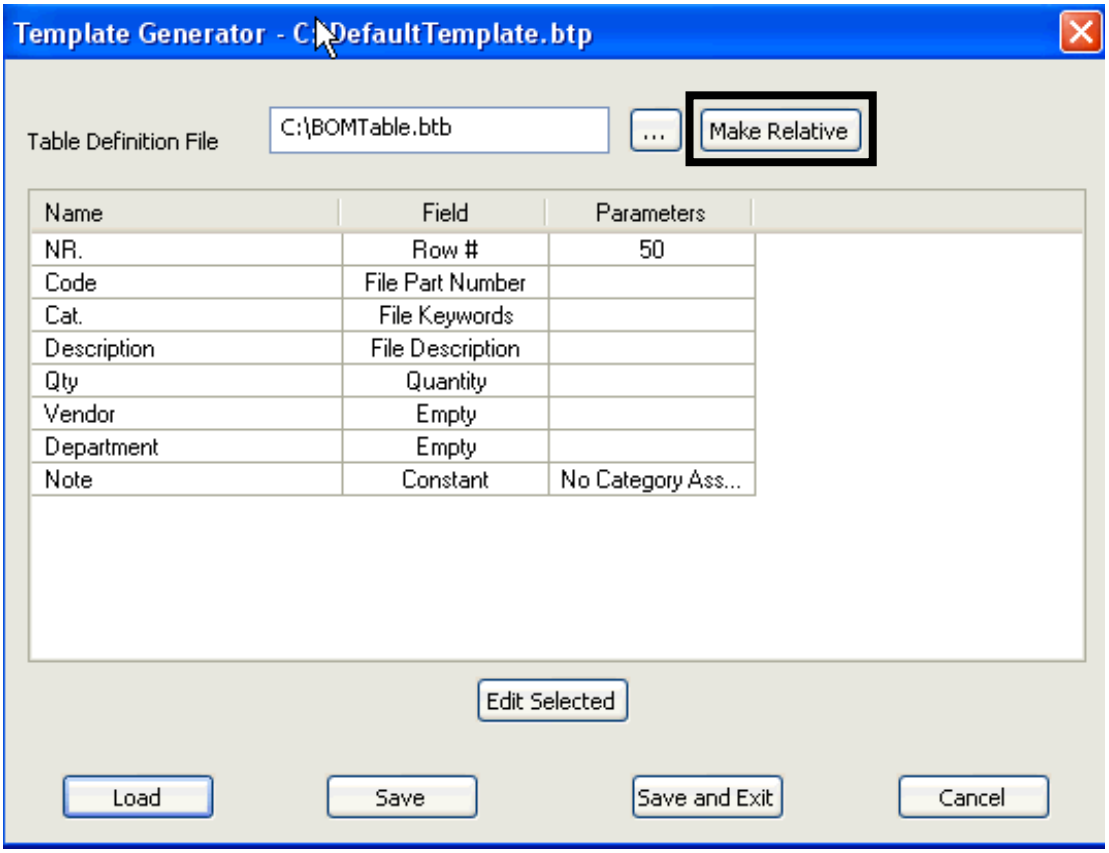
Message Added when Lightweight Does Not Occur

Location: Assemblies>Lightweight, and **Part Splitter>Part References>**then select **Lightweight** for the reference. With this release, when you attempt to apply the Lightweight function to a part reference and the function does change the part to lightweight, a message appears indicating this. As shown below, with the message, in the **Lightweight** column of the **Part Reference** pane of the **Parts Splitter**, the check box in the part's row is shown unavailable (gray). Usually, the problem is that the **Streamed** option on the **Level Sets** pane of the **Parts Splitter** is not selected, as the message indicates.



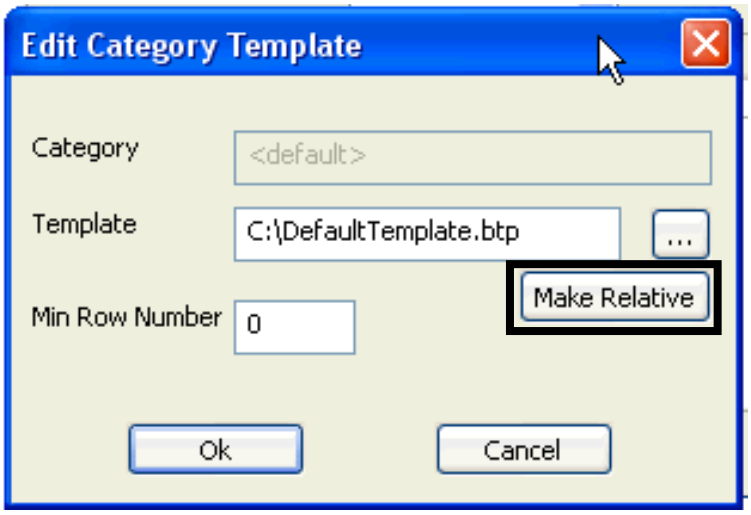
Make Relative Button Added to Create Template Dialog

Location: Tools>BOM and Table>Create Template. This release adds the **Make Relative** button on the dialog shown below that appears when you select the Create Template function. The button allows you to make the template file path relative to the part (that is, to the open .ckd file).

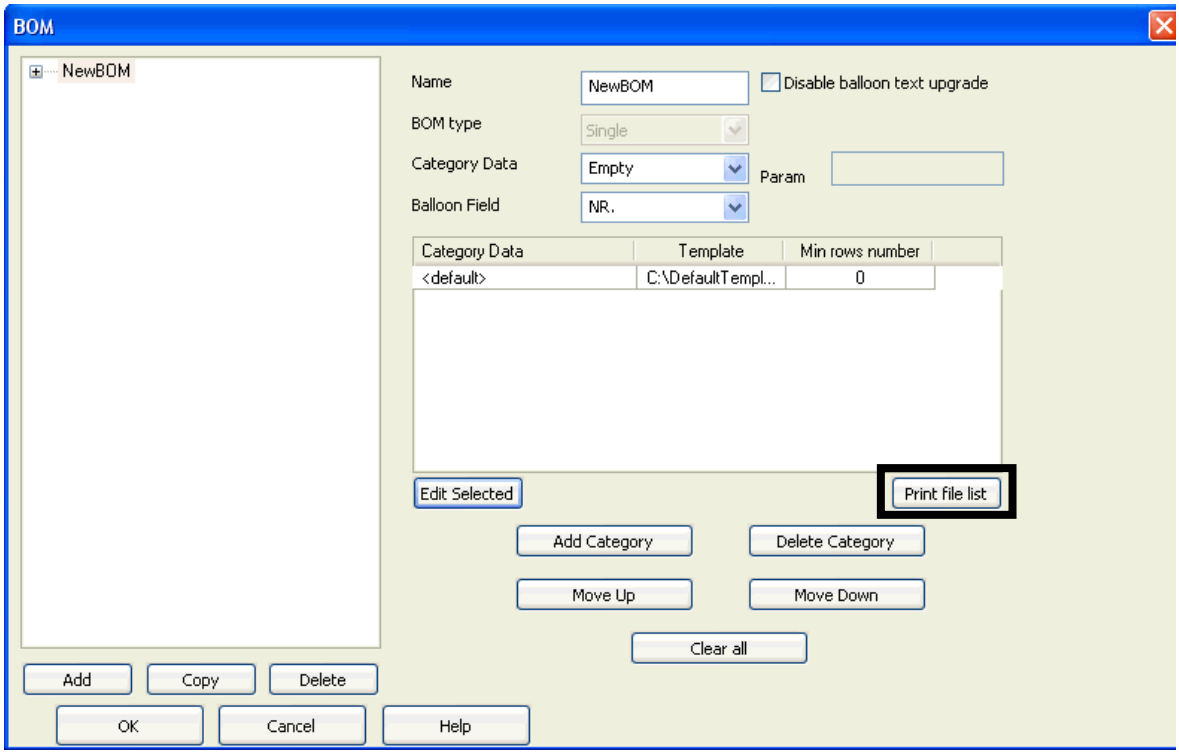


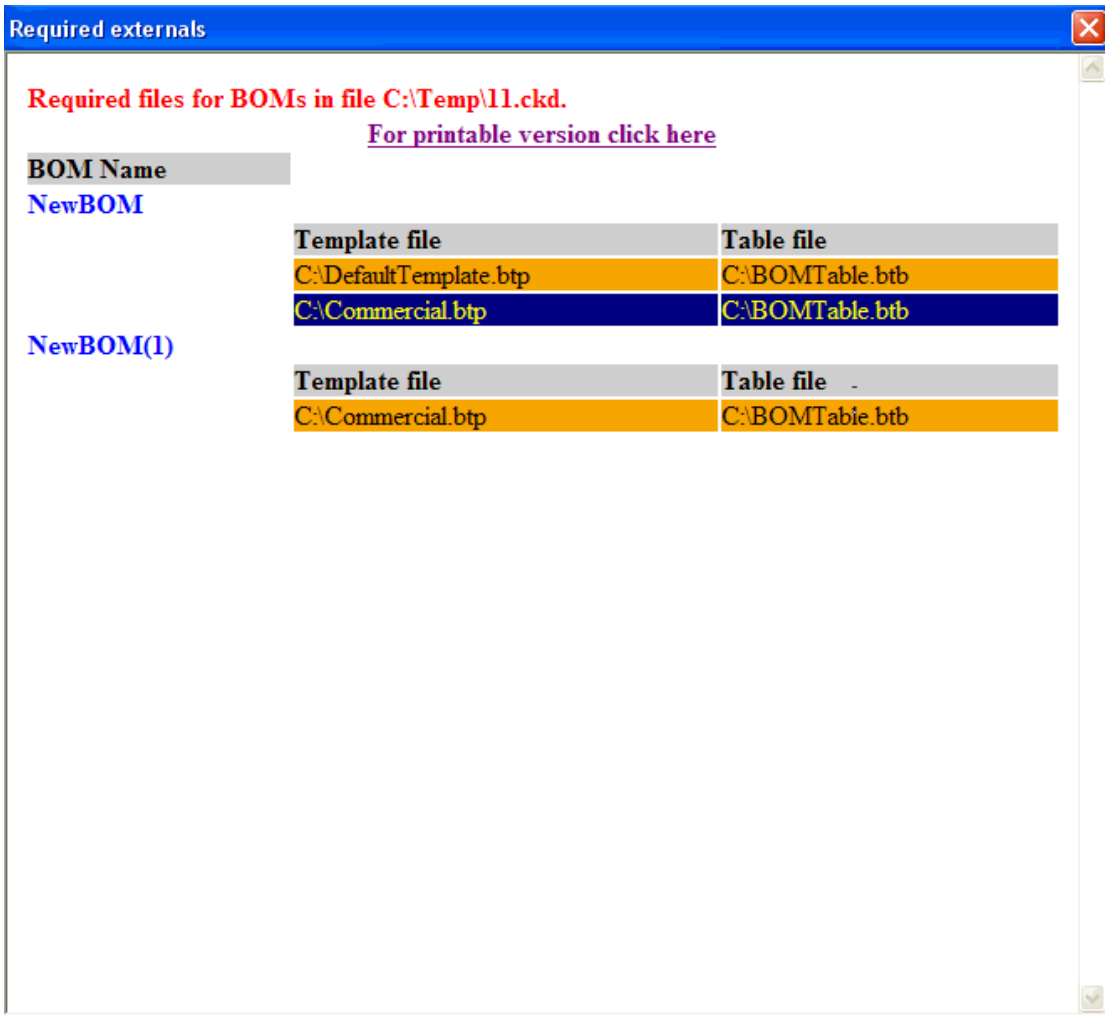
Buttons Added to Create BOM Dialogs

Location: Tools>BOM and Table>Create BOM. This release adds buttons to several dialogs that appear when using the Create BOM function. It adds the **Make Relative** button to the **Edit Category Template** dialog, shown below, which appears when you click the **Add** button on the **BOM** dialog. See *Button Added to Create Template Dialog* above for an explanation of this button.

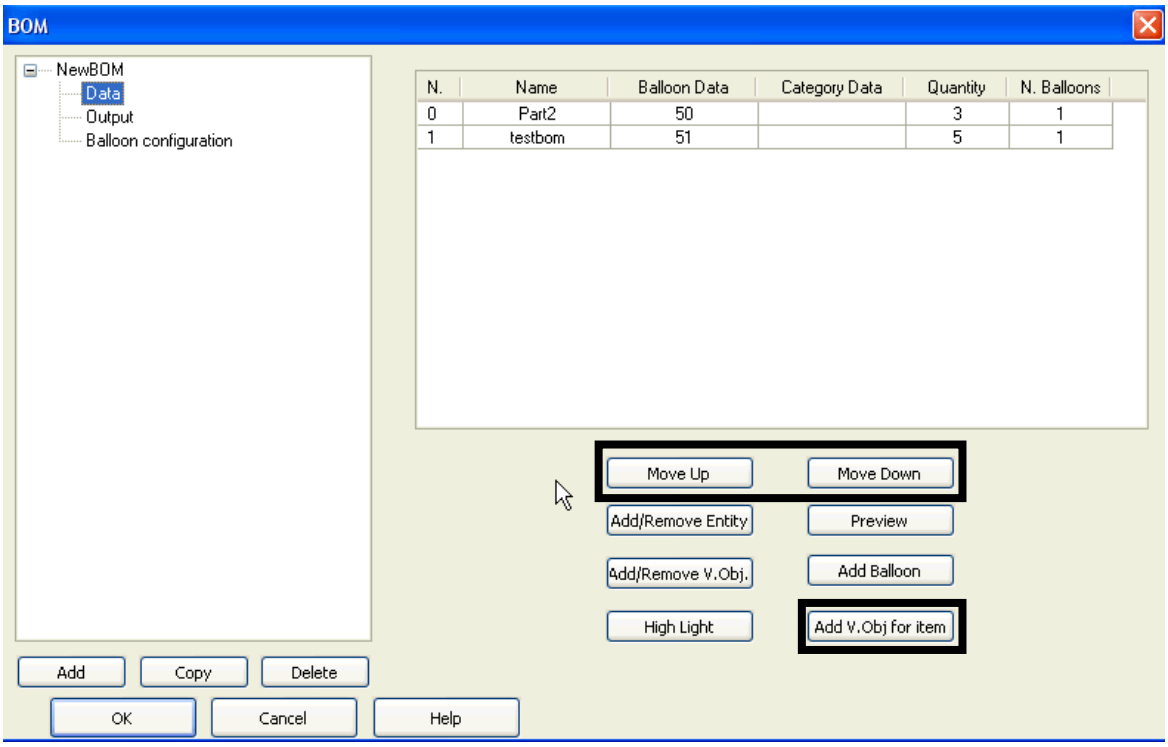


The release adds the **Print file list** button to the **NEWBOM** pane of the **BOM** dialog. This button prints a report like that shown below. The report indicates the files used by all the BOMs for the current part.




Required externals
Required files for BOMs in file C:\Temp\11.ckd.
For printable version click here
BOM Name
NewBOM
Template file
Table file
C:\DefaultTemplate.btp
C:\BOMTable.btb
C:\Commercial.btp
C:\BOMTable.btb
NewBOM(1)
Template file
Table file
C:\Commercial.btp
C:\BOMTable.btb

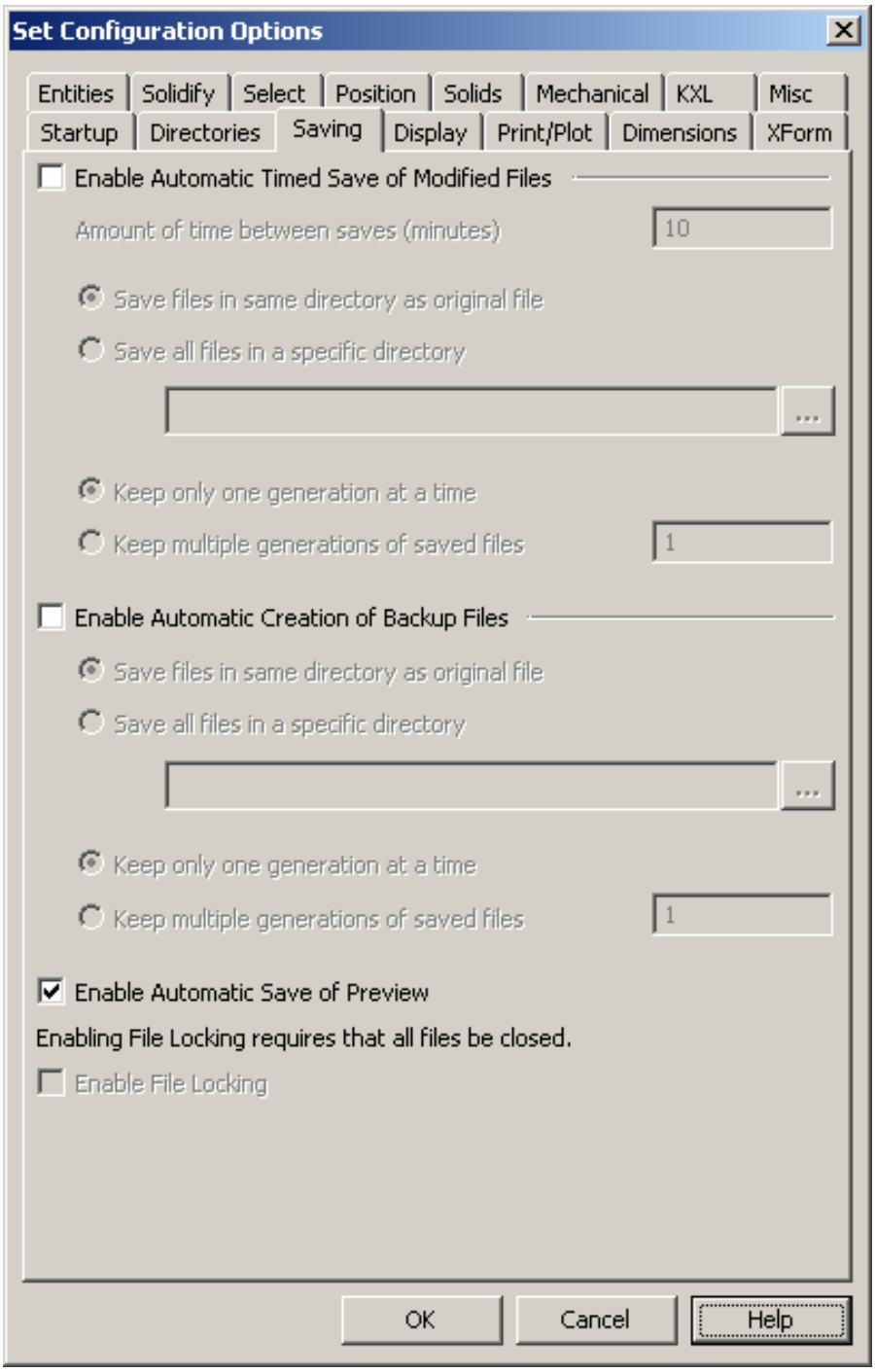
The release adds the **Move Up**, **Move Down** and **Add V. Obj from item** buttons to the **Data** pane of the **BOM** dialog, as shown below. **Move Up** and **Move Down** allow you to determine the order in which items appear on the BOM. Clicking **Add V. Obj from item** displays the same dialog that appears when you select that **Tools>BOM and Table>Create Virtual Object** function. This dialog allows you to generate virtual objects on the BOM.



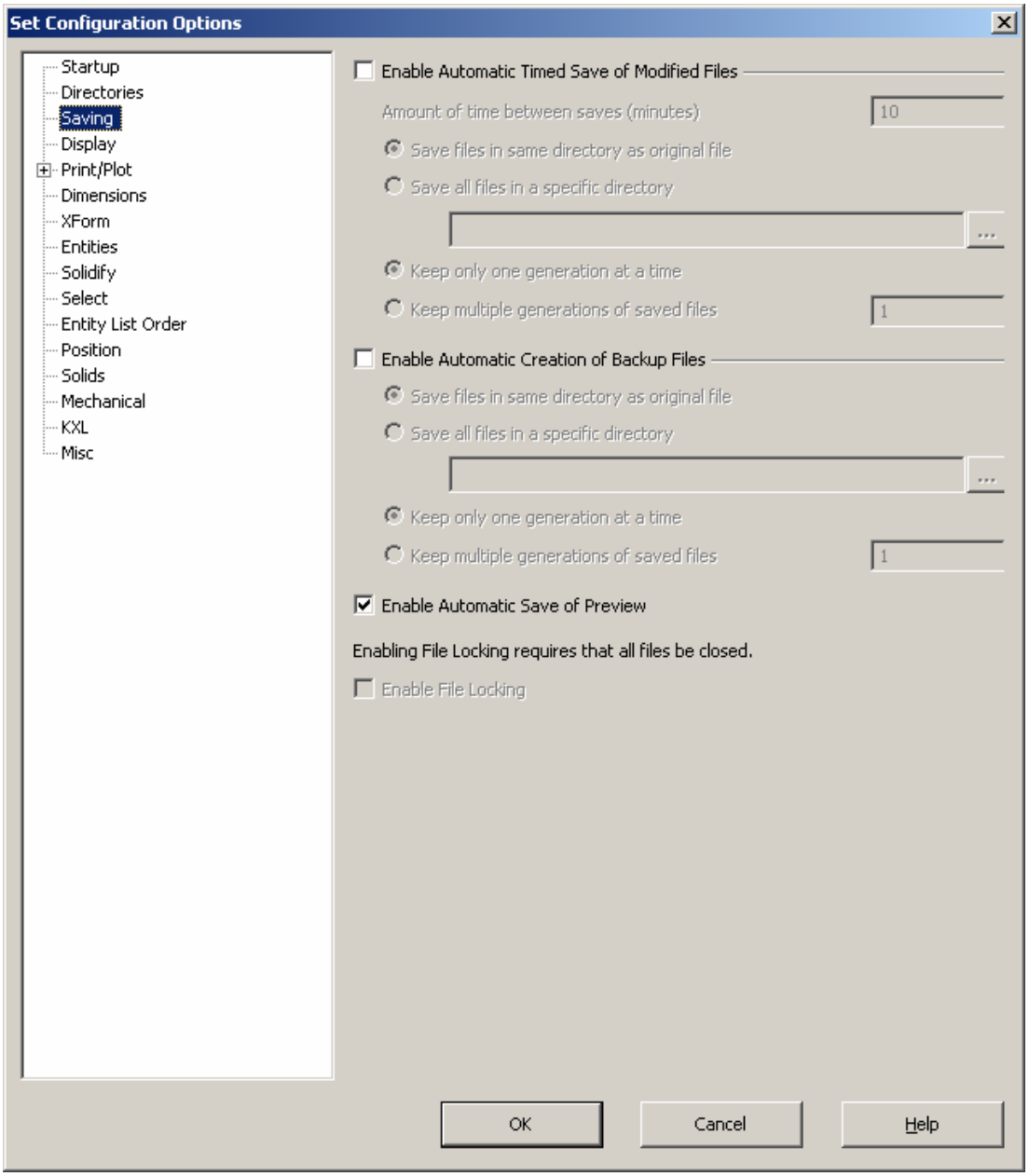
Options Dialog Panes Changed from Tabs to Tree Structure

Location: Tools>Options. This release makes changes to the **Set Configuration Options** dialog to make it have a tree structure. Examples of the previous and new panes, showing the **Saving** pane selected as the example, are shown below. For the appearance and details of each pane, see the *Set Configuration Options Panes* topic in Help.

Before this release:

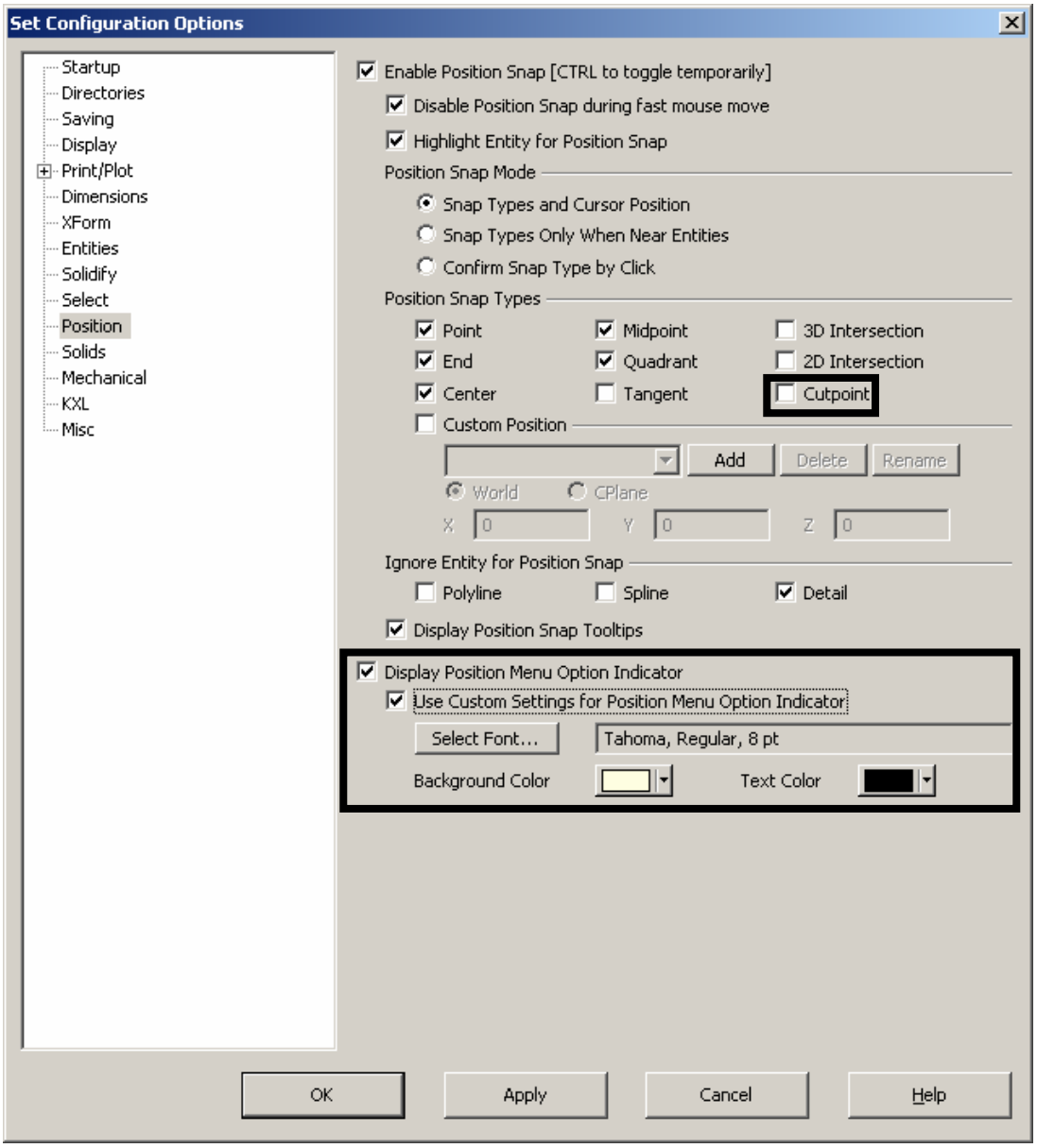


New with this release:



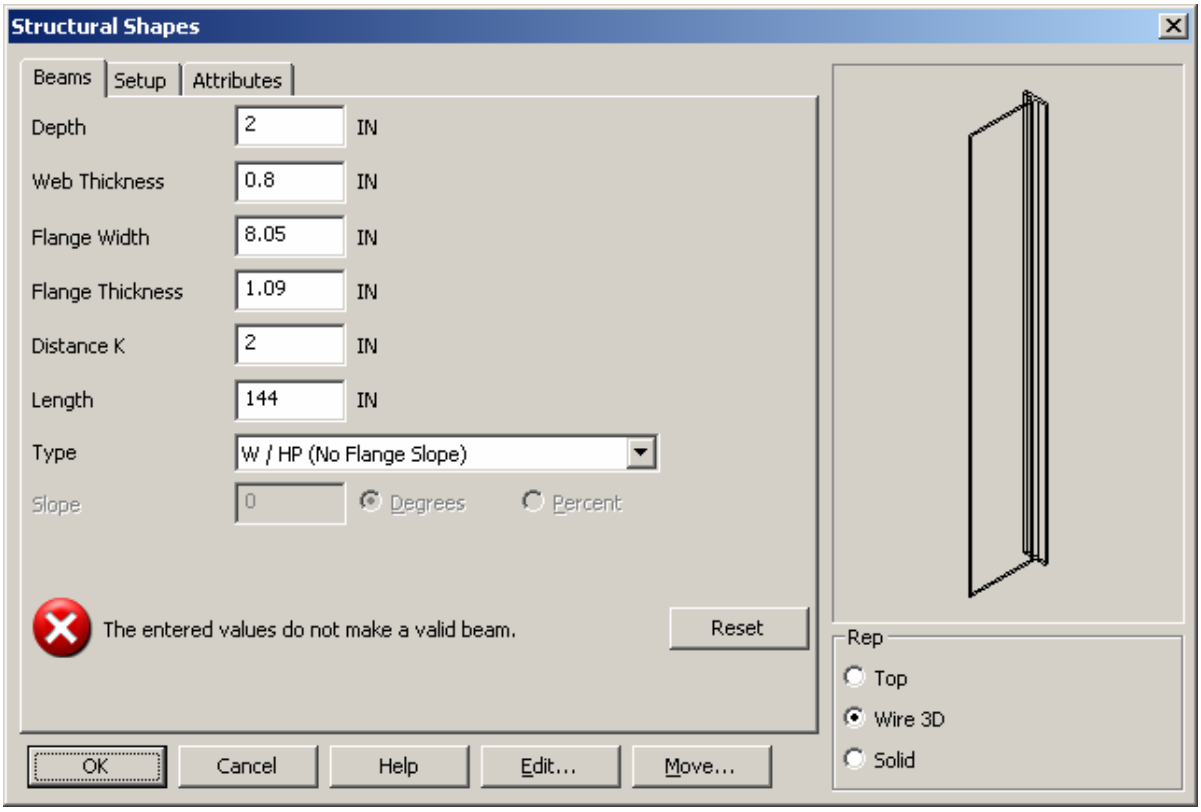
Options Added to Position Pane of Set Configuration Options Dialog

Location: Tools>Options>Position. This release adds the options indicated below. **Cutpoint** allows you to snap to the intersection of a wireframe entity (line, polyline, arc, conic, spline, NURBS) and cutting plane. When you select the **Display Position Menu Option Indicator** check box, the **Use custom settings for position menu option indicator** check box becomes available. This allows you to configure the custom settings shown under the check box for the position menu option indicator.



Error Message and Reset Button Added to Structural Shapes Dialog

Location: Tools>Structural Shapes. This release adds **The entered values do not make a valid beam** error message to the **Structural Shapes** dialog, plus a **Reset** button to it, as shown below. The dialog allows you to enter values that do not create a valid shape. But when you do so, this error message and the **Reset** button appear. Clicking the **Reset** button restores the last set of valid values in the dialog and makes the error message disappear. (Prior to this release, when you entered an invalid value in a field, KeyCreator displayed an error message. After you cleared the error message, the invalid value in the field was changed back to the original value.)



Record KXL Macro Supports More Functions

Location: Add-Ins>Record KXL Macro. Beginning with this release, the Record KXL Macro function supports the following functions:

Modify>Trim:
Both
Divide
Double
First
Modal
SmartSingle
SmartBoth
Position

Modify>Break:
Both
Divide
Double
First
Modal
Position

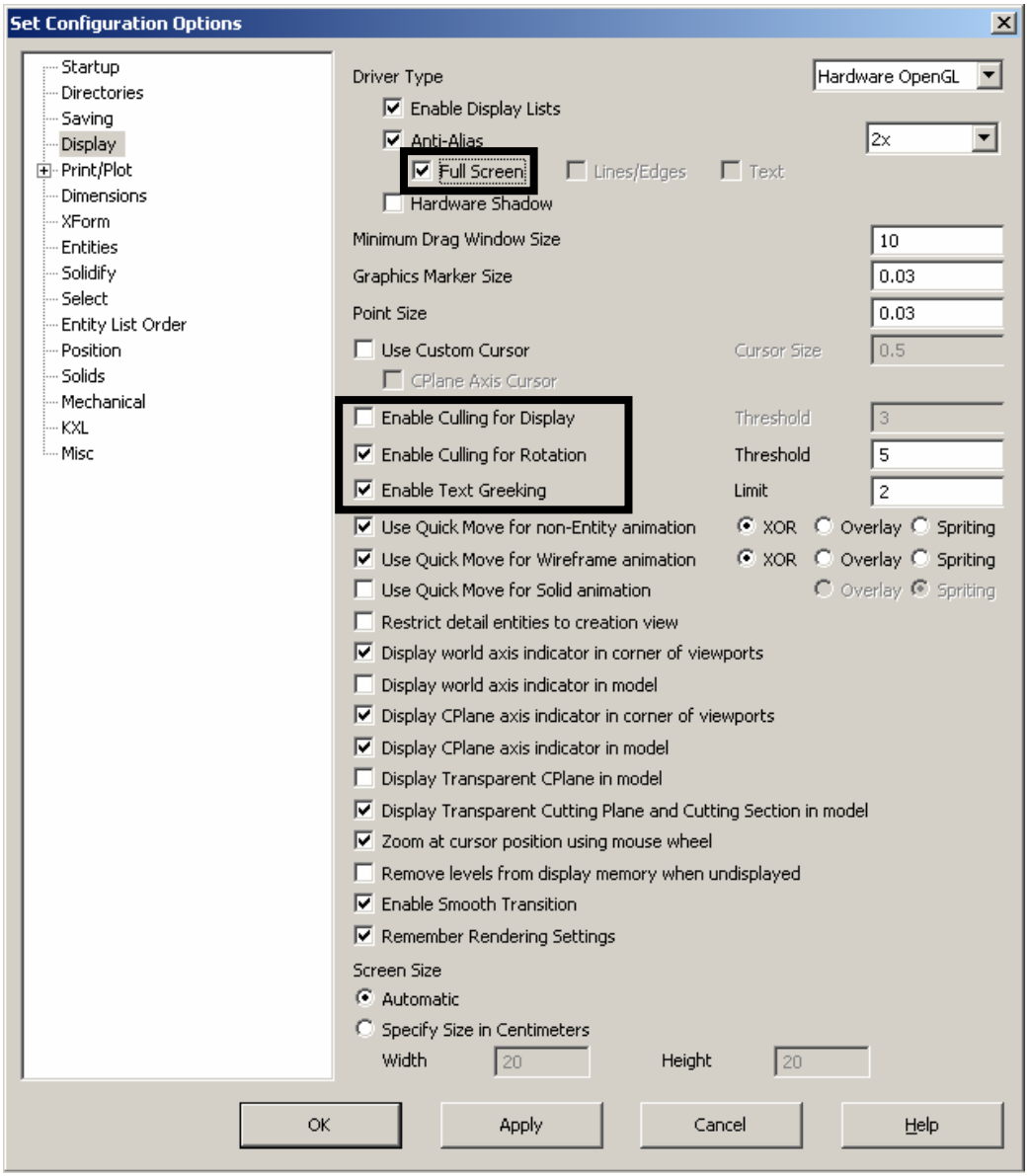
Modify>Fillet:
Trim
No Trim

Modify>Chamfer:
Trim
No Trim

Options Added to Display Pane of Set Configuration Options Dialog

Location: Tools>Options>Display. This release adds the following options to the **Display** pane as shown below:

Full Screen: When selected, the anti-aliasing applies to everything displayed.
Enable Culling for Display: When selected, solids and shells whose pixel size is less than the value specified in the **Threshold** field are not displayed.
Enable Culling for Rotation: When selected, solids and shells whose pixel size is less than the value specified in the **Threshold** field are not displayed during model rotation.
Text Greeking: When selected, KeyCreator substitutes a grid of lines for displayed True Type text that is less than the point size specified in the **Limit** field. (1 point – 1/72 inch).



NEW SDK FUNCTIONS

To see any new SDK functions for this release, see the SDK Help. To do this, run the install program (setup.exe) located in the Debug folder on the Kubotek KeyCreator product CD. Then, click the following Help file: x/SDK/DOCS/SDK.hlp, where "x" is the install folder. Then click the **What's New in the KeyCreator.sdk** link.

NEW KXL FUNCTIONS

To see any new KXL functions for this release, see the product's KXL Help (**Help>KXL Help>What's New>New Functions for KC8.0**). Note that, if you are running KeyCreator with the Windows Vista operating system, to view *KXL* Help you must install the Windows Help program, whose filename is `WinHlp32.exe`. Follow the download instructions at **http://www.microsoft.com/downloads/details.aspx?FamilyId=6EBCFAD9-D3F5-4365-8070-334CD175D4BB&displaylang=en**.
This is necessary because the Windows Vista operating system does not include `WinHlp32.exe`, the program necessary to read WinHelp files. (Previous Windows operating systems include this file.)

DOCUMENTATION ONLY

The following information is relevant to documentation only.

HTML Help with This Release

Starting with this release, the Help system is HTML based rather than WinHelp based. (This eliminates the need for users who have the Vista operating system to download and install the Windows Help program, whose filename is `WinHlp32.exe`. The Windows Vista operating system does not include `WinHlp32.exe`, the program necessary to read WinHelp files. Previous Windows operating systems include this file.) You will notice some changes in the Help interface but these are minimal and intuitive to those familiar with the previous (WinHelp) system.

The KeyCreator tutorial Help system also is HTML based with this release.

As explained under *New KXL Functions* above, KXL Help still uses WinHelp.

Printout of KeyCreator Help System Available

The KeyCreator Help system is available on the product DVD at the location Documentation/Printed_KeyCreatorHelp_Release80.pdf. This approximately one-thousand page document was generated from the Help system and then converted to a PDF. It is provided for your convenience. However, due to the size, no attempt was made to change the Help system to accommodate perfectly this printed document. Thus, there are some imperfections in this document, including occasional formatting issues and references to links that apply only in Help.

MISCELLANEOUS

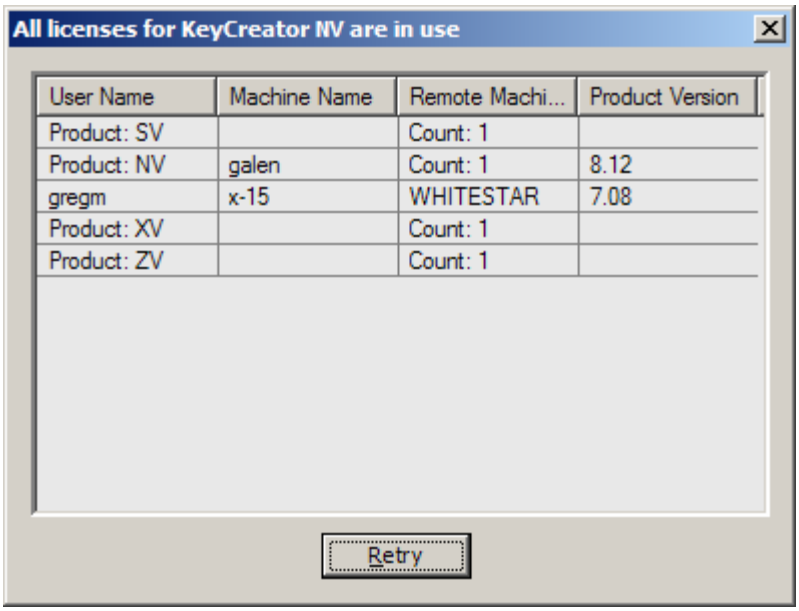
Supported Translators

The table below identifies the translator versions that KeyCreator 8.0 supports. The term "to" means "up to and including." (Bold under a column heading indicates a change for this release: so **this release makes no change**.)

File Format	Direction	Supported File Versions	Imports/Exports as Assembly File?
ACIS	Import	1.5 to R18	No
	Export		No
CATIA V4	Import	4.1.9 to 4.2.4	No
	Export		No
CATIA V5	Import	V5 R2 to V5 R18	Yes
	Export	V5 R6 to R18	Yes
DWG/DXF	Import	All Versions to 2007	No
	Export	R12 to R14, 2000, 2002, 2004 to 2006	No
IGES	Import	All Versions to 5.3	Yes
	Export	5.3	Yes
AutoDesk Inventor	Import Only	Part Files: 6 to 11, 2008	N/A
		Assembly Files: 11, 2008	Yes
Parasolid	Import	10.0 to 19.0.172	Yes
	Export	12.0 to 19.0.172	Yes
Pro/ENGINEER	Import Only	16 to 2001, Wildfire 1 to 3	Yes
SolidWorks	Import Only	96-2008	Yes
STEP	Import	AP203, AP214	Yes
	Export		Yes
Unigraphics	Import Only	11 to 18, NX1 to NX5	Yes
CADL 19	Export Only	Up to CADKEY 19	No

Server License Failed Dialog

With this release, if you try to use a KeyCreator floating license while all are being used, the **All licenses for KeyCreator are in use** dialog appears, like that shown below. As you can see, this dialog identifies who is using the licenses, their machine name and other related information, and provides a **Retry** button.



Support for Pro/E Assembly Cut Feature

Location: File>Import>Pro/E. With this release, flat translation and translation as internal or external references now support assembly cut features. These are features that are created in Pro/E at the assembly level, by subtracting tools from part references. This has no user-interface effect. (Previous releases either ignored the tools or brought them in as is without generating the features.)

Handling Selected Profile Entities Improved

Location: Create > Swept Solid > Extrude, **Create > Swept Solid > Revolve**, **Create > Swept Solid > Sweep**, **Create > Solid Feature > Boss**, **Create > Solid Feature > Pocket Create > Surfaces > Boundary Curves**. This release improves the handling of selected profile entities. Included in these improvements are

- Closed loops are now inferred from profiles
- The functions above (beside "Location") are more tolerant of gaps. Previous releases required that there be no gaps larger than 1.0e-06.

Name Function Moved to Entities Menu

Location: Edit>Entities>Name. This release moves the Name function from the **Tools>Composite** menu to the **Edit>Entities** menu. The topic describing the function in KeyCreator Help is now *Name*, instead of *Composite Name*.

Display Performance Improved

This release improves display performance. The table below shows an example of improved benchmark results for the Hardware OpenGL driver. (Actual values for your system could vary from those shown.) The term "fps" means frames per second. This refers to the number of times per second KeyCreator can rotate the view of the model and then perform a full update of the screen.

Item	Release 7.5	Release 8.0
10000 lines	127 fps	214 fps
10000 arcs	37 fps	192 fps
10000 ellipses	33 fps	192 fps
10000 polylines	101 fps	212 fps
10000 splines	25 fps	101 fps

Also, the following options have been added that improve display performance:

- **Edge/Curve Normal Deviation** on the **Units/Tolerances** pane of the **Properties** dialog. See *Options Added to Units/Tolerances Pane of Properties Dialog* on page 10 for a description.
- **Enabling Culling for Display**, **Enabling Culling for Rotation** and **Enable Text Greeking** on the **Display** pane of the **Set Configuration Options** dialog. See *Options Added to Display Pane of Set Configuration Options Dialog* on page 35 for descriptions.

Direct3D Driver Requires Software Install for Windows 2000 or XP

The **Driver Type** field on the **Display** pane of the **Set Configuration Options** dialog allows you to select the desired graphics driver. (**Tools>Options**, then click the **Display** node) As explained in KeyCreator Help, in the *Display* topic, one of these selections is Direct3D. To use the Direct3D driver with the Windows 2000 or Windows XP operating system, you must install the Direct3D End-User Run Time software. This software is available on the product DVD in the following directory: Release/DirectX. The file is named DXSETUP.exe. Double-click this file and follow the prompts to install it.

Unsaved Construction Plane Now CP=DV

With Release 8.0.1, if a part (`.ckd`) file is saved and closed, while the active Construction Plane (CPlane) is unsaved, the CPlane will be in CP=DV mode when the part is reopened. Previous to this release, the CPlane was in CP=1 mode when the part was reopened. For information on setting and saving a CPlane, see the *Set Construction Plane* topic in KeyCreator Help.

Extract-Profile CDE Added

This release adds the Extract-Profile function. The function creates a profile from selected entities, after trimming them automatically.

Starting Extract Profile

You can start Extract-Profile using either of the following procedures:

To Display the Extract-Profile Palette

1. Click **Add-Ins>CDE Open**. The **Select a CDE to Open** dialog appears.
2. Select qw_extractprofile`.cde` and click **Open**. The Qwins palette appears.
3. Click the **Extract Profiles from entities** button on the Qwins palette. Proceed to Using the Function below.

To Select from the Qwins Menu

Click Add-Ins>Qwins>Qwins>Extract Profile. Proceed to Using the Function below.

Using the Function

1. Specify active level.
2. Select the first entity.
3. Specify the start point on the first entity. (You can skip this by pressing the ENTER key.)
4. Select the second and the subsequent entities.
5. Specify the end point on the last entity. (You can skip this by pressing the ENTER key.)
6. If the result profile becomes OPEN, specify
   - Add a line to close, or
   - Leave it open

   If it is closed, this process is automatically skipped.
7. Specify the attributes.

Notes:

1. Selectable entities are
   - Line, Circle/Arc, Conic, Polyline, Spline, NURBS (open or closed)
   - Edges of solid or other entities are selectable
2. Only the intersections in 3D space are recognized
3. Line, Arc and Conic are automatically extended to create the intersection
4. When the entities are trimmed, the selected portion will remain
5. Backup is possible, with Undo of shape
6. If the command is aborted by ESC, all the intermediate results are cancelled
7. Specifying the ActiveLevel:
   - Value 0 means the current active level
   - Nested level can be specified (ex. 10.1.1)
   - Negative value brings ActiveLevel back to the previous one after finishing the command
8. Specifying Attributes:

- "Entity" button = Attributes are kept the same as the original entities selected
- "System" button = Attributes are the same as the current system Attributes setup

9. If the first and/or the last entities are "closed," the process of specifying the start and end points cannot be skipped
10. When this command is successfully finished, the extracted profile chain is kept as PreSelected, so that the commands, such as Extrude, can use this pre-selected profile. This PreSelection can be cancelled with the ESC key. This is effective only if "PreSelection" option is set ON.
11. Entities should be selected at the vicinity of the intersection point, and aimed at the portion to be kept. (If the selection position is not appropriate, unexpected trimming will occur.)
12. For a curved entity, which may intersect with the next element at multiple locations, the closest intersection to the selected position will be chosen as the intersection to the next.
13. This command can be Backed Up to the beginning, by pressing the F10 key several times.
14. While specifying the attributes, pressing the ESC key will abort the command and all the process of this command is undone. This command is completed when and only when you click either the **Entity** or **System** button.