

ZW3D What's New

V 2014



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ZW3D™ V2014 What's New

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Highlights of ZW3D 2014

Higher Customer Productivity, Better Usability and More Design Capabilities are the key words for ZW3D 2014. Many improvements to existing modules and innovative new functionalities are led by these 3 themes and by customer requests. Here are the top selective enhancements for ZW3D 2014:

Basics:	<u>New Pick Toolbar</u>
	<u>United Print/Plot with Enhanced Functionalities</u>
Import & Export:	<u>Upgraded Import Engine to Support New Format and Speed-up</u>
	<u>Upgraded Export Engine to Support More Formats</u>
	<u>Better DWG/DXF Export</u>
Modeling:	<u>Faster and Cleared History Manager</u>
	<u>More Robust Boolean Operation</u>
	<u>New Flex Tools</u>
	<u>New Part Configuration to Build a Part Family</u>
Assembly Design:	<u>Upgraded Assembly Tree</u>
	<u>New “Multi-insert” and Improved “Assembly Cut”</u>
Sheet Metal:	<u>Flange on Curved Edges</u>
Point Cloud:	<u>History-free for Point Cloud Operations</u>
Drafting:	<u>Faster and Better View Projection</u>
	<u>New Re-orientate View In-place and Drag-drop View Edit</u>
	<u>New Tolerance Grade and Grip Edit of Dimension</u>
	<u>Upgraded Table to Work like Excel</u>
CAM:	<u>Re-arranged Operation Parameter Form Based on Practical Workflow</u>
	<u>More Intelligent Region Priority Detection</u>
	<u>Optimized Link of Rest Roughing</u>
	<u>New Multi-step-down in Roughing</u>
	<u>Upgraded Minimum Tool Hang Length to be Accurate</u>
	<u>More Powerful Toolpath Editor</u>

Note: Important enhancements in this article are marked with★.

Basics

File Management

- **Smaller Z3 file saved by ZW3D 2014 than before.**
Several techniques are used to reduce the size of Z3 file. For example, display data is reduced to record essential information. Therefore opening a Z3 file saved by 2014 may be a little slower than before.
- **Dragging a file to ZW3D icon will open the file in ZW3D, and double-clicking a Z3 file can open it in a running ZW3D instance.**
- **New “Don’t list in root” option in “Part Attribute” and corresponding option “Show/Hide part not listed” in Root manager.**

UI

- **More On-screen floating toolbars**
Besides the original DA toolbar on the top of the graphic area, Left, Right and Bottom sides are now available to place customized on-screen toolbars. An auto-hidden drop-down menu is provided on upper-left corner. When the pointer stays over the corner, ▼ button for the menu will shows up. All provided toolbar can be customized on Customize form.
Layer list on DA toolbar supports multiple operations when it is expanded.

Datum

Upgraded Datum Plane

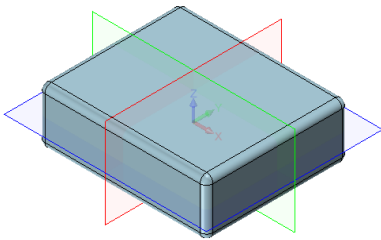
- **“3 Point Plane” can be finished after Origin field is defined with the other 2 fields optional.**
After defining the Origin field, the plane is oriented parallel with the default XY plane, which is similar with offsetting the XY plane to the point you picked. You can finish the command if you intend to leave the other 2 fields empty. Or you can keep picking points to define X point and Y point.
- **New “View Plane” to create a plane parallel with the screen plane.**
During definition you can rotate the view as you want, and the plane being created will be re-oriented to keep itself parallel with screen plane.
- **New “2 Entities Plane” to create a plane by making it through 2 selected entities.**
Point, Edge/Line, Curves, Planar face and Datum can be used to define the plane. For example:
 - A point and a plane will create a new plane through the point and parallel to the picked plane.
 - A line and a plane which is not coplanar and perpendicular with the line will create a new plane which is through the line and perpendicular to the picked plane.

- Picked curve can only make a new plane with a coplanar point, line, curve or parallel plane or face.

Auto Size Datum Plane

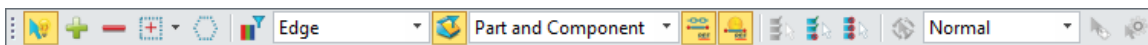
Datum planes will automatically adjust their own sizes basing on the bounding box of existing shapes to enclose all the shapes and facilitate observation and picking datum.

You can disable this feature through the option “Visualize manager > Datum > Auto Resize”, or the “Fix Size” option of the right-click menu on certain plane.



Picking

★New Pick Toolbar



Icons	Name	Description
	Normal Pick	Same picking behavior with last version.
	Add Pick	New picking mode to make new selection to add into existing pick list.
	Del Pick	New picking mode to make new selection to subtract from existing pick list.
	Window Pick	Window pick command set to specify which pick behavior you will like to use for next window pick behavior.
	Poly Pick	Poly pick to do multiple picks.
	Attribute Filter	Layer, feature, and color are available to set to filter the selection of

		entities. Entities with the attributes defined in this form will be either included or excluded from a pick selection. The filter will only remain active while the form is displayed.
	Filter List	Provide available entity types to filter and to help pick specified target.
	“Direct-Edit” Toggle	By this, you can specify whether the features are the first choice of the selection, or the common geometries like edge, face, shape are the first choice instead of features.
Part and Component	Extended Pick	To control whether the geometries of the active part’s components are valid for selection, or just the ones of the active part are up to the selection. If it’s set to Part and Component, points, edges, faces and shapes from components can be selected directly as the input for current command.
	Associative Copy	Only available when the above “Part and Component” is set. It’s to control whether the reference geometry generated from other components should keep association with its parent entity. It works just like the same option on “Reference Geometry”.
	Record State	Only available when the above “Part and Component” is set and “Associative Copy” is on. It’s to control whether the reference geometry operation should record the feature state of its parent object. It works just like the same option on “Reference Geometry”.
	Pick All	Pick all available objects for current input field.
	Unpick Last	Cancel the last selection.
	Unpick All	Cancel all selections of current input field.
	Invert Pick	Unpick all objects of the current selection and basing on the current setting of Filter List, pick the rest objects of the current selection.
	Smart Point Ref Toggle	It works the same with the one from right-click menu and is controlled by “Configuration > Part > General > Ref Point Timer” option. When it’s on, after the pointer stay over a point or edge, an offset coordinate frame on the point or a 2 dimension reference on the edge will be active to locate the new point.

Normal	Pick Rules List	The provided pick behaviors of point, edge and face are the same with the ones from right-click menu, like Pick Faces.
	Regenerate Pick	It controls whether the pick path is recorded or just the pick result is recorded. If pick path is recorded, during each regeneration, the pick result will be reevaluated basing the pick seed and its pick rule. It works the same with the same option on Pick Curves or Pick Face form.
	Custom-Pick Settings	To set how the Custom Pick of Pick Rules List works.

❖ Where to Find

Right-click Menu (on blank area of Ribbon) > Toolbars > Pick Toolbar

Right-click Menu (on blank area of Ribbon) > Customized > Transfer

New Face Pick by Its Relations

The following relationships for selecting faces based on a seed face are supported:

- **Tangent:** All tangent connected faces will be selected
- **Coplanar:** All planar faces that are within the same plane with the seed face will be selected.
- **Coaxial:** All cylindrical, conical faces whose axes are coaxial with the axial of seed face will be selected.
- **Coplanar Axis:** All cylindrical, conical faces whose axes are coplanar with the axial of seed face will be selected.
- **Equal Radius:** All cylindrical, spherical faces whose radiuses are equal with the one of seed face will be selected.
- **Symmetric:** All planar faces, cylindrical faces whose are symmetric with the seed face basing on specified datum will be selected.

❖ Where to Find

Right-click Menu (on blank graphic area when Pick Filter sets to Face) > Pick Faces

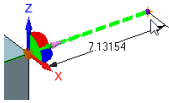
Pick Toolbar

★Upgraded “Smart Point Ref” with a Triad Frame

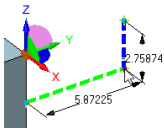
A new triad frame is added into smart point ref to give a better and easier control to locate a new point from

the ref point.

- The triad frame will appear on a point after the pointer stay over the point over the time set on “Configuration > Part > Ref Point Timer” option.
- Picking one of the 3 axes of the triad frame can locate the new point by offsetting a distance along the axis from the ref point.



- Picking one of the 3 planes of the triad frame can locate the new point by offsetting 2 distances along the 2 dimensions of the planes from the ref point.



- Only one axis or one plane of the triad frame can be picked each time. Re-picking the same axis or plane, or picking another one will de-select it.

❖ Where to Find

Configuration > Part > Ref Point Timer

Pick Tools Bar > Smart Point Ref

Text

➤ New stack property for Text

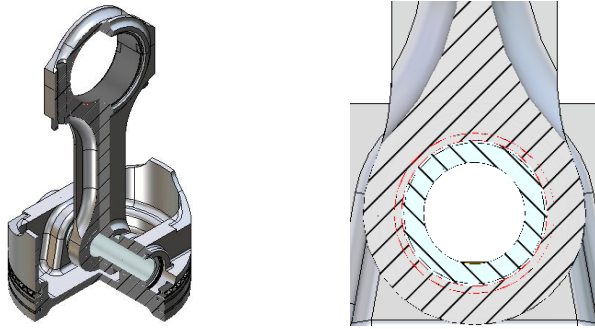
Stack property starts with VxSTK and provides 3 forms with ^, / and # symbols to distinguish.

[VxSTKAAA^BBB]	[VxSTKAAA/BBB]	[VxSTKAAA#BBB]
$\begin{matrix} A & A & A \\ B & B & B \end{matrix}$	$\frac{A \ A \ A}{B \ B \ B}$	$A \ A \ A / B \ B \ B$

Inquiry

- ★New “Section with Profile” and other improvements in “View Dynamic Section” command.
 - New “Section with Profile” mode to support step section, 1/4 and 3/4 section.
 - Different hatches are used for different components.
 - New “Hatch color” option to customize section color, or to use the components’ own color.

- New “Display hatch for non-closed shapes” option.



- New “Show/Hide” buttons in “Drafting Checking” form of “Analyze Faces” to control which faces are displayed.
- Improved “Draft Checking” in “Analyze Faces” to use the tolerance set in Part Settings.
- “3D Surface” Inquire command supports multiple surfaces.
- New “Projection” in “Distance” Inquire command supports projecting the resultant distance in specified direction.

★Print/Plot

- **Unify print and plot settings into one form**

Due to this unification, all functionalities which belonged to Plot will be supported by Print, such as Color, Line type, Line width.

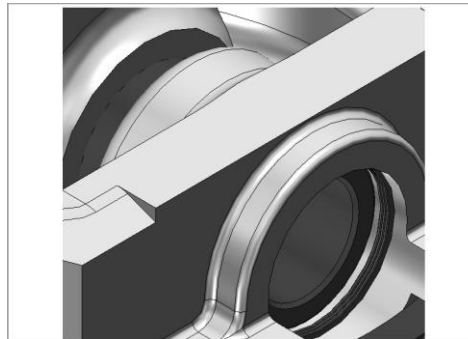
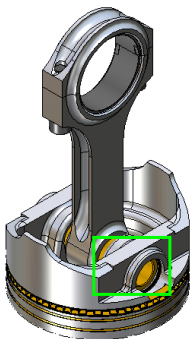
- **New layout preview**

The arrangement of paper size and margins can be viewed through the layout preview window.

- **New black and white print**

Found in Print Setup > Advanced > Color, this option works for wireframe objects.

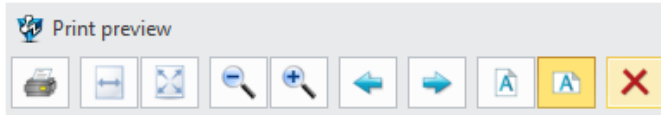
- **Pick specified area to print**



- **Batch print drafting**

3 options to batch print are provided: All sheets, Current sheet and Specified sheet.

- **New stamp setting form to customize your own page header**
 - New “User String” input to customize your own header.
 - New “Position” option to arrange where to put the header.
- **Upgraded print preview to provide more functions**



Troubleshooting

- **New input validity message box on form**

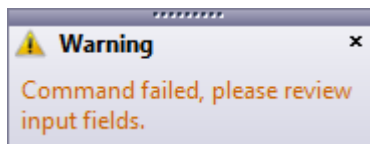
When you input some data into a field on a command form, the validity will be checked automatically. If invalid data is detected, the message box will pop up to give you notice.



- **New pop-up message box**

The error messages box pops up on the lower-right corner of the screen, and will stack up one by one if different types of errors occur. The box will disappear automatically after a few seconds.

When this kind of message box appears, there is something important that you need to handle. For example, when the inputs of a command are inadequate but still be executed, an error message box will pop up to warn you that the command failed and you need to review your input. You can directly change the input since the form is not taken down, and the message box will disappear automatically while you're editing the input.



- **Classified messages by colors in output window**

Error: System and application level errors which are severe and caused by undefined behaviors like file corruption, crash, etc.

Alert: Command level errors for which user operations are expected to solve the issue.

Warning: An unexpected result for the last operation occurred, and it's up to users to decide to what to do with it.

Note: Notification of success for important operations like regeneration, constraint solving, import, export, etc.

Info: Trivial message that indicates what has been done or what's going on.

Message: General output message.

Import/Export

★Upgraded Import/Export Engine

Upgraded Import Engine to Support New Format and Speedup

➤ **Faster 3rd formats import and new formats support**

Due to this upgrade, the following formats are improved. Solid Edge is the new supported format.

Formats	Extension	Version
ACIS	.sat, .sab, .asat, .asab	R1 – R24
CATIA V4	.model, .exp,	4.1.9 – 4.2.4
CATIA V5	.CATPart, .CATProduct,	R6 – R23 (V5 – 6R2013)
Inventor	.ipt (V6 – V2013), .iam (V11 – V2013)	V6 – V2013
NX	.prt	11 – NX 8.5
Creo (Pro/E)	.prt, .prt.*, .asm, .asm.*	16 – Creo 2.0
SolidWorks	.sldprt, .sldasm	98 – 2013
Solid Edge	.par, .asm, .psm	V18 – ST5

- **Points supported in STEP import.**
- **Improved IGES import to better handle planar faces.**
- **Speedup the import of 3D data in DWG/DXF.**
- **Screen the Ray and Xline objects for DWG/DXF import.**

Upgraded Export to Support More Formats and Provide Better Quality

- **New support to export CATIA V4 and V5.**
- **New support to export SAT binary format .sab.**

Formats	Extension	Version
---------	-----------	---------

IGES	.igs, .iges	Version 5.3
STEP	.stp, .step	STEP AP203, AP214(Only Geometry)。
Parasolid	.x_t, .x_b	7.0 ~ 25.0
VDA	.vda	VDA-FS 1.0 & 2.0
3D PDF	.pdf	1.7
ACIS	.sat, .sab	11 ~
Catia V4	.model	CATIA 4.1.9 ~ 4.2.4
Catia V5	.CATPart, .CATProduct	R6 to R21

- **Primitive objects are supported to export as primitives for STEP, Parasolid, and ACIS.**
- **The size of the exported image is read out basing on its source. And new Ratio button is added to maintain the proportion of image's Height and Width.**

For example, if export image area is defined by window pick, the read-out size is the same with pixel rang of the picked area.

Simplified Import/Export Setting Options

- **Attribute filters are removed for all formats except DWG.**
- **Simplified the import/export settings of all formats.**
- **Reduced import history options to one.**

Upgraded DWG Export

- **Reduced export options.**
 - Coordinate options are deleted, and export is implemented based on current active frame by default.
 - No color replacement options.
- **New “Merge Arc” option to merge concentric arcs which are within tolerance.**
- **Drafting is able to export into layout space of a DWG/DXF file, besides to model space, and all sheets of a drafting can be exported together into the layout space of one DWG/DXF file.**
- **Views are exported into blocks.**
- **Eliminated duplicated lines exported from hatches.**
- **Improved text export.**

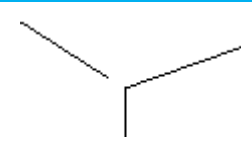
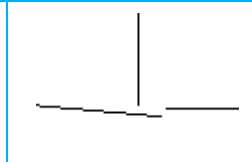
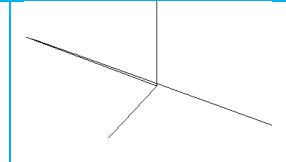
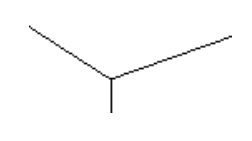
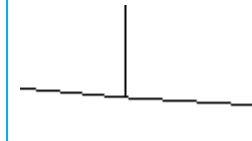
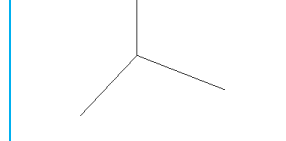
- The value of a variable gets exported, not the variable itself.
 - Made sure that text sticks to its original position and no offset occurs.
 - Texts used in fonts whose names start with ZW3D are exported as DWG MText objects.
 - TTF texts are exported as DWG MText objects in same TTF.
- **Improved dimension export to make sure that exported dimension texts are in the right place and attributes of exported dimensions are correct, such as the arrow and the decimal places.**
 - **Tables are exported as corresponding DWG table object.**
 - **Improved entity attributes export, such as color, line type, line width and layer.**

Heal

New “Check Corners” to Analyze Defective Corner

“Check Corners” analyzes all corners of the model, and then lists the result on a form. Picking one item from the list can highlight the corresponding corner in graphic area. “Zoom To” button can zoom in to highlighted corner. “Heal one” sends the picked item from the list to “Heal Corners”, and “Heal All” tries to fix all listed corners in one time.

Samples of defective corners and fixed corners:

Defective Corners				
Fixed Corners				

New “Heal Corners” to Fix Corners

3 different healing modes are provided by “Heal Corners”:

- **Heal Shapes** -> heals the inconsistency between the coordinates of objective corner vertex and the end points of its adjacent edges.
- **Merge Vertices** -> drags and merge the vertices of open or self-intersected loops to a basic corner vertex.
- **Heal Corners** -> the combination of Heal Shapes and Merge Vertices.

CAD

Sketcher

Improved Auto Constraint System

- “Points Horizontal” and “Points Vertical” constraints generated by auto constraint during drawing are restored to add into the entities being-created.
- Holding down Shift key can stop auto constraint from adding any new constraints into the entities being-created.

During drawing, auto constraint will provide you available constraints it can find. If you don’t want to add these constraints into entities you are creating but still want to snap to these positions, just hold down the Shift key.

- Clicking the Shift key can navigate current available constraints provided by auto constraint to choose which one to add into the entities being-created.

Improved “Image” to Support Multiple Pictures

- Support multiple background pictures.
- Background pictures can be picked to Blank/Unblank, or Delete.
- Background pictures inserted in a sketch can be seen from outside the sketch.
- 2 point to dragging to adjust picture size is much easier than before.
- New “Angle” option helps you rotate the picture.
- New “Flip” option to change the picture by mirror with X or Y direction.



Upgraded “Reference” to Support Timestamp

Now the sketch “Reference” command will record the timestamp of the referred geometry to allow ZW3D to selectively determine whether the reference geometry needs to be re-evaluated during regen (reference part or transform has changed) and whether the reference part needs to be rolled back to the state at which the reference geometry was recorded.

The sketch Ref-Geom option forms were upgraded to show the reference part and reference feature corresponding to the selected reference geometry.

Others

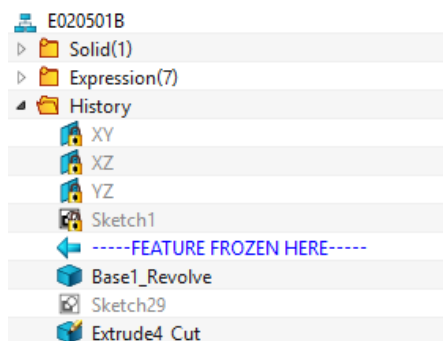
- New “Flip the direction” option is added to change the offset direction.
- Auto hide the sketch dimensions when exit the sketch.

Modeling

★History Manager

Feature Freeze

New “Feature Freeze” is to reduce the history regeneration time by freezing features to exclude from replay. After it is activated through the option “Configuration > Part > Use feature frozen bar”, a “FEATURE FROZEN HERE” bar will appear in history tree. You drag the bar up and down around the tree to freeze the features you want.



- Each replay will start from the Frozen Bar.
- Froze features cannot be redefined, deleted or cut.
- Freeze Bar provides a right-click menu with the following 3 options:
Go to Start, Go to End, Regen with Freeze Feature.
- Frozen state will be recoded in Part Configuration.

❖ Where to Find

Configuration > Part > Use feature frozen bar

Simplified History Tree by Removing Attribute-related Operations

Attribute-related operations are the following:

- Show/Blank operations, like blanking an entity.
- Attribute operations, such as color, size, name tag, density etc.
- Layer related operations.
- Texture related operations.

Regeneration will be faster because there are fewer things to replay. If an old file saved before 2014 is open and regenerated in 2014, relevant attribute operations will be removed.

Since texture operations are no longer logged into the history, a new “Edit Texture” command is provided to work as a tool to redefine the assigned texture to the shape.

History-free for Point Cloud Operations

Commands to import and edit point clouds and STL mesh are no longer logged in the history. Point and STL blocks are treated as explicit geometry, and are unaffected by history regeneration. Point picks on explicit point/STL blocks are stored in the history as absolute coordinates with no associativity to the explicit geometry. Surfaces and curves created from a point/STL block are stored as “static” geometry in the history without associativity to the point/STL block. That is, if the point/STL block from which geometry is created is deleted or modified, the geometry will not regenerate.

A big speedup on history regeneration will be easily obtained on a model involving point/STL objects.

Non-historical for Variables and Equation Set

All variables and equation sets will only be listed under the Expression node on the Design Manager, and they will not participate in the replay. After changing the values of variable or equation set, all the variables and equation sets under Expression node will be reevaluated, but the history will not replay. Users need to replay the history manually.

Non-historical for “Inert a Component” Command

The “Insert Component” operation is no longer recorded in history. Old files created before 2013 version still work the old way, that is, the “Insert Component” operation is still recorded in history. If you prefer to continue editing the file in 2014 style, you can encapsulate its history to update the file. Files created by 2013 version with new assembly mode will be updated to 2014 if they are regenerated in 2014.

Smoother and Robust History Replay

- Enhanced persistent labeling to encode broader info to help ensure subsequent features can find their target shape when it changes a lot during history redefinition.
- Improved persistent labeling match to allow partial match between a face label and a label derived from it for more reliable history replay.

Save display data with backup states

New "Save display data with backup states" option is added into the "Part" tab of the Configuration menu.

When the setting is enabled, display data is saved with backup states inserted by the user. Otherwise, display data is not saved and is regenerated each time the backup state is restored. Saving display data with backup states speeds up restoration of a model state but increases file size.

Others

- **Updated the feature icons displayed in the history tree to use the icons used to identify features in menus.**

Wireframe Design

New Triad Frame to Help Point Redefinition

When redefining a point, a triad frame will appear on the original position of the point. Same functionalities mentioned on section [“Upgraded Smart Point Ref with a Triad Frame”](#) are provided.

❖ Where to Find

All commands that have point input field, such as Wireframe > Line.

❖ How to Use

After defining 5 points for “Points” field of “Through Point Curve” command, move the pointer over one of the 5 points. And then after the triad frame shows up, pick one axis of the triad frame and move along the pointer. Pick one plane of the triad frame to de-select and move around the pointer to locate a new point by offsetting 2 dimensions from the original point.

New “Point on Align Plane” to Help Point Picking

After “Align Plane” option is defined, the “Point on Align Plane” is available to activate. Under this option, all picked point are limited onto the defined align plane.

❖ Where to Find

All wireframe commands that have “Align Plane” field, such as Wireframe > Line.

Part Design

★More Robust and Updated Boolean

Several improvements for Boolean Operations have been implemented through analysis of complex cases reported by end users.

A new “BooleanShapes” field specifies which shapes can be involved in a Boolean operation. If no shape is

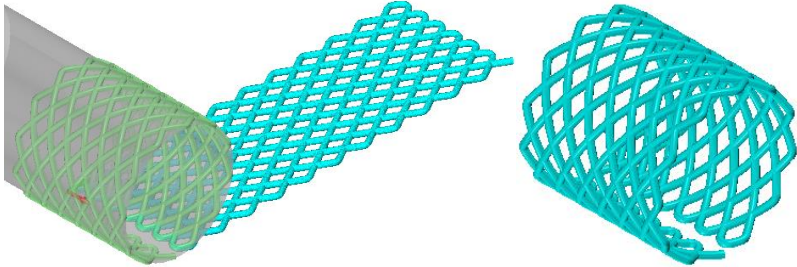
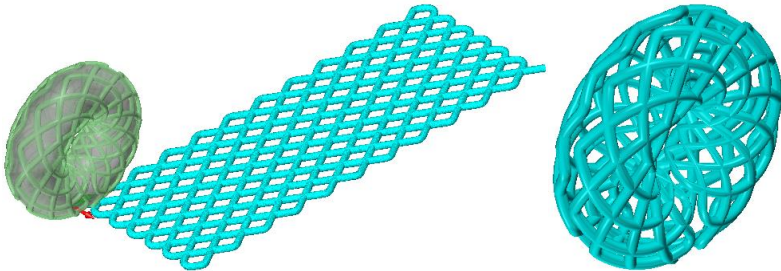
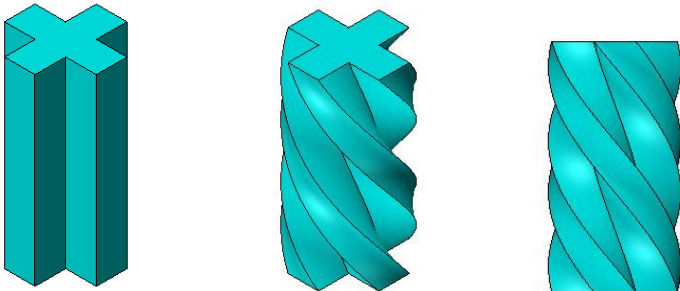
specified by the user, the one that is actually involved will be automatically registered in the field on the command's first execution. Only the shapes recorded in the "BooleanShapes" field are considered when the Boolean operation is regenerated.

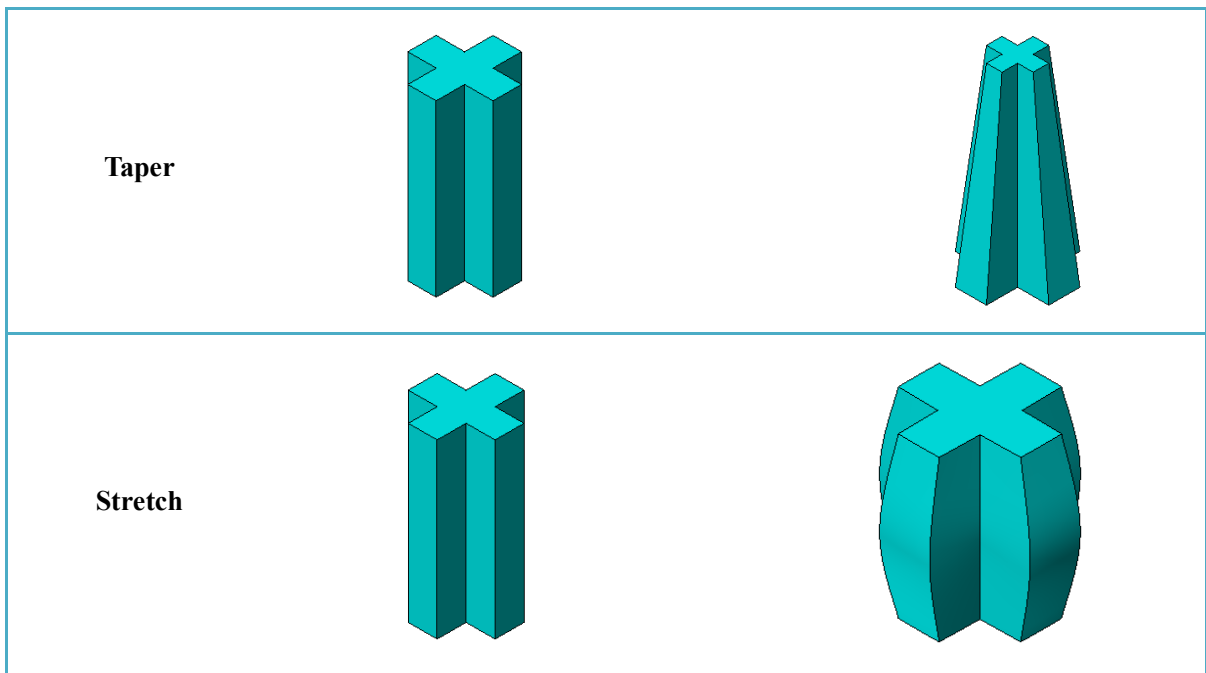
❖ Where to Find

All commands that provide Boolean operation, such as Block, Extrude, Revolve and so on.

★New Flex Tools

5 new Flex tools are provided as below.

Tools	Samples
Cylindrical Bend	
Toroidal Bend	
Twist	

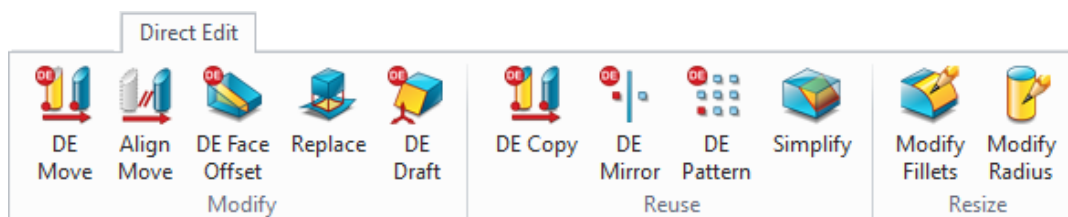


❖ Where to Find

Shape Ribbon > Flex

Direct Edit

➤ Specified Tools for Direct Edit



A new ribbon tab is added to gather all Direct Edit commands into one location. The workflow of these commands is the same with the old commands, except “face” is the sole entity that these commands will deal with.

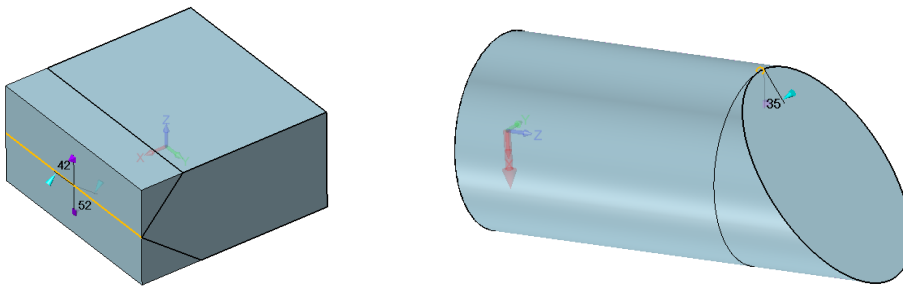
➤ New “Modify Radius” to change the radius of cylindrical or spherical faces

New “Join Face”

Co-planar faces, co-cylindrical faces, faces having same geometry data and faces lying on the same trimmed surface can be merged together automatically.

New “Asymmetrical Draft”

Asymmetrical Draft is the update version of “Draft” but without Face draft. It can add 2 different angles to a neutral edge and different angle sets to different edges with the use of the new set list widget. It also supports to draft a face around a point.



★New Part Configuration to Build a Part Family

- Features can be configured to suppress, and their dimensions can be configured to different values.
- Feature frozen state can be set differently for each configuration.
- All part configurations can be activated, copied or deleted.
- A new “Part Configuration” form is provided.
- Different part configurations can be set in different views of the same part in Drafting.
- One of the part configurations of an inserted component can be picked to activate, but only for one active session.

❖ Where to Find

Configuration > Part > Folders display settings > Configuration

History tree > Part Config node

Right-click menu (on blank area of history manager) > Create Config

Right-click menu (on a feature operation) > Configure Feature

Right-click menu (on blank area of graphic area) > Config Table

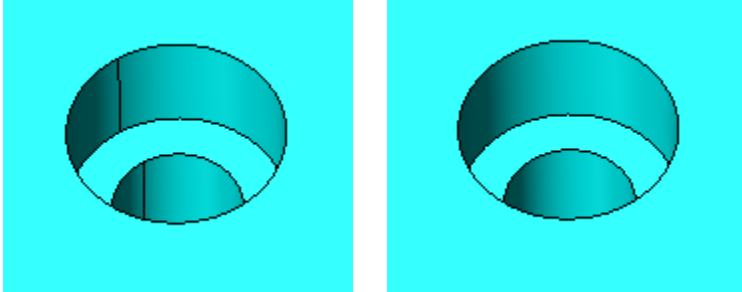
Right-click menu (on dimension within graphic area) > Config Dimension

Wireframe Entities of a Sub-part are Available in Its Parent Part

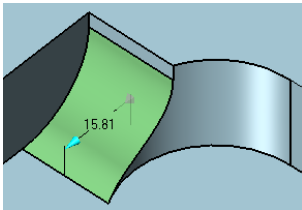
Wireframe entities of a sub-part are available in its parent part, and work just the same as the one created in its parent part.

Others

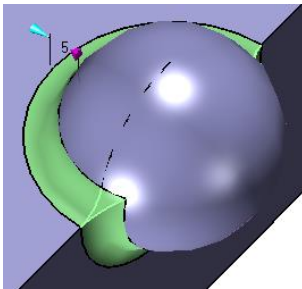
- “Flag Hole Feature” supports merging of faces into a cylindrical face when they are flagged as one hole, if they are recognized as one of supported hole types.



- Improved extension of cylindrical faces for Filleting



- Improved Fillet on edges interfering with the seam edge of a sphere.



- “Delete Loop” will only edit the copy of its selected face if the face is seen to other faces.
- Improved “Wrap” on closed source surface to better wrap shape around the seam of closed shape.
- New “Create sub-part in destination part” option in “Geom to Part” command.
- New “Copy” option in “Scale” command.
- Make sure the preview of Extrude is the same with its result.
- Profile self-intersection check is added into “Extrude” command. If self-intersection is found, the command will fail. Users can specify regions of the self-intersected profile to extrude.
- New “Direction” field in “Inlay” command.
- “Extend Face” supports variable distance extension.

Assembly Design

Updated Assembly Manager

➤ **No History for “Insert a Component”**

Inserting a component into an assembly will not be logged into the assembly’s history tree any more in version 2014 and beyond. All inserted components are explicit and excluded from the regeneration of the parent assembly’s history, except, if they contain a time-stamped “Reference Geometry” operation or an “ImportGeom” operation that references the parent assembly, in which case they will be regenerated at the end of regeneration of their parent assembly.

➤ **New “Suppress Component” to temporarily erase the inserted component**

2 different modes of suppression are provided:

- **Normal Suppress** -> simply suppress the picked component.
- **Conditional Suppress** -> suppression state is basing on evaluation of an expression.

If an inserted component cannot be found when it parent assembly is opened, it will be suppressed automatically. It can be un-suppressed by adding its parent folder to “File/Search Paths” or manually selecting its new location with the File Browser.

New “Multi-insert” Command for Batch Insert

- **Support for inserting different parts from different files at the same time.**
- **“Duplicate” option to control inserted instances for each part.**
- **“Scatter” option to spread inserted instances in a linear pattern way for better observation.**

Upgraded “Component Pattern”

➤ **New assembly tree node for “Component Pattern”**

Components generated from a Pattern will be listed under a node of the assembly tree. Component pattern operations still log into the history, their redefinition or deletion from the assembly tree will trigger the regeneration of the parent assembly’s history. Likewise, their redefinition from the parent assembly’s history will trigger an update of pattern node in assembly tree.

➤ **New “Instanced as component” option**

The option dismisses all the associative pattern relationship between pattern instances and makes the pattern work like a multi-insert operation. Therefore pattern operations with this option are not logged into history, which means they cannot be redefined with associativity.

Improved Assembly Cut

The improved “Assembly Cut” command uses associative “Reference Geometry” to reference cutting shapes into the history of each component that is cut with corresponding Boolean operations. Parts instanced more than once within an assembly are duplicated before they are cut.

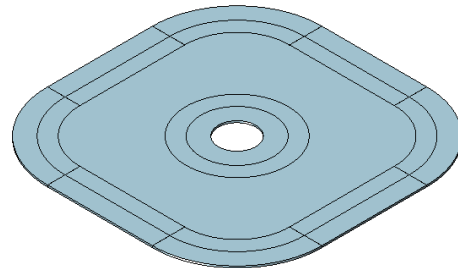
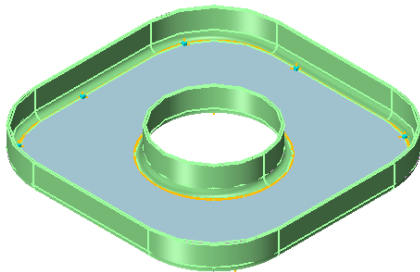
New “Copy multi-instanced part before cutting it” option is to control whether copies of the multi-instanced part should be generated first before the cutting. If the option is unchecked, only the first cut is going to perform, and the rest intersections will be ignored.

Naming the copies is changed to PartName[Copy].X. And the “Derive from” attributes of the copies are set to their original parts, which make them to be recognized as their original parts.

Sheet Metal

★Flange on Curved Edges

Now Full or Partial Flanges can be added to the curved edges directly. Flanges on curved edges can be unfolded.



Point Cloud

★History-Free for Point Cloud Operations

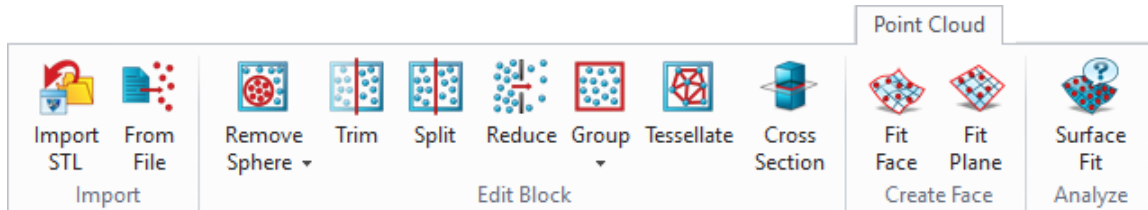
Commands to import and edit point clouds and STL mesh are no longer logged into the history. Point and STL blocks are treated as explicit geometry, and are unaffected by history regeneration. Point picks on explicit point/STL blocks are stored in the history as absolute coordinates with no associativity to the explicit geometry. Surfaces and curves created from a point/STL block will be added into history as static geometry which cannot be redefined.

Point-cloud/STL import/edit commands function in the same historical way for pre-version 2014 parts. New parts made in version 2014 or later are tagged to use the new history-free point-cloud/STL-mesh management.

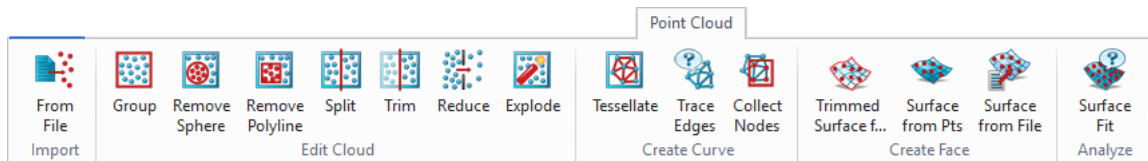
New and Adjusted Point Cloud Commands

➤ Ribbon adjustment

2014 Ribbon

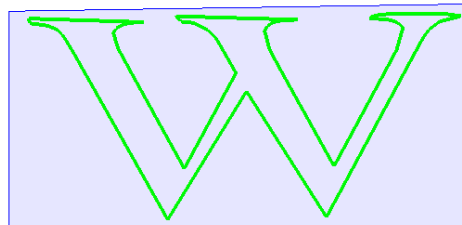
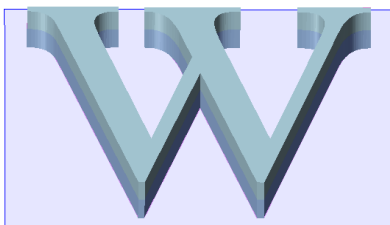


2013 Ribbon



- New “Import STL” command to import STL file directly
- Radius of “Remove Sphere” is now required input and only one sphere can be defined for each operation.
- “Trim” and “Split” commands can operate on an STL block directly.
- ★New “Fit Face” command is the combination of “Surface from Points” and “Trimmed Surface from Points” commands.
- ★New “Cross Section” command generates the intersection curves of a triangle mesh block and a plane.

Lines, arcs and free curves will be used to fit the cross-section curves. After extracting the outlines of the STL model, you can build the NURBS model using common modeling tools like Extrude, Loft, Sweep, etc.



- New “Fit Plane” command generates a planar face by auto fitting detected triangular meshes which are around the picked point and within tolerance.



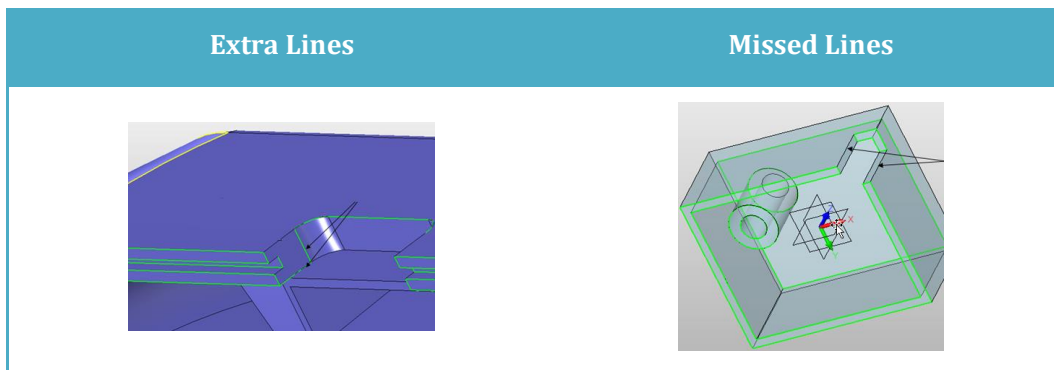
- “Trace Edges”, “Collect Nodes” and “Surface from File” commands are removed from the Ribbon.
- *.pts file format is supported by “From File” command.

ZWMold

Parting Design Improvements

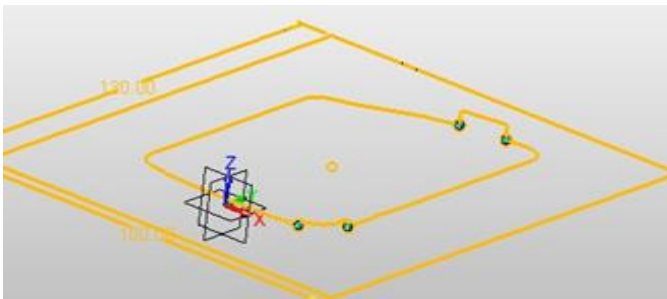
➤ Parting Line Improvements

Improved Part Line to settle the extra and missed parting lines.

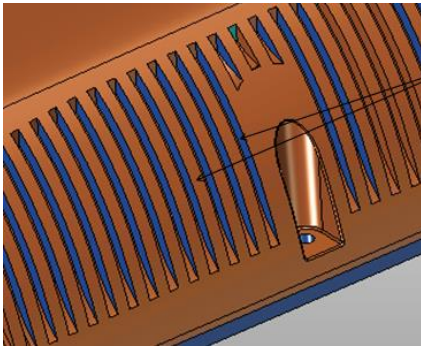


➤ Customized Parting Face Creation for Each Parting Line'

Each parting line can be manually split into several segments, and how the parting face grow for each parting line can be controlled by users to the way they think is appropriate.



➤ Improved the accuracy of regional analysis



Mold Project's Improvements

➤ Improved Project's Process

New option is added to specify whether a mold project is single or multi cavity.

No combination operation is needed for a single cavity.

The process of inserting product to layout command is adjusted.

➤ Improved Layout Command

New datum is added to control the product's location, and to support multi cavity design.

➤ Improved Stock Command

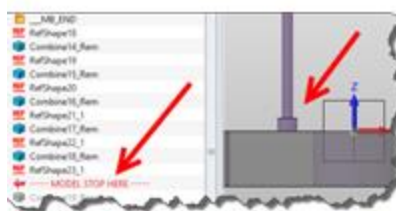
Mold Library Improvements

➤ Simplified the Number of Standards Listed in Object List

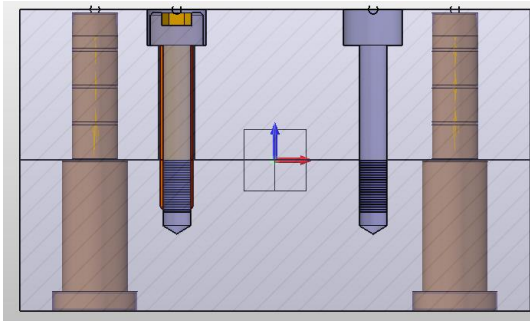
“Do not list in Root” option is added in “Part Attribute” panel to control whether this part is shown in the object manager. Corresponding “Show/Hide part not listed” button is provided in root manager form to show or hide these parts.

➤ Added Standards Parts of Rabourdin and Mesburger

➤ Improved Intersection Judgement of Critical Face



➤ Fixed PB's Value Becoming Zero while Modifying a Standard Part.



- **Library Commands Supports Middle-repeat Operation.**

Drafting

★Updated Style Manager

- **New Local Style Support**

A local style file with ANSI, ISO, DIN, JIS, GB style standards is provided within user folder which allows users to save their pre-defined styles for re-use. The old behavior, in which styles are saved in the Z3 file, is adjusted to save styles only used by its parent Z3 file. Styles saved with the Z3 file can be saved into the local style file or replaced by the one from the local style file.

- **New Default Local Style Standard Edit and Original Style Standard**

5 default local style standards provided in the user folder are editable. For resetting all the changes of style standards done by users, an original style standard file is provided in the installation folder and can't be edited. All styles from local or used in current document can be replaced by the original one. A new style standard can be created based on the active standard or the original one.

- **New Active Style State**

The active style is the one you are using currently. By default, the style used by active style standard is the active style. If you prefer another style to use for next operation, you can activate it by “Activate” button. If a style is set as active style, its name turns blue and attached with “Active” keyword.

- **New Style Types**

The following new style types are added:

Point, Line, Text, Hatch (all these are not supported by style standard), Table, View.

All styles defined for Point, Line, Text, and Hatch only work on newly created entities.

- **New Nodes for Defined Styles of Each Style Type**

- *All available defined styles are listed under their type nodes, and have following statuses:*
 - If it is used by active style standard, its name is bolded.

- If it is set to work as active style, its name is blue and tagged with “Active” keyword.
- If it has been modified but not saved, its name is italicized and tagged with a star.
- **Right-click Menu are supported for nodes with the following options:**
 - New
 - Rename
 - Delete
 - Replace
 - Export
 - Save to Local
 - Save to Document
 - Update from Local
 - Delete from Local
- **New Filter Function in Style Tree**

★Updated View

Better and Faster View Projection

- **More Robust in Hidden Line Projection**
- **50% Speedup in View Projection.** More cores the system installs, faster views are generated
- **Better Hatch in Section View by Improving “Dynamic Hatch Scaling and Angle” Option**
- **Improved Thread Hole Curves Trimming**



View Attributes

- **New View Style in Style Manager**
 - A Views node is added in the style manager with the following sub-node:
 - Base, Projection, Auxiliary, Section, Detail, Break Line**
 - Attributes you can set in view style include layer, label, scale, arrow and text.
 - Style standard includes view style.
- **More Attributes in View Creation Form**

- Style list, label, scale and text are added in “Standard”, “Projection”, “Auxiliary” and “Detail” command form.
- A component list is provided to choose which component not to section in Section view form
- New attributes for break lines in Break Line view.

View Creation

➤ New Coordinate Option in Layout and Standard View to Support Customized Project Frame

View Advanced Setting > Coordinate

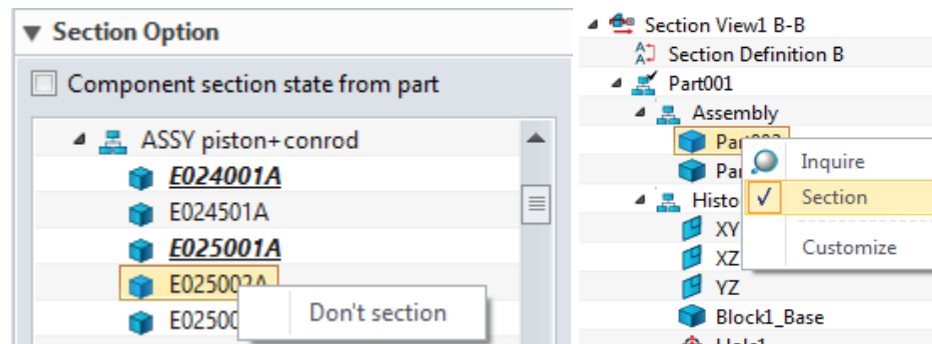
➤ Upgraded Section View

- ***Break down into “Full Section”, “Aligned Section”, “3D Named” and “Bent Section”***

Revolved type is merged into Full Section. Leaving Location field empty can create a revolved section.

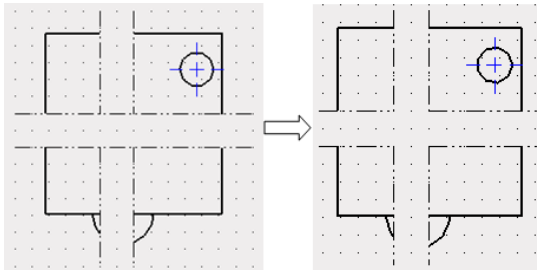
- ***Simplified Section Point Input for Full Section and Aligned Section***
- ***New Support to Customize Which Component to be Section***

Simply picking a component from the assembly tree listed on Section view form specifies which one is not sectioned. Or you can do the same job through the right-click menu on a component of an assembly tree under section view node of the sheet manager if you have unchecked the option “Component section state from part” on view attribute.



➤ Improved Break View to Provide More Modes

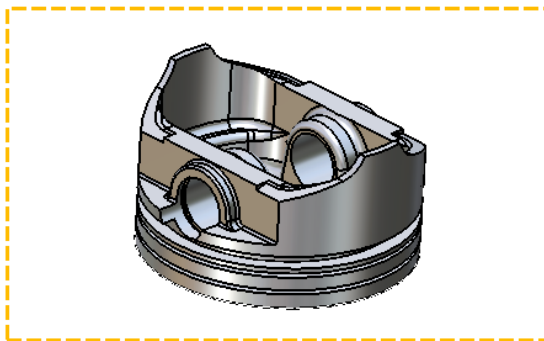
- ***Two new modes: Horizontal and Vertical***
- ***Auto-adjust the length of break line basing on view's bounding box***
- ***Removed the intersection of multiple break line***



- *Make sure that section view can be created from a break view with or without zigzags.*

View Edit

➤ New “Rotate View” to Re-orientate View In-place



- Only available for base view.
- Defined views in part are provided from view list.
- Specific rotation angle can be given to rotate the view basing on current view plane.
- Rotation can be done by right button of mouse if “Rotate” button is pressed down.
- Pan can be done by middle button of mouse if “Move” button is pressed down.
- Zooming is supported.

❖ Where to Find

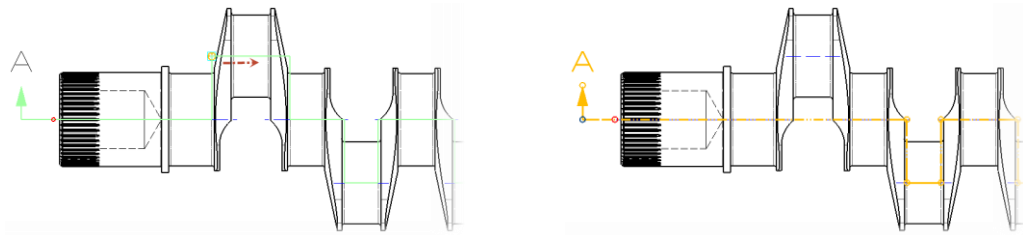
Drafting > Layout Ribbon > Rotate View

➤ New “Alignment” to Break the View Projection Association

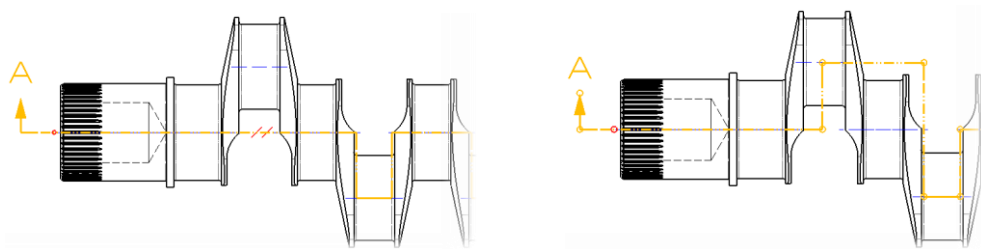
Uncheck the Alignment from right-click menu of a projected view to break, and re-check it and move the view if you want to resume.

➤ Direct Edit View

- **Section View - Drag-drop section line can redefine the section plane location**
 - Base point will be highlighted after section line is picked and can be dragged to change.
 - Corner point can be picked to change its position.
 - Segments of section line can be dragged to move around and will be merged into one if they encounter one another, or deleted if they are from a zig step and dragged to cross each other.



- Projection arrow can be dragged to adjust its position.
- Projection direction can be changed by dragging the tail of the arrow or using “Reversion Direction” from right-click menu.
- New step can be added by “Insert Step” from right-click menu.



- **Detail View - Drag-drop boundary lines can redefine its zooming area**
 - The circle periphery can be dragged to change the extent of detail view while center point is to change the view location.
 - The corner point of rectangle is to change the extent of detail view while the edges are to relocate the entire detailed area.
 - Behavior for dragging poly line is similar to rectangle.
- **Reverse the projection arrow by “Reverse Direction” from right-click menu**
- **Sub objects of a view can’t be deleted, moved, copied and cut, but they can be blanked or changed their attributes.**
- **After standard view is deleted, its remaining projection view will be converted as standard view.**
- **View Picking**
 - **Only the bounding box is highlighted during pre-highlight.**
 - **Sub objects of a view like a hatch in a view can be picked directly even if the View on filter on filter option is enabled.**
 - **Over-lapping views can be picked by holding Alt key.**

Drawing

New “Auto Associate Entity”

New entities will be automatically associated with a view if they are drawn within the bounding box of the view when the Turn “Auto Associate Entity” On/Off button on the DA toolbar is pressed down. Users can disassociate the entities with the “Associate/Disassociate” command as before.

Upgraded Basic Edit Tools to Constraint View Edit by View Projection Relation

Views operated by “Move”, “Rotate” and “Copy” commands are only moved around on the track of projection direction if their projection alignment is not broke. Break the projection alignment first if you want to manipulate them randomly.

★Dimension

New Tolerance Zone and Fit Tolerance

- Tolerance zone is newly supported with 4 different formats as following:

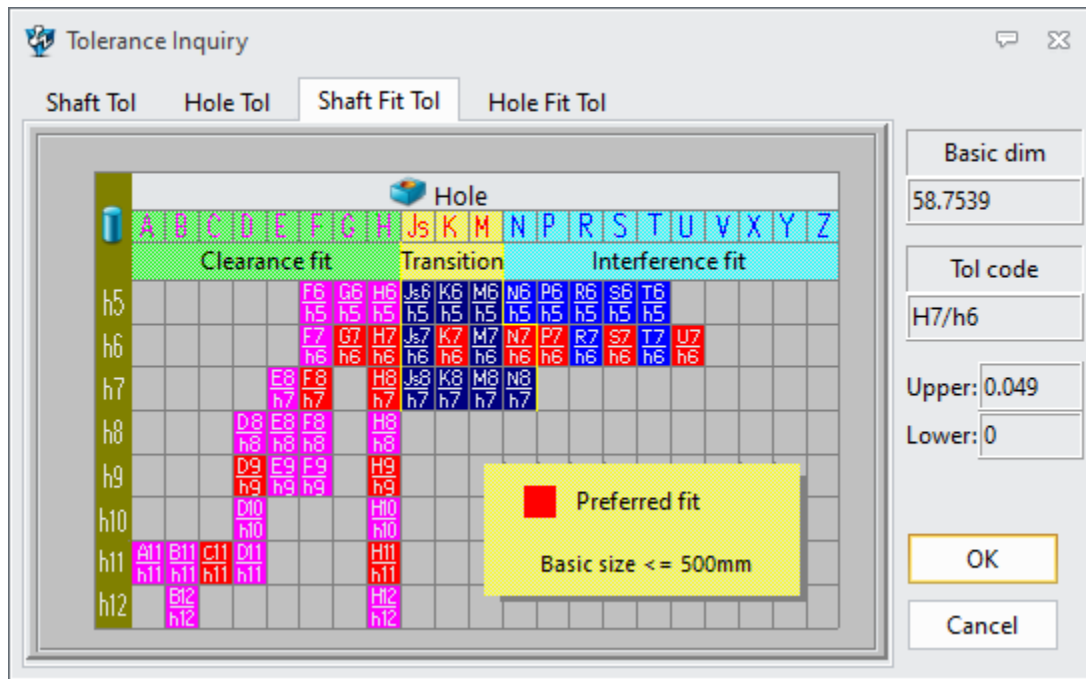
Fit	Fit with tolerance	Only tolerance	Show parentheses

- New fit tolerance with 2 different formats as following:



- New fit tolerance inquiry form

All available tolerance codes are provided in a graphic table which you can click onto. Tolerance zone and fit tolerance are both supported. After picking a fit tolerance, Tolerance code and Upper/Lower tolerance will be read out based on the Basic dimension.



❖ Where to Find

Drafting > All dimension tools > Dim Attributes

Right-click menu (on a dimension) > Attributes

Dimension Elements Used as Dimension Input

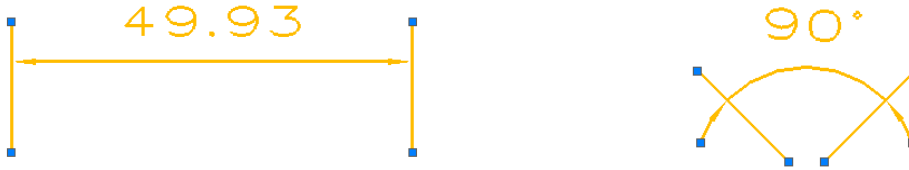
Dimension elements like extension lines now can be picked as input for the creation of a new dimension. Here is the list of specific dimensions whose elements can be picked as dimension input.

New Dim To pick	Valid Elements	Linear Dim	Chamfer	Angular	3 Points Angular	Arc Angular	Radial/Diametric	Arc Length	Hole Callout	Center Mark	Center Line	Label	Datum Feature	Feature Control	Datum Target	Surface Finish	Weld	User Symbol
Linear Dim	Ext Ln + CM Ln	√	√	√	√	√		√			√							
Chamfer	-																	
Angular	Ext Ln + CM Ln	√	√	√	√	√		√			√							

3 Points Angular	-														
Arc Angular	-			√		√	√	√			√				
Radial/Diameter	Ext Arc+CM Arc			√		√	√	√			√				
Arc Length	-			√		√	√	√			√				
Hole Callout	-														
Center Mark	-														
Center Line	-														
Label	Ext Ln + CM Ln + Ext Arc										√				
Feature Control Datum Feature	Ext Ln + CM Ln + Ext Arc	√	√	√	√	√		√			√				
Datum Target	-														
Weld Surface Finish	CM Arc + CM Ln + Ext Arc + Ext Ln	√	√	√	√	√		√							
User Symbol	-														

Grip Edit of Dimension

Elements of dimensions like dimension point, extension line, and arrow can be dragged to redefine their position or shape. Grip points are provided on these valid elements after a dimension is picked.



- Dimension points can be dragged to snap to another entity to redefine its dimension target.
- Extension lines can be dragged to extend or shorten their length.
- Single click on arrows can reverse their direction, while double click is to bring up the Modify Arrow form to change arrow style.

Auto Center Line Generation for Cylindrical or Conical Faces

Axes of cylindrical and conical faces will be auto-generated if they are available during view creation. The center line of circular patterns is also supported.

New options are provided in Configuration > 2D to control which center mark and centerline will be auto generated.

❖ Where to Find

View Attribute > Show center lines

Configuration > 2D > Drawing Sheet

Diameter Auto Recognition to Add ø

When using “Dimension” and “Linear Offset” to dimension the entities of cylindrical or conical surface, the ø symbol will be added into the dimension being-created.

Dangling and Modified Dimension

Dangling or modified dimensions are still marked in a pre-defined color to indicate their status but their original color attributes remain unchanged. After the issue that made them dangling is fixed or after users have confirmed the modification, dimensions will be restored to their original color.

To fix a dangling dimension, you can redefine the dimension points by dragging their grip point to valid locations.

Regenerating a view and its dimensions, or moving/redefining modified dimensions will cause modified

dimensions to restore to their original colors on the assumption that you have noted the changes.

New Bend Line in View for Bends of Sheet Metal



“Show Bend Line from Sheet Metal” button is added in view attribute form to control whether the bend line of sheet metal should be displayed. A new Bend Line is added into Lines type in view attributes form for customization of its attributes.

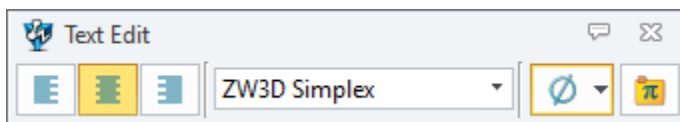
Others

- **Auto orientation of surface finish symbol to make it perpendicular with the entity it dimensions to.**
- **New mode in Ordinate dimension to add Horizontal and Vertical ordinates at the same time.**

★Updated Table

More Table Attributes to Control

- Table header can be at the bottom or top of the table corresponding to its sorting rule “From Bottom to Top” or “From Top to Bottom”.
- Color, line style and line width of table board lines are supported to customize.
- New formats for text: Normal, Text, Integer, Number and Percent. Decimal places and zero suppression are supported for number and percent format.
- More text attributes: Bold, Underline and common attributes of regular text. Through the new Text Edit form, characters in the same cell can be set to different fonts, and special characters and variables can be easily inserted. Multi-row text is supported through Alt + Enter combination keys.



- 3 different height settings of rows and columns: manually drag-drop to adjust the height, auto height adjustment and locking the height.
- Rows and columns can be hidden and shown.
- Cell can be locked to block the edit. If the data of cells comes from part attributes like BOM table does, these cells will be locked automatically. This is also done for a table linked to an external spread sheet.
- Each cell can be set to different attributes.
- Formulas are supported in tables.
- Table attributes are included in style manager.

Common Table Picking and In-place Edit

- Titles of rows and column will be displayed after a table is highlighted or picked.

	A	B	C
1			
2			
3			

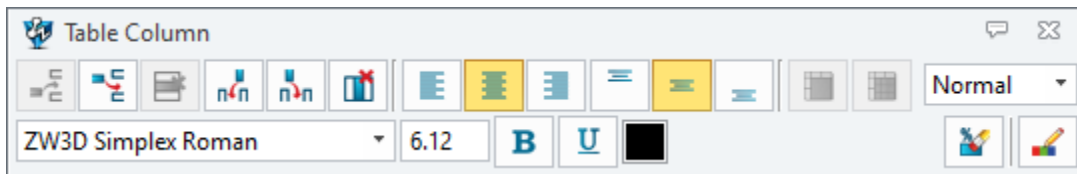
- Single-click on  is to pick the whole table, and dragging around with holding down the key can move the table. Double-click will start to edit the table. Delete key is supported. A table on-screen form is provided after a table is picked. Right-click menu is also provided.



- An arrow mark ⇨ will appear if the pointer is on left side of a table. Single-click when the arrow shows can pick the whole row. Picking a row and holding down the key can drag the row to move to another location. Table row on-screen form is provided after a table is picked. Right-click menu is also provided.



- Same functionalities are provided for column as row.



- Holding and dragging around the pointer on cells can select multiple cells around the picked cell. Same work can be achieved by dragging the handle . The Shift key is supported. Double-click on a cell can start in-place editing, or direct input from keyboard to edit the text after the cell is picked. Cell on-screen form is provided. Combination keys are supported as following:
 - Alt + Enter: start a new line.
 - Shift + Enter: pick the cell above the active cell.
 - Tab: pick the right cell of the active cell.
 - Shift + Tab: pick the left cell of the active cell

- Arrow keys: supported.

	A	B	C
1			
2			
3			



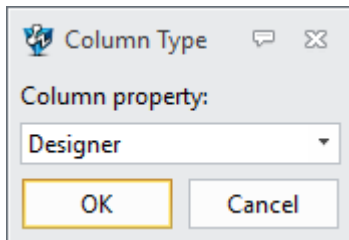
- Ctrl + C, Ctrl + X, Ctrl + V are supported for texts in a cell.
- The sizes of cell, row, column and the table can be changed by their drag handles ↕↔↕↔.
- Connective cells can be merged together. The merging can be canceled, and original data will be resumed.
- Table can be linked to an external spread sheet. And after the linkage, the table can be updated to the spread sheet. The link can be broken.
- After picking a row or a cell, a table can be split into 2 tables from this. A table can be split into several small tables, and remerged back.
- Table can automatically and magnetically attach to title block, border lines or other table border lines.

Formula

- Similar with Excel, formula always starts with “=” symbol, and can include following content:
Function, Reference, Operators and Constant.
- Reference uses \$ symbol to distinguish relative reference or absolute reference.
- 4 different types of operators are supported:
 - Arithmetic operators: +, -, *, /, ^ (Power), %.
 - Comparison operators: =, >, <, >=, <=, <>.
 - & operator can combine several strings into one.
 - Reference operators: colon: (argument separator), comma,(combination operation), space (intersection operation)
- Supported functions: SUM, AVERAGE, COUNT, MAX, MIN, IF, ABS, ATAN, COS, EXP, LN, ROUND, SIN, SQRT, ACOS, ASIN, TAN.
- Error mark will appear if the formula cannot be resolved: "#NULL!", "#DIV/0!", "#VALUE!", "#REF!", "#NAME?", "#NUM!", "#N/A", "#FUNC!".
- During auto fill, formula will handle the reference basing on copy/past rule.
- When importing or linking to an external spreadsheet, the formula within will be imported, not its result.

Upgraded Table Commands

- All improvements mentioned above are implemented to BOM, Hole and Electrode and User tables.
- Style list is added into table attributes.
- During ID modification, if a new id is the duplicate of an existing id, these conflicted ids will exchange their number, and their balloons will be updated at the same time.
- The header row of a table can't be deleted, nor the label id column.
- If a BOM/Hole table/Electrode table is linked to an external spreadsheet, its association with a view and part will be deleted, and it will become a user table.
- Double-click on column title will bring up the column type form to change its property.



- Picking a row from a BOM table will highlight the corresponding component.
- New "Save to File" button in BOM table form.
- New "Full Depth" column property in Hole table form is to calculate the hole length including the tip.
- New "Thread" column property in Hold table form is to callout holes' specification. And the old "Thread" property is renamed to Pitch.
- Updated User table to define quantity of row and column, and attributes.

CAM

Basic

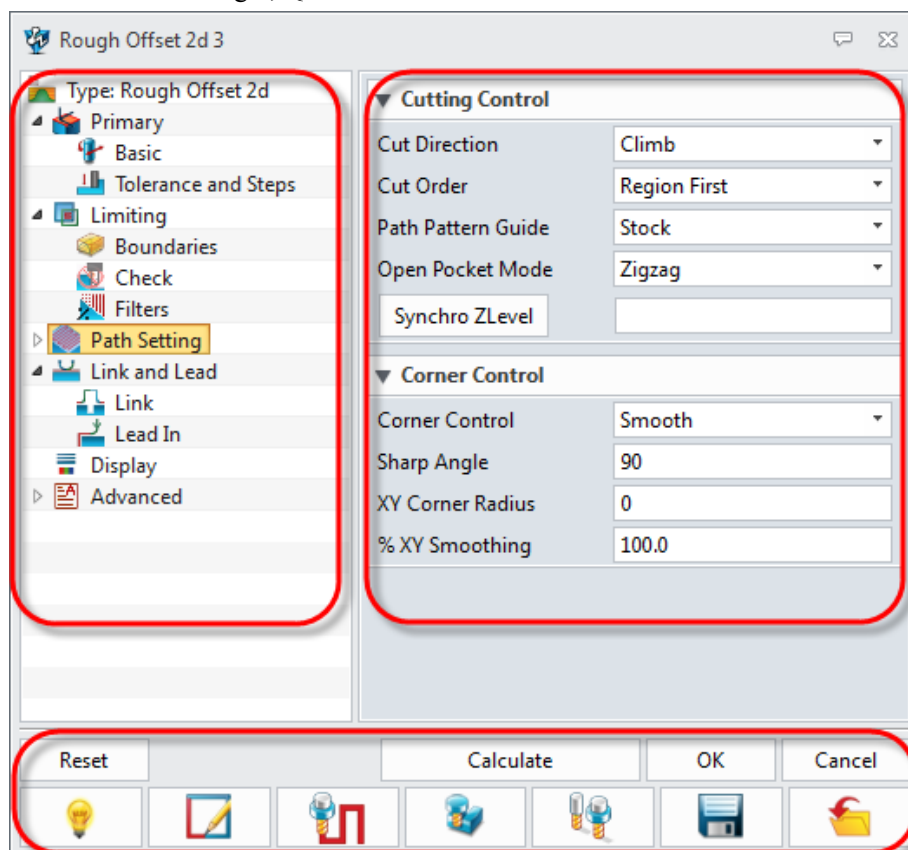
★CAM Operation Parameters Setting

CAM operations setting forms have a new style and new layout, which is more simple and clear to use. And all parameters of different operation kind have been re-sorted and located; they will be associated and learned more easily.

❖ How to Use

➤ See the new layout

Open an operation setting form, there are three parts: Navigation Tree on the left, Detailed Parameters on the right, Quick Buttons on the bottom.



➤ Meet the new sorts of parameters

As you can see, there are four major sorts: Primary, Limiting, Path Setting, Link and Lead.

Primary:

Including frame, speeds & feeds, tolerance, thick and cutting steps.

Rough SmoothFlow 1

Type: Rough SmoothFlow

- Primary
- Basic
- Tolerance and Steps
- Limiting
- Boundaries
- Check
- Filters
- Path Setting
- Link and Lead
- Link
- Lead In
- Display
- Advanced

Basic

Frame

Speeds, Feeds: Rough SmoothFlow 1

Tolerance and Thick

Path Tolerance: 0.1

Surface Tolerance:

Surface Thick: 1

Z Surface Thick:

Cutting Steps

Stepover: % Tool Dia 45.0

Stepdown

Stepdown: Absolute 5

Number of Cuts: 0

Reset Calculate OK Cancel

Limiting:

Including XY/Z boundaries, reference tool, check and filters of tool path.

Rough SmoothFlow 1

Type: Rough SmoothFlow

- Primary
- Basic
- Tolerance and Steps
- Limiting
- Boundaries
- Check
- Filters
- Path Setting
- Link and Lead
- Link
- Lead In
- Display
- Advanced

XY

Containment Type: Simple Box

% Offset: 300.0

Z

Top:

Bottom:

Check

Check all of Part: No

Check Shortest Tool: No

Reset Calculate OK Cancel

Path Setting:

Including tool path setting and the control of finish passes and corner.

Rough SmoothFlow 1

Type: Rough SmoothFlow

- Primary
- Basic
- Tolerance and Steps
- Limiting
- Boundaries
- Check
- Filters
- Path Setting
- Link and Lead
- Link
- Lead In
- Display
- Advanced

Cutting Control

Cut Direction: Climb

Cut Order: Region First

Path Pattern Guide: Stock

Open Pocket Mode: Zigzag

Synchro ZLevel:

Finish Pass

Add Finish Pass: Yes

% Finish Distance: 50.0

Corner Control

Corner Control: Smooth

Sharp Angle: 90

XY Corner Radius: 0

% XY Smoothing: 30.0

Reset Calculate OK Cancel

Link and Lead:

Including link, lead in and lead out setting.

Rough SmoothFlow 1

Type: Rough SmoothFlow

- Primary
- Basic
- Tolerance and Steps
- Limiting
- Boundaries
- Check
- Filters
- Path Setting
- Link and Lead
- Link
- Lead In
- Display
- Advanced

Lead Mode: Intelligent

Link

Long Link Type: Optimized

% Short Link Limit: 300.0

Safe Distance: 5

Lead In

Ramp Angle: 2.0

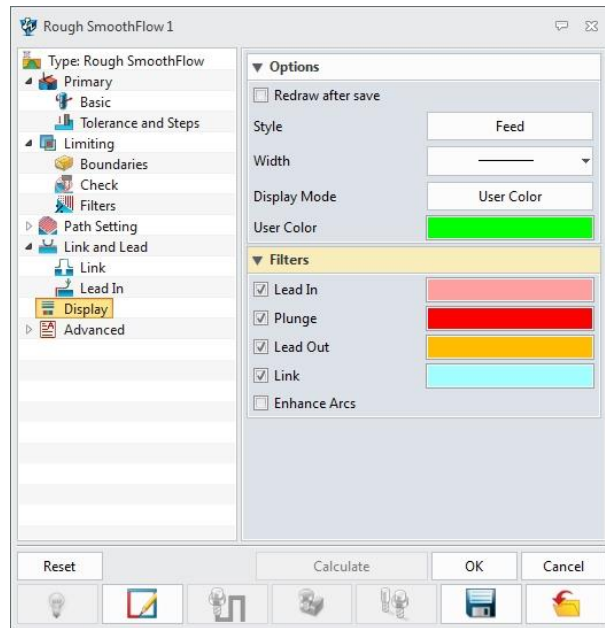
% Ramp Extent: 80.0

Safe Ramp: No

Reset Calculate OK Cancel

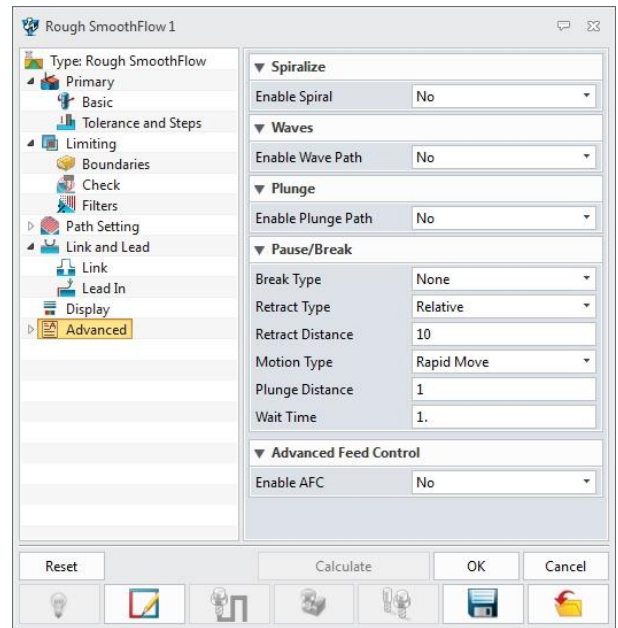
Display:

Including the display options and filters setting of tool path.



Advanced:

Including more path pattern, advanced feed control and other setting.



➤ The recommended way to set

It is quite recommended to set the parameters just follow the up-down order of Navigation Tree.

- 1) We should set the “primary” parameters like frame, tolerance and cutting steps, which is basic and related to the selected tool and work processing.
- 2) Some limits may be need including the boundaries, checks and so on, which are helpful to get tool path in the desired regions and reduce the time to calculate.
- 3) It is going to focus on “Path Setting” which decides what tool path looks like. For example, “Cut Direction” is about Climb/Conventional/Zigzag cutting, “Cut Order” is to choose if cutting by regions or by level, “Path Pattern” is showing all supported patterns of current operation.
- 4) “Link and Lead” may be needed to deal with, select desired Link Type and set Lead in/out based on the part or the tool.
- 5) “Display” is remained because it is convenient to modify the display mode of tool path.

➤ How to find the “missing” parameters

In the new UI, it is annoying when we cannot find parameters. These parameters are not “missing” and may be re-name or re-location. And in Quick Mill, they also may be put in “Advanced” which is including the advanced and detailed to control.

❖ What are Changed

➤ About their names

Sorts	Old name	New name
Cutting Steps	XY Step	Stepover
	Z Step	Stepdown
	Step Size	Step Value
Thickness	Depth Thick/ Vertical Thick	Bottom Thick
	Radial Thick/ Horizontal Thick	Side Thick
Check	Min Tool Length	Check Shortest Tool
Path Setting	Cutting Mode	Cut Direction: Climb, Conventional, Zigzag
	Cut Regions	Cut Order: Region First, Level First
	Cut Dir	Cut Angle
	Cut Pattern/Cut Mode	Path Pattern
Finish Pass	Side Cleanup	Side Finish Type
	Side Cleanup Distance	Side Finish Distance

➤ About their locations

Operations	Parameters	Old Location	New Location
Mill 2	Depth of Cut	TAB4 – Stock and Depth	Cutting Steps
	Stock Data	TAB4 – Stock and Depth	Tolerance and Thick/Boundaries
Quick Mill	Analysis Accuracy	TAB1 - Cut Parameters	Advanced – More Setting
	Uniform Cut	TAB2 - More Parameters	Advanced – Path Pattern
	Spiralize	TAB2 - More Parameters	Advanced – Path Pattern
	Matte Finish	TAB2 - More Parameters	Advanced – More Setting
	Advanced Feed Control	TAB5 – Feed Parameters	Advanced – Feed Control
	Curve and Surface	TAB4 – Limiting	Advanced – More Setting
Mill 5	Depth of Cut	TAB4 – Stock and Depth	Cutting Steps/Limiting

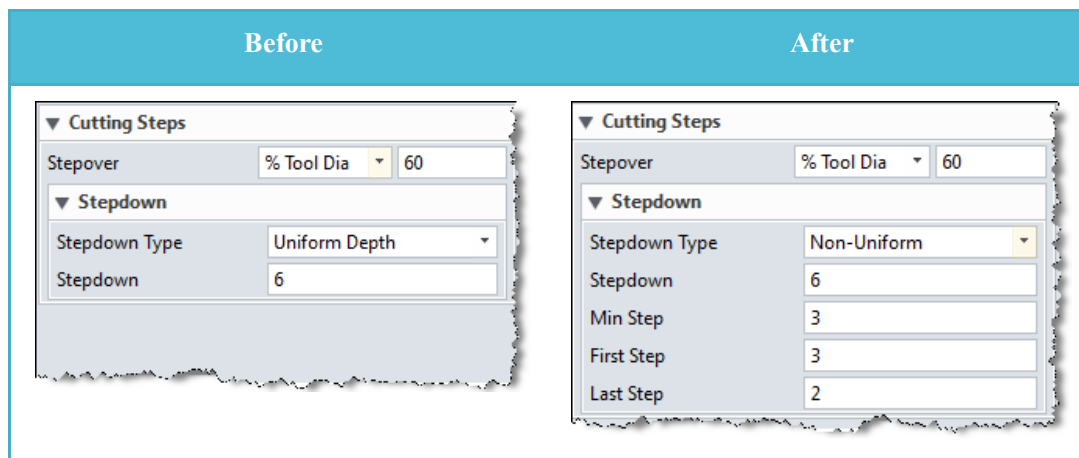
➤ About their options

Parameters	Old Options	New Options
Stepover Type	Center Dist	Absolute
	% Step Size	% Step
	Contact Dist	Contact Dist
	Dia Percent	% Tool Dia
	% Flute Length	% Flute Length
	Cusp Height	Scallop
	Number of Steps	Number of Cuts
Corner Control	Standard	Standard

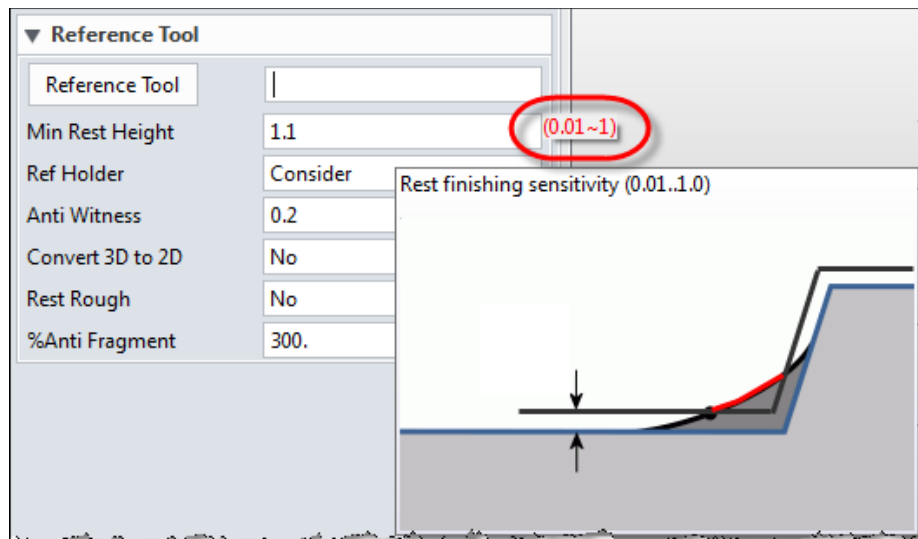
	Extend Tangent	Extend
	Tusk Cut	Tusk Cut
	Extend/Round Sharp	Round Sharp
	Fillet	Round
	d Loop	D Loop

❖ What is More

- Dynamically show/hide specific parameters after switching to another option.



- Add Red Hints to notice user when inputting value out of range.



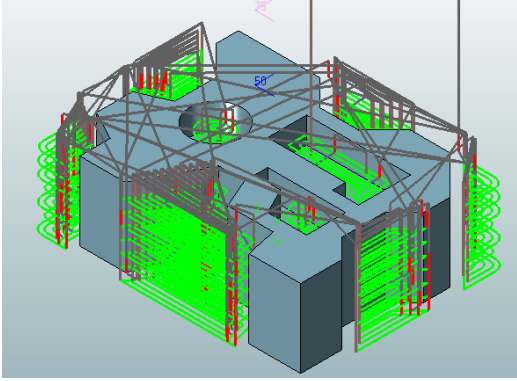
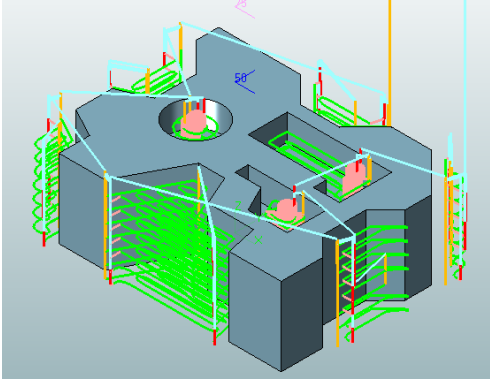
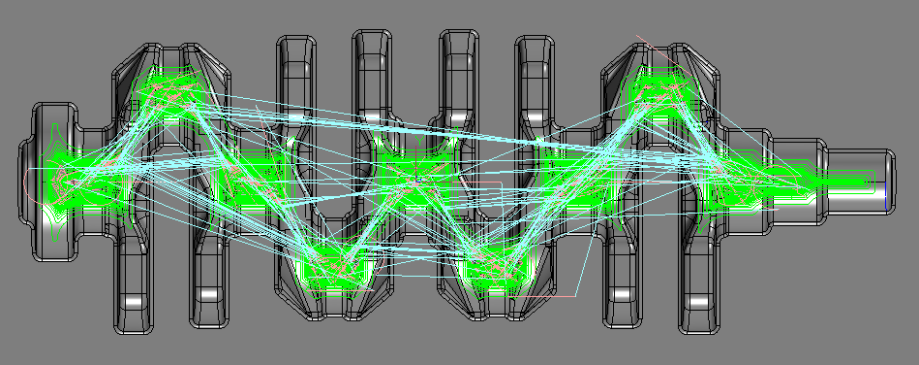
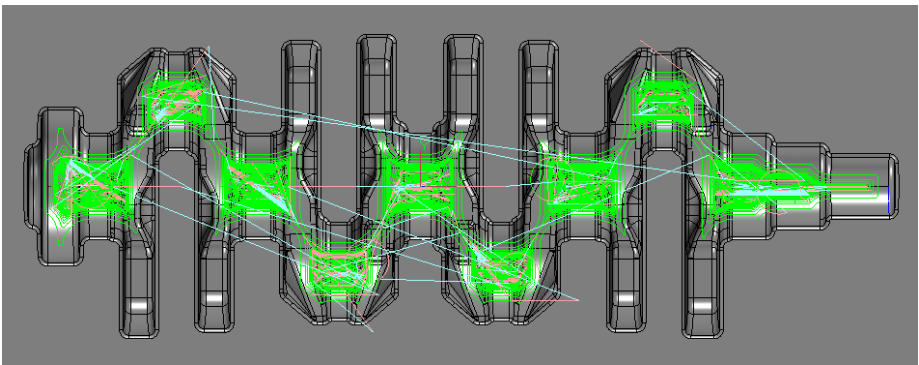
- The form will be collapsed when double-click on the top of parameter form.
- Default Up-Down-Left-Right Hot key to explore Navigation Tree when activating the tree.

Generating Tool Path

★More Optimized Cut Order of Roughing

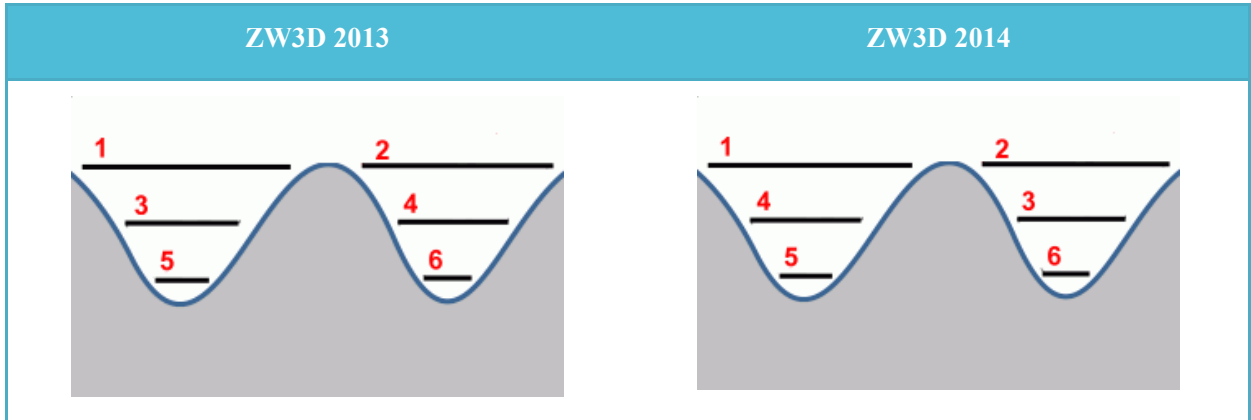
➤ Region First

In this version, the algorithm of Region recognition in Quick Mill Roughing is re-construction and the accuracy will be improved apparently if setting “Region First”.

ZW3D 2013	ZW3D 2014
	
ZW3D 2013	
ZW3D 2014	

➤ Level First

When setting “Level First”, it will lead in and cut the next level directly, to avoid the unnecessary tool lift back to the first cutting region.



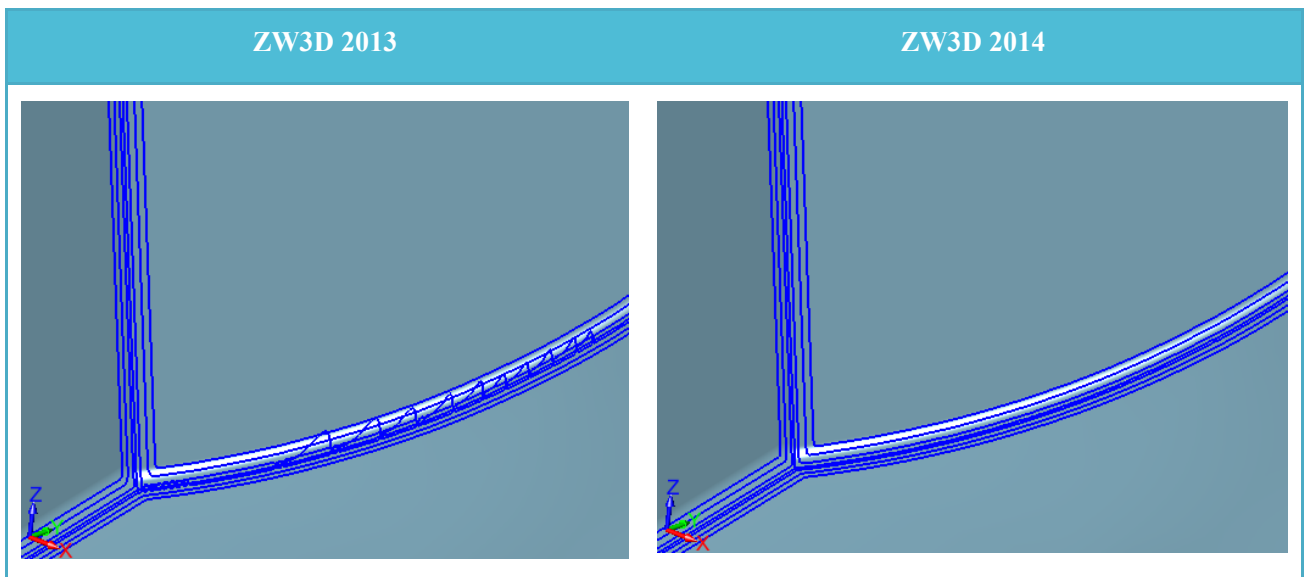
❖ How to Use

(In Rough Smoothflow, Rough Lace and Rough Offset2d)

Parameter Form > Path Setting > Cut Order

Smoother Tool Path

After adjusting the analysis slider to “Fine” in Surface Mesh Analysis, the tool path will be smoother which improve safety to machine. Setting with smaller Path Tolerance will be better. And because the mesh analysis is changed, the calculation efficiency will get a fall.



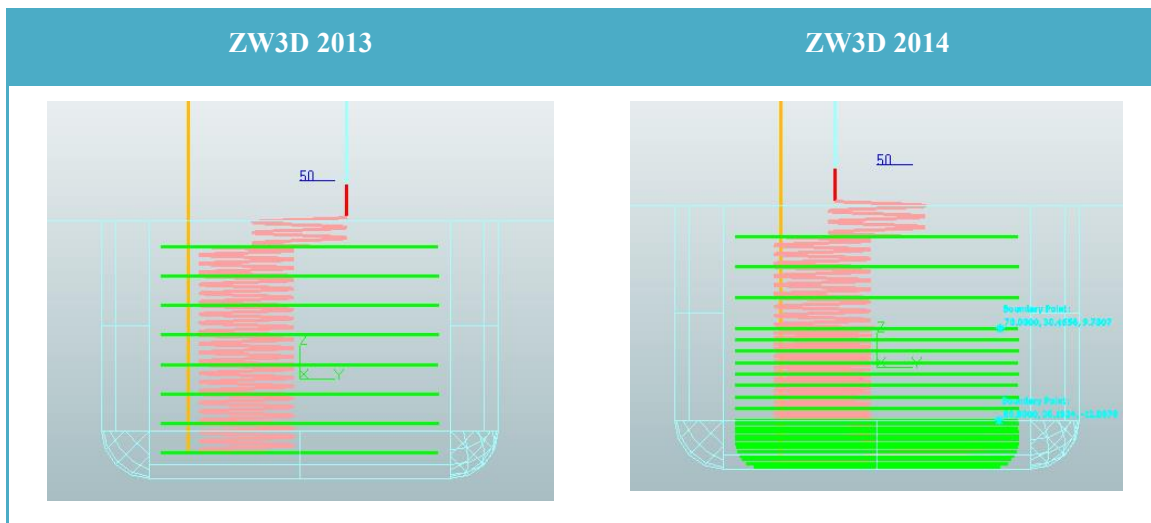
❖ Where to Find

Parameter Form > Advanced > More Setting > Analysis Accuracy > Fine/Coarse

Parameter Form > Tolerance and Steps > Path Tolerance

Support Multi-Step-Down in Roughing

Multi-Step-Down setting is supported in Roughing, which is similar with the setting in Zlevel operation. It allows user to get un-uniform step-down tool path.



❖ How to Use

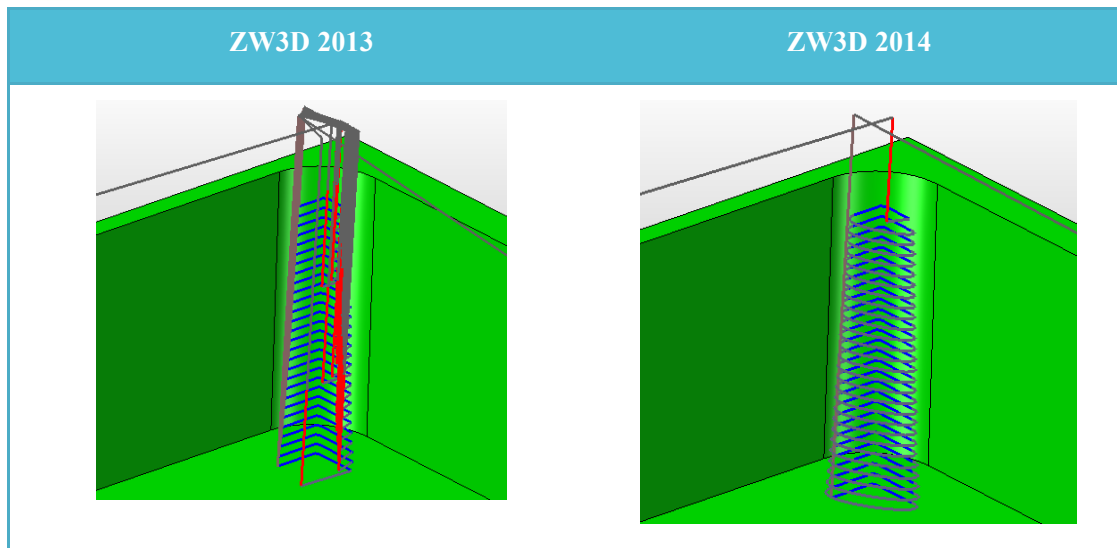
- Switch “Non-Uniform Cuts” to Yes.
- Select “Boundary Point” and set Stepdown Value to define Cut Layer.
- Click the buttons to Add, Edit or Delete layer.

❖ Where to Find

Parameter Form (Roughing) > Primary > Tolerance and Steps

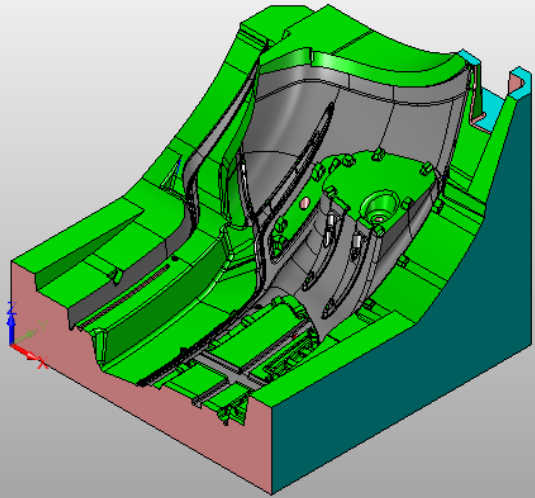
Optimized Link for Rest Roughing

The link is optimized between the cuts of two levels in Quick Mill Rest Roughing, just like the “ramp” link, which is more benefit to improve processing efficiency. And when setting the specific “Ramp Angle” refer to the value of XY stepover and Z stepdown, it will be better to create a “ramp” link for each level.



★Improved Accuracy of Minimum Tool Hang Length

The method of minimum tool hang length calculation is updated and the holder of the tool does collision detection with the dynamic stock, which will get more accurate result. And the minimum length will be outputted in XML with default setting.

ZW3D 2014	32.24 mm	
Other Software	32.6 mm	

New Extension of Tool Path

Extension tool path will make sure the whole surfaces can be machined completely and help to protect the corner shape. Now three extension types are supported.

- **Linear**

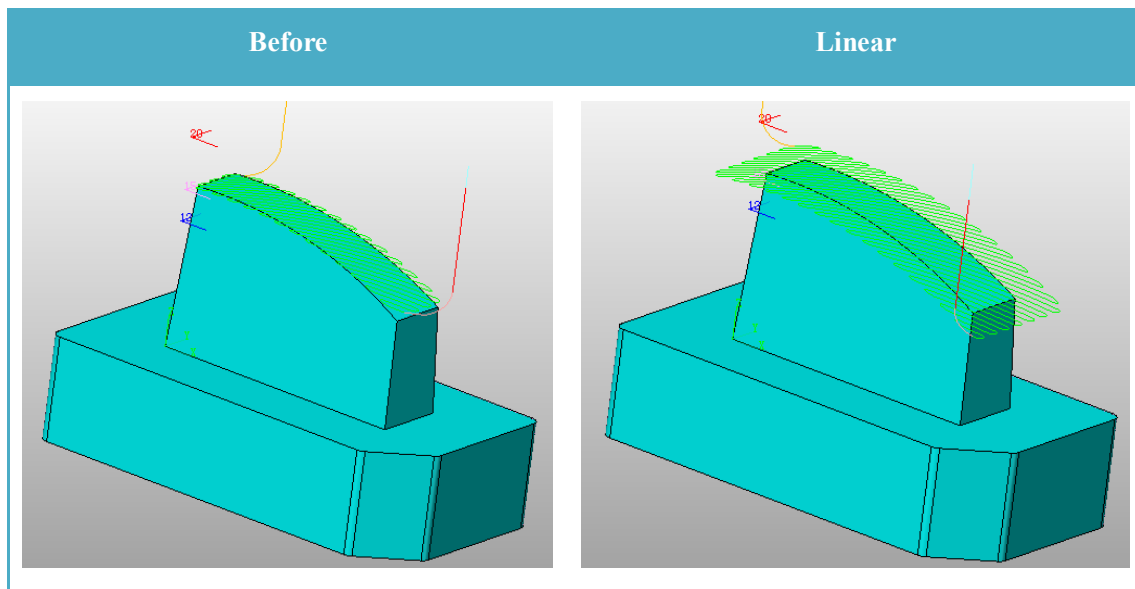
Extend tool path away from the extension point in the direction of the tangent

➤ **Circular**

Extend tool path as an curve path which matches the curvature of extension point.

➤ **Curvature diminishing**

Extend tool path with both Linear and Curve Method. It matches curvature at extension point and the curvature reduces, it eventually becomes linear.



❖ **How to Use**

- Create a surface feature of the surface that needs to extend, and change its class to “Extend Surface”.
- Insert Lace or Offset 3D operation, select the surface feature and a tool.
- Open parameter form and select an extension type, input a distance value, do the calculation.

❖ **Where to Find**

Finish Lace and Offset 3D operation:

Parameter Form > Limiting > Boundaries

❖ **Attention**

The tool path extension is not stable enough when the part is imported, especially with bad topology, like intersecting, gap, duplicated face and so on.

Others

➤ **Support T-Slot with radius**

Add the two options to define the top radius and bottom radius in T-Slot tool, and the defined radius can be used to calculate tool path.

➤ **More Stable in Angle Limiting**

Angle Limiting operation works better in imported cases, and the limiting boundary is more reliable.

➤ **Reference Tool**

Improve the accuracy of rest finishing by referring bigger ball mill tool.

➤ **Speeds and Feeds for Drill Tactic**

Operation of Drill tactic will select a suited tool and inherit its speeds/feeds in the library with tool and speeds/feeds. And the association between tool library and speeds/feeds library is still working in tactic.

➤ **Support to Solid Verify More Complicated Casting Stock**

Upgrade solid verify function and support to verify casting stock, which will be convenient to check the machining result in casting.

Processing Tool Path

★New Tool Path Editor

In this version, all the functions of tool path editor are classified and extended, the selection method of target object, the trim operation and the universal re-link setting is mainly updated and strengthened. The new tool path editor will be more helpful for user to edit and get the desired tool path.

❖ **Where to Find**

CAM Manager > Target Operation (Right-Click Menu) > Edit tool path

Ribbon > Edit Tool Path

Trim

Now three methods are supported to select the cuts of tool path:

➤ **Box Select**

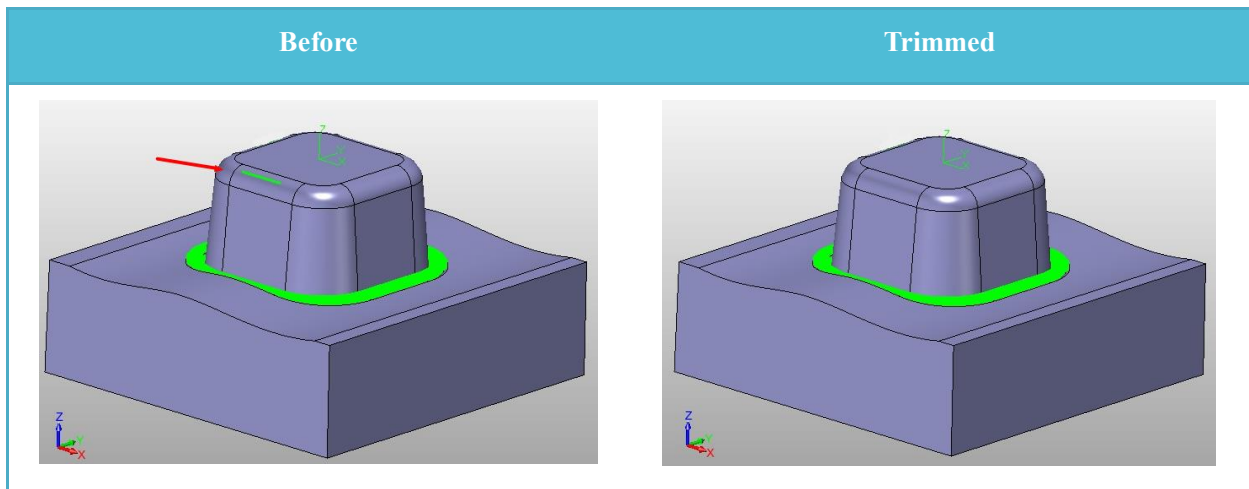
Use cursor drag a frame box to select cuts and apply to trim.

➤ **Chain Select**

Select cuts on chain with “Shift” key and apply to trim.

➤ **Polygon**

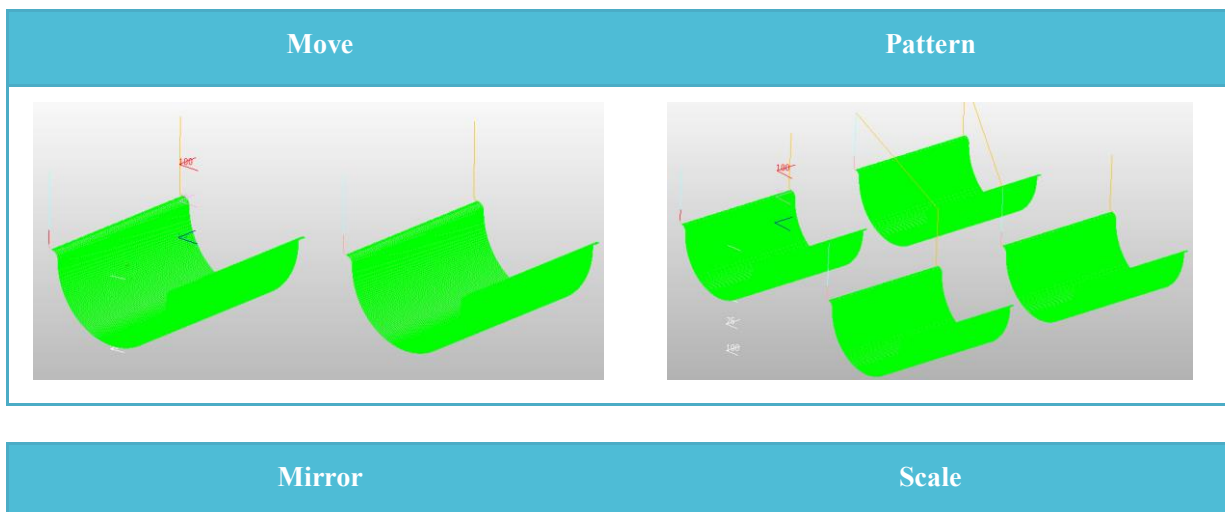
Draw a polygon in the viewer area and Middle-Click to apply.

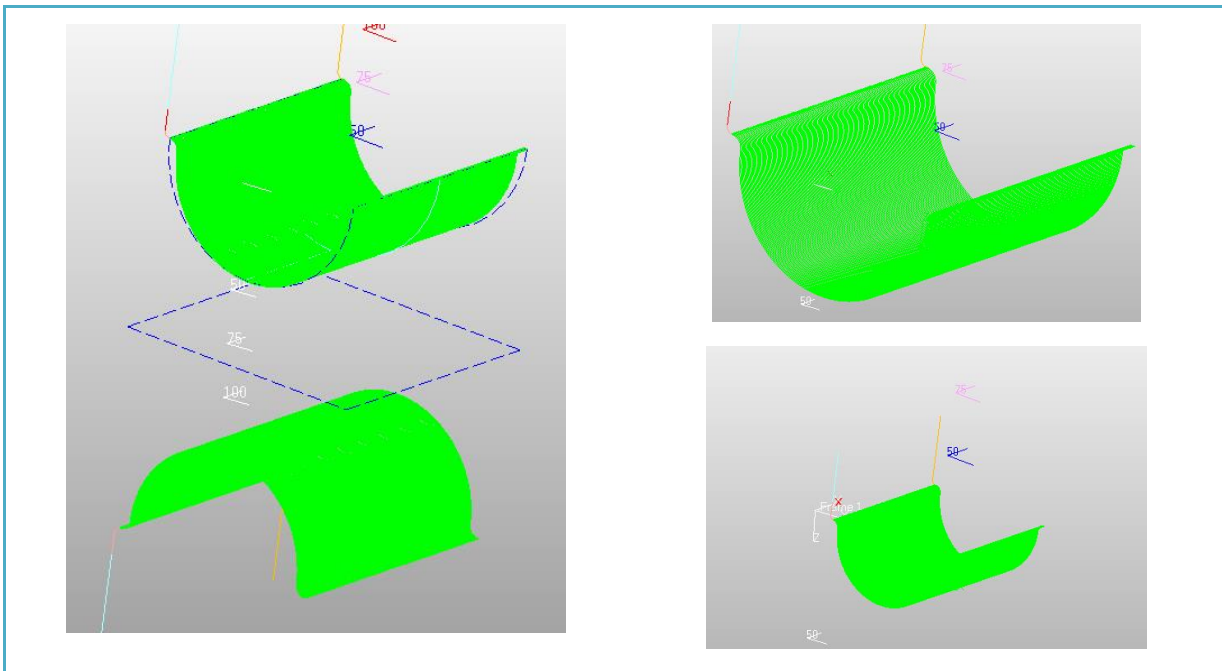


Transform

The workflow of transforming tool path is simplified: right click the target tool path and select the transform method, then choose the point or frame and confirm, then it will insert a new transform operation of the target tool path in CAM Manager. There are four methods of transform:

- **Move**
Move the target tool path by point to point, a direction, rotate around a direction or aligning frame to another frame.
- **Pattern**
Pattern the target tool path by Linear or Circular.
- **Mirror**
Mirror the target tool path by a selected plane (or datum).
- **Scale**
Expand or shrink the target tool path by %Scale value.





Re-Order

Tool path are made up with cuts and Re-Order will list all cuts in the sheet, allowing user to select the target cuts to edit. Re-Order methods include:

- **Delete**
Delete the selected cuts in the list.
- **Top**
Move the selected cuts in the list to the top.
- **Up**
Move the selected cuts in the list up one position.
- **Down**
Move the selected cuts in the list down one position.
- **Bottom**
Move the selected cuts in the list to the bottom.
- **Re-Order**
Reverse the order of selected cuts. If no selected cuts, the whole tool path is reversed.
- **Re-Direction**

Reverse the direction of each selected cuts, the start and end points are switched round. If no selected cuts, the direction of the whole tool path is reversed.

➤ **Change Direction**

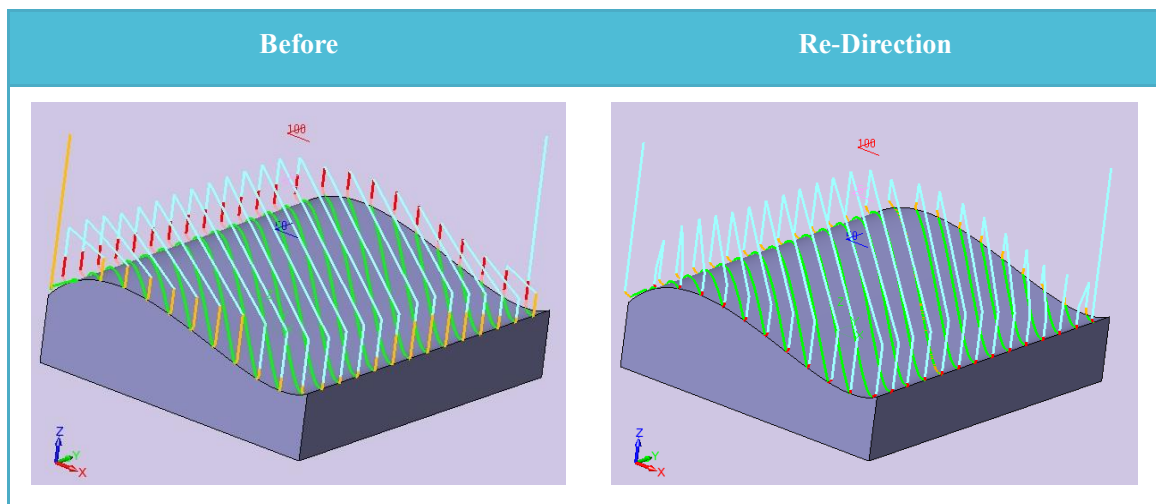
Reverses the direction of each alternate selected cuts, the start and end points of every other cut are switched round. If no selected cuts, it will be applied to the whole tool path.

➤ **Auto Reorder**

Reorder tool path to minimize 3D distance between ends of cuts, while maintaining the cutting direction of cuts. This should work on the entire tool path (not on selected cuts).

➤ **Auto Reorder and Reverse**

As for “Auto Reorder”, it allows cuts to be reversed the cut direction to minimize the lift between cuts. This usually results in fewer lifts than Auto Reorder.



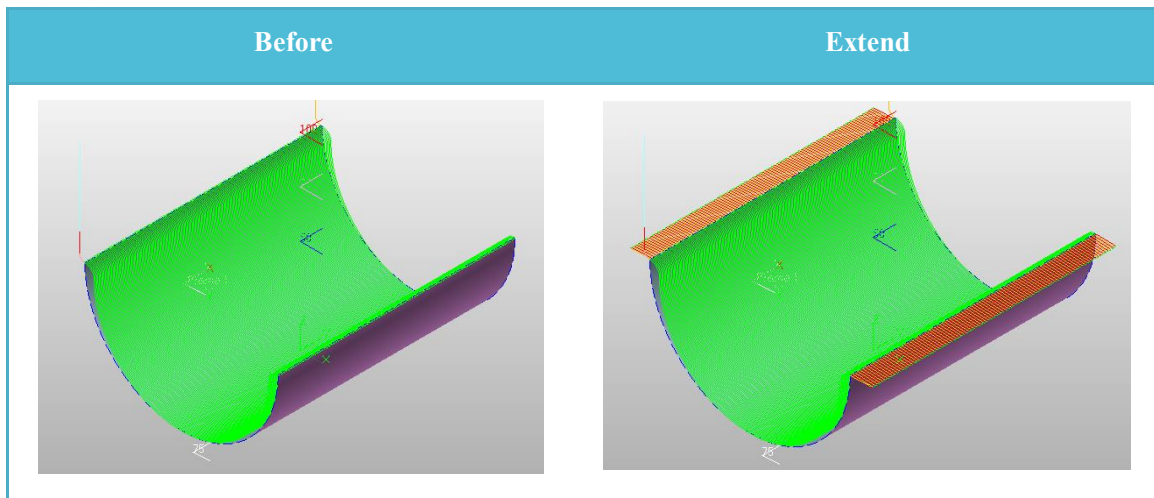
Re-Link

It allows to re-set links and leads, and apply to the target tool path.

What is more, in “Link and Lead” of Mill2 and Quick Mill, “Apply” button is added which will apply the setting at once without re-calculation after modifying parameters.

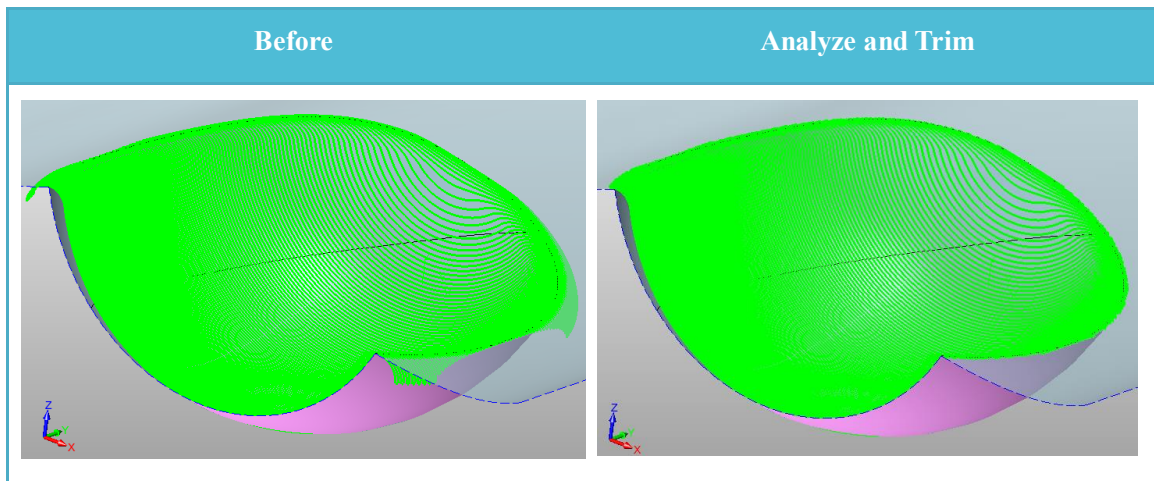
Extend

Enter a extend distance after selecting tool path and all the “Short Link” of the target tool path and will be extended with lead in and lead out.



Upgraded Tool Path Analysis

The algorithm of tool path analysis in Quick Mill has been updated and the analysis stability is improved. An option “Trim Unsafe Tool path” is added which will automatically trim the gouged and collided (unsafe) tool path if checked. And it allows user to select more than one part to analyze now.



❖ Where to Find

CAM Manager > Target Operation (Right-Click Menu) > Analyze Tool Path

Ribbon > Inquire > Detect > Analyze Tool Path