



ZW3D WhatsNew

V WuKong 2027

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ZW3D™ V WuKong 2027 What's New

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ZW3D WuKong 2027 Highlight Improvements



ZW3D is an easy-to-learn, user-friendly integrated CAD/CAE/CAM/collaboration solution that seamlessly combines "design, manufacturing, simulation, and collaboration" into one platform. It enables multi-disciplinary collaborative design based on a unified data source. With robust data compatibility, efficient design tools, and comprehensive industry-specific modules, ZW3D empowers engineers to deliver higher-quality designs in less time and shorten the product development cycles.

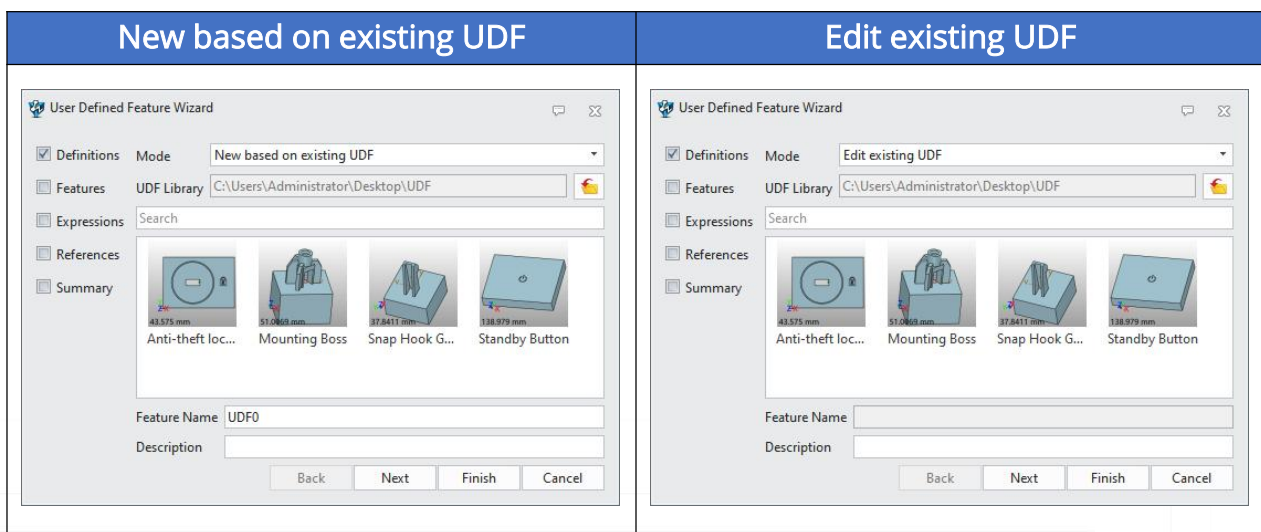
ZW3D is widely applicable across industries such as machinery, mold & die, high-tech electronics, and home appliances. It covers the entire product development life cycle — from conceptual design to manufacturing — and supports simulation, machining strategy development, and seamless integration of collaborative systems. By enabling unified management of full-process data from a single source and fostering cross-functional collaboration, ZW3D helps enterprises streamline workflows and accelerate their digital transformation journey.

ZW3D WuKong 2027 focuses on the critical needs of design engineers for parametric design in complex design scenarios, and systematically innovates and reconstructs the parametric modeling mechanism of ZW3D. By deeply optimizing the underlying architecture and algorithms, we have newly launched a highly reliable, stable, and efficient parametric design platform. This platform enables engineers to swiftly construct model designs that meet expectations when facing complex and ever-changing design challenges, achieving a significant improvement in design efficiency and substantially shortening the product development cycle. Meanwhile, this version further enhances aspects such as modeling capabilities, and user experience in assembly and engineering drawings. These improvements will collectively drive efficient design and collaboration within teams.

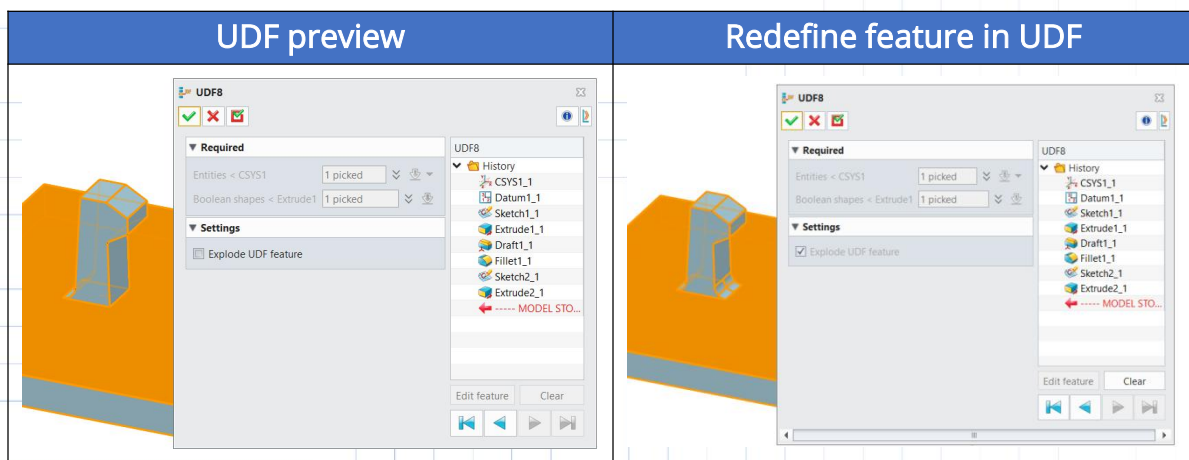
1 Basic

1.1 UDF Function Optimization

After creating a UDF file, you can create a new UDF file from the existing UDF file by the **UDF Wizard**. The new version also supports redefining existing UDF by adding, removing, or modifying required input fields such as descriptions, features, expressions, and references.



Inserting UDF becomes more flexible and controllable. It supports users to play/play-back features in UDF to display result directly. It is also available for users to edit features when inserting a UDF feature.

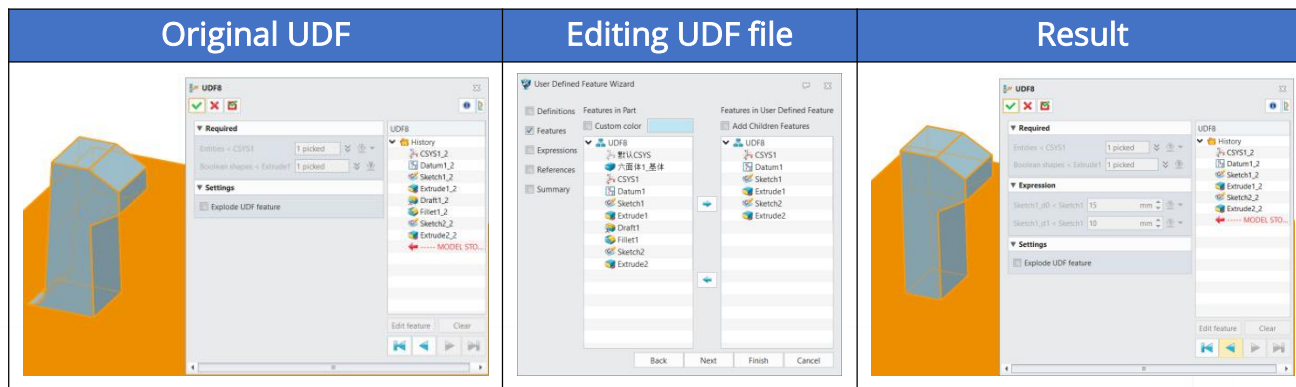


【Notes】

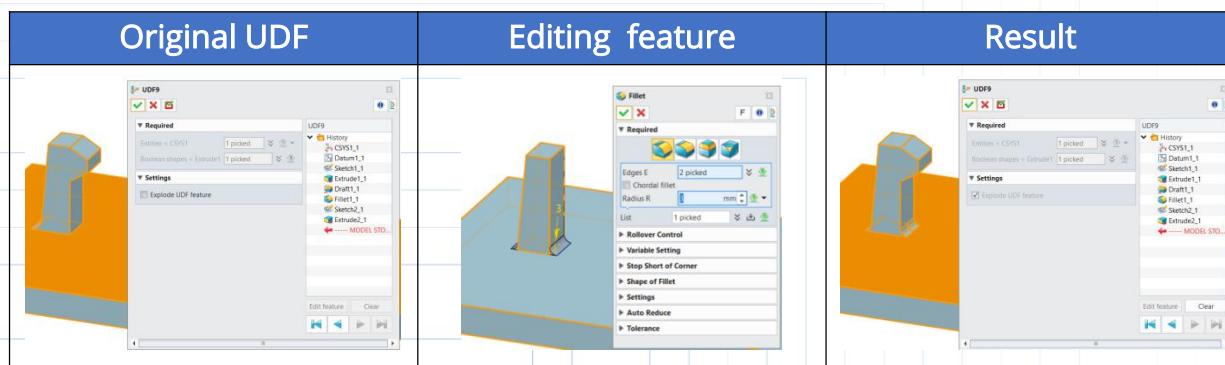
- UDF file created in the version before ZW3D WuKong 2027 need to convert to new version before inserting or modifying.
- New created UDF file in ZW3D WuKong 2027 or its later version will be saved as Z3PRT instead of Z3 file.

【Example】

1) Use **UDF Wizard** command to modify the UDF file after creating it, such as adding or removing features, adjusting expression or reference, and modifying definitions.



2) Insert a UDF feature of a hook and edit single feature (Adding fillet edge as an example) in it. Then play forward step by step to observe UDF result after editing single UDF feature.



【Where is it】

Part/Assembly >> Tools >> UDF Wizard

Part/Assembly >> Tools >> Insert UDF

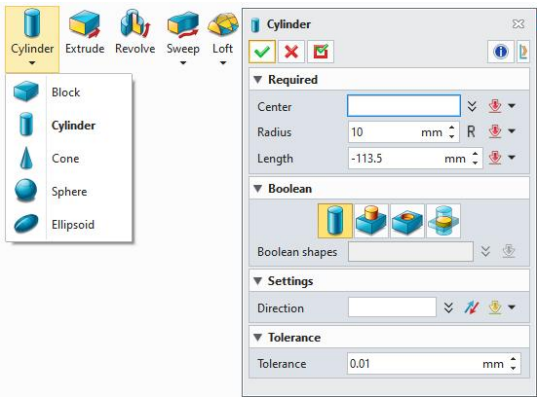
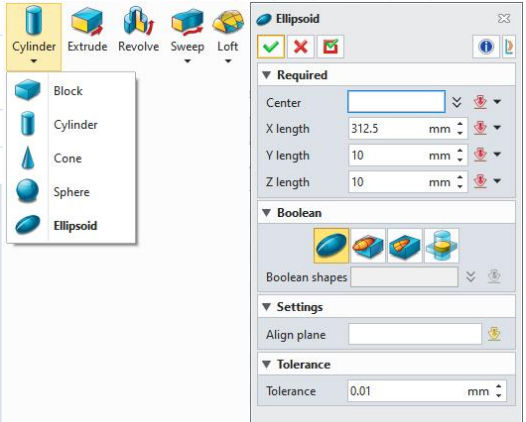
1.2 Customize Configuration Optimization

1.2.1 Ribbon Adds Dynamic Ordering Configuration

The Ribbon has introduced a new **Dynamic ordering** function, which allows for the management of the command displayed on the drop-down menu with a collapsed state. When this function is enabled, the system will automatically record the most recently selected command, and that command will be displayed on the Ribbon. When the function is disabled, the Ribbon will fixedly display the command that was last selected before the function was turned off. This function is enabled by default.

【Example】

Observe the command display in the quick modeling drop-down menu during the design process. By default, **Dynamic Ordering** is enabled. When switching command selections, the Ribbon will display the most recently selected command. If you right-click on a blank area of the Ribbon and disable dynamic ordering in the Ribbon appearance settings, then regardless of which quick modeling command is selected afterward, the Ribbon will only display the command that was last selected before dynamic ordering was disabled.

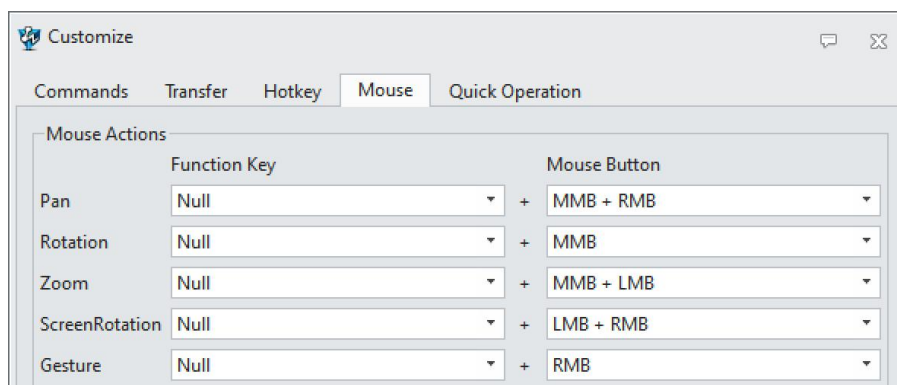
Enable Dynamic Ordering	Disable Dynamic Ordering
	
The command display on the Ribbon switches dynamically based on user selections.	The command display on the Ribbon is fixed to the last selection made before the function was disabled.

【Where is it】

[All Environments](#) >> [Ribbon](#) >> [Ribbon Appearance](#) >> [Dynamic Ordering](#)

1.2.2 More Flexible Mouse Behavior Configuration

The button configuration for mouse behavior now offers dual-button combination options, allowing you to customize mouse gestures with greater flexibility. This even enables all view operations to be performed by one hand.



【Notes】

- When the **Function key** is set to Ctrl/Shift/Alt, the **Null** option for **Mouse Button** is unavailable to prevent invalid configurations.
- When the **Mouse Button** is set to **Null**, and the **Function key** is changed to Ctrl/Shift/Alt, the **Mouse Button** automatically switches to **Middle Button** to prevent invalid configurations.

【Where is it】

[All Environments](#) >> [Custom](#) >> [Mouse Actions](#)

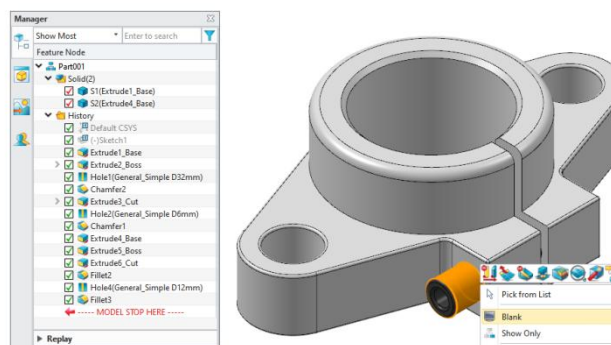
1.3 User Interface Optimization

1.3.1 Hidden Features Appear Grayed Out

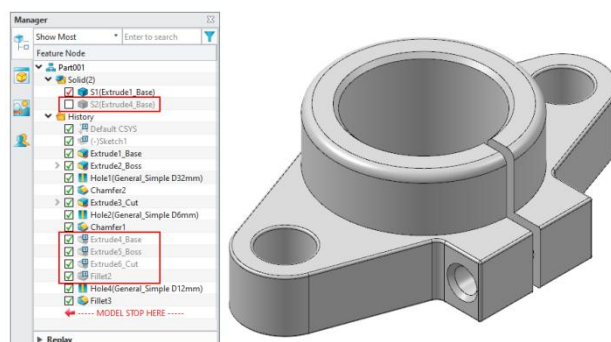
After hiding selected geometry or features in the feature tree or drawing area, the corresponding geometry in the feature tree and associated feature nodes are simultaneously grayed out. This allows designers to intuitively identify which features are linked to the current geometry, enabling quick localization of relevant features and significantly improving editing efficiency.

【Example】

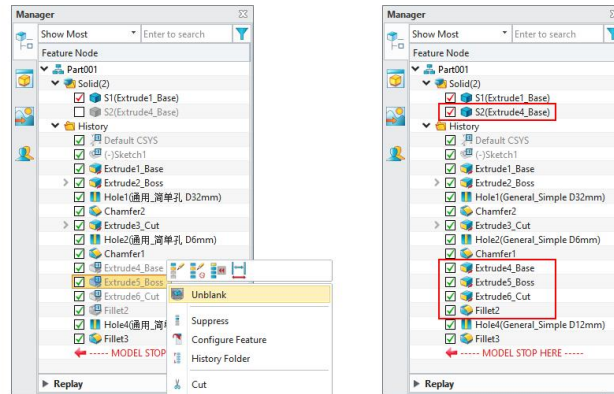
- 1) In the drawing area, select a face and hide the geometric body it belongs to.



- 2) In the feature tree, the corresponding body node and its associated feature nodes will be grayed out.



- 3) If any one of the hidden features is made visible again, its associated geometric body and feature nodes will be visible.



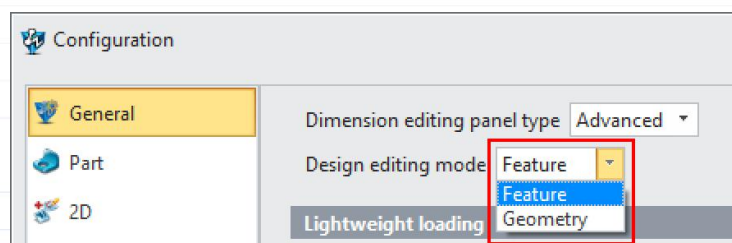
【Where is it】

Part Environment >> **Feature Tree**

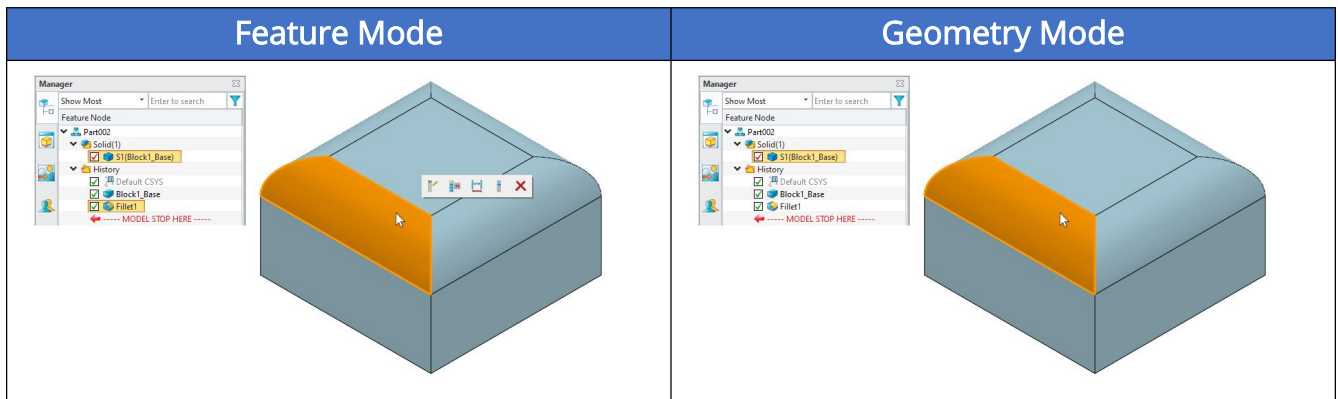
Assembly Environment >> **Feature Tree**

1.3.2 New Feature Editing Design Mode(ZW3D WuKong 2027)

Design editing mode option has been added to the system general configuration, with **Feature** set as the default.



In Feature Design Editing Mode, when users select a geometric object in the graphics area, they can edit or delete the dependent features.

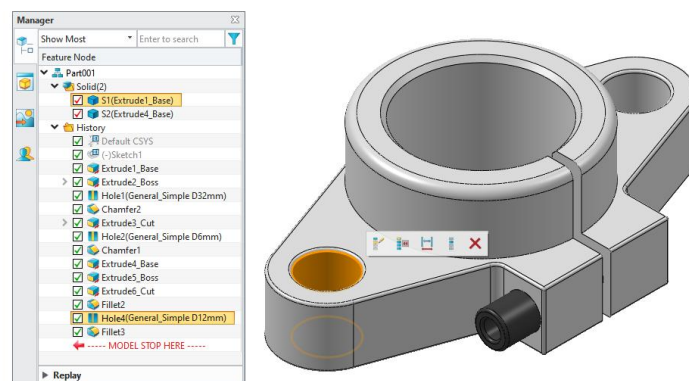


【What users can do】

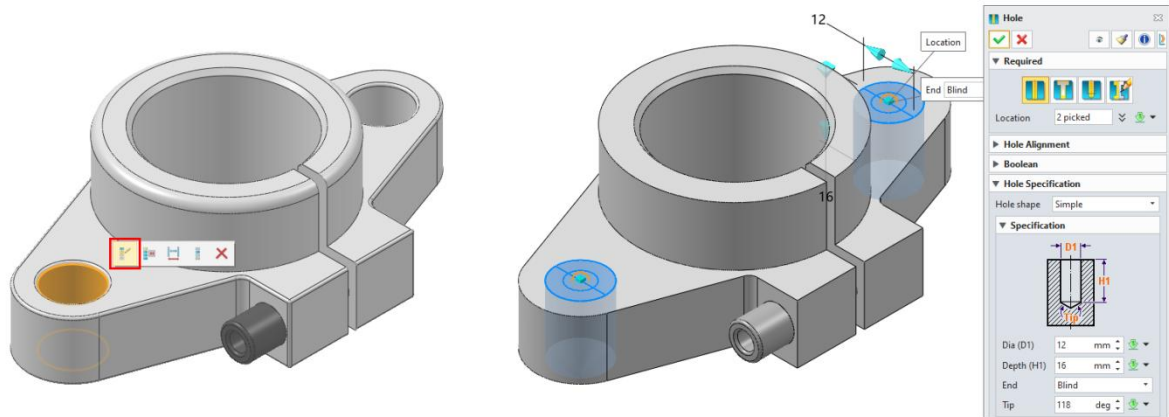
- ✓ Clicking a highlighted geometric object in the graphics area will synchronously highlight the dependent features in the feature tree.
- ✓ Selecting a single geometric object in the graphics area will display a floating toolbar for feature editing.
- ✓ Pressing **Delete** after selecting a geometric object will delete the dependent features instead.

【Example】

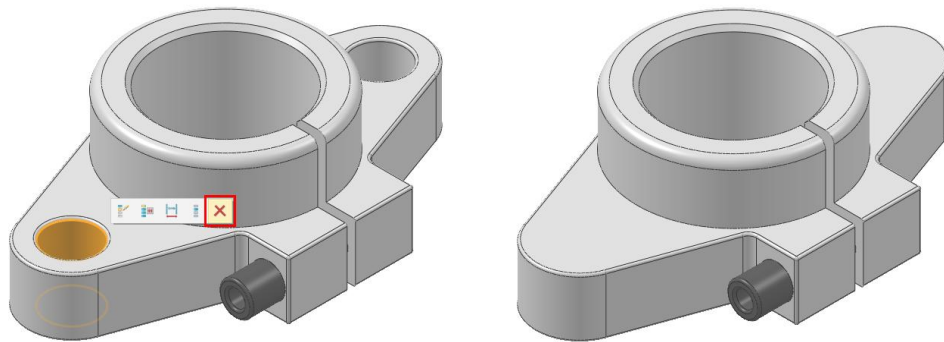
- 1) When you click a face in the graphics area, the dependent feature is highlighted in the feature tree.



- 2) Click Redefine to roll back the history and edit that feature.



3) Click Delete Feature or press Delete to remove the feature.



【Where is it】

Part/Assembly Environment >> Graphics Area

1.4 Inquire

1.4.1 Section View Enhancement

When creating a section view, switching between section modes automatically synchronizes the offset values in the same direction for **Section at plane**, **Section with slice**, and **Section with envelope** modes, ensuring a smoother transition.

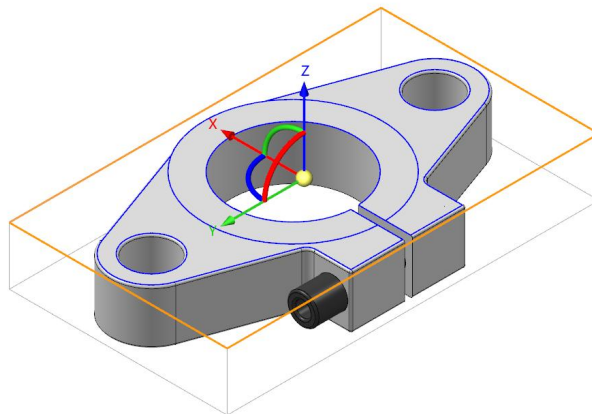
On the Section View Manager, the right-click context menu for the active section view node now includes **Section on/off** and **Reverse direction** options. This allows designers to quickly toggle the section visibility and reverse the cutting direction.

【Notes】

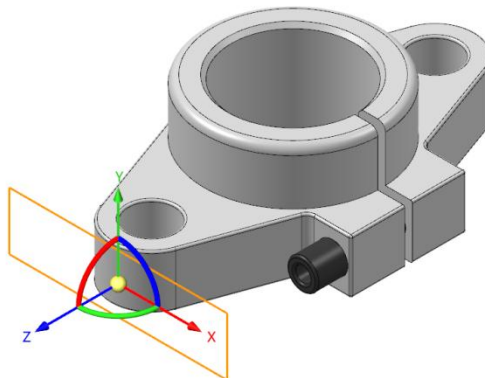
- The **Reset position** function behaves differently based on the presence of an align-in plane. If no plane is defined, it returns the section to the model center (offset=0). If a plane is defined, it moves the section back to that aligning plane.
- When a section view is activated based on a different datum plane orientation, the section is only re-positioned to the corresponding boundary of the section envelope box, without creating a half-section of the model in that direction.

【Example】

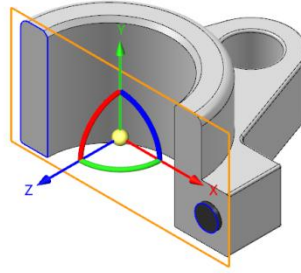
- 1) When a **Section at plane** view is created, it is positioned at the model center by default, using the top plane for the cut.



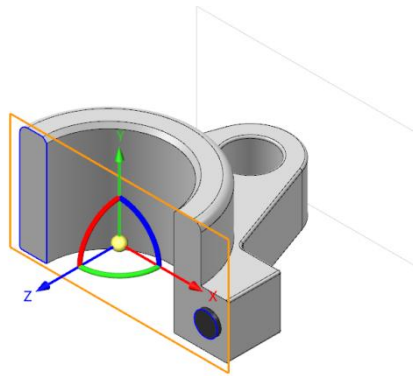
- 2) After activating the front Plane, the section moves to the front view of the section envelope box.



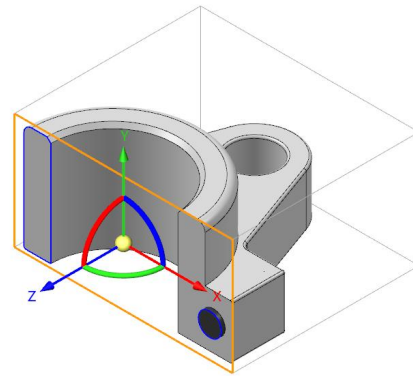
- 3) Clicking **Reset position** returns the section to the model center.



- 4) When switching successively to **Section with slice** and **Section with envelope** modes, the section results remain identical, and the offset values share the same value under the same base plane orientation.

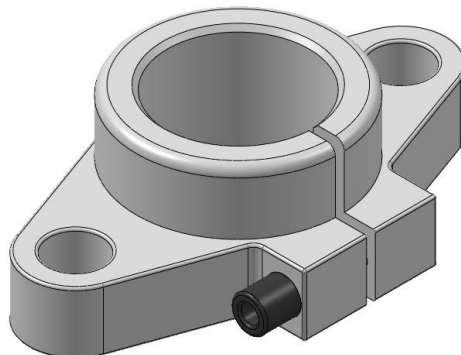
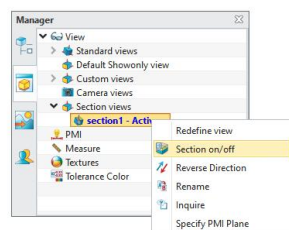


Section with slice



Section with envelope

- 5) Right-click a section view node in the Manager to turn it off.



【Where is it】

[Part Environment >> Inquire >> Section View](#)

[Assembly Environment >> Inquire >> Section View](#)

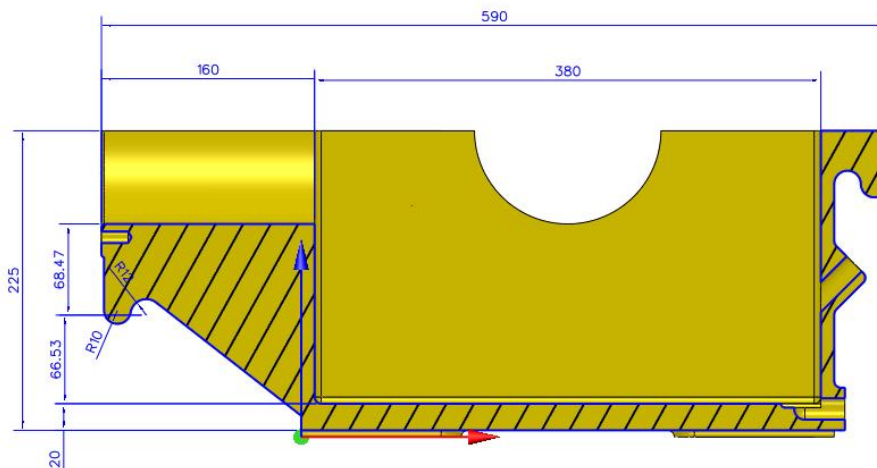
[View Manager >> Section View](#)

1.4.2 Section Support Annotation and Measurement(ZW3D WuKong 2027)

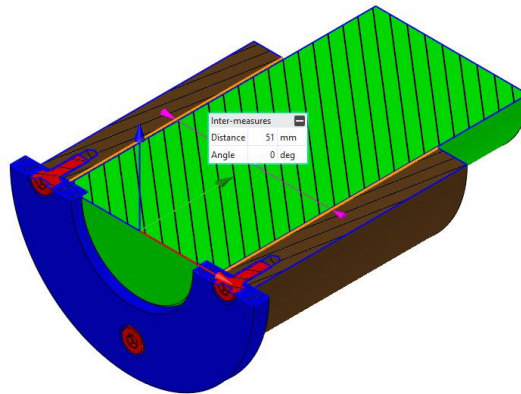
After creating a section in the model, you can pick up the section object to perform PMI annotations, measurement queries, and more.

【What users can do】

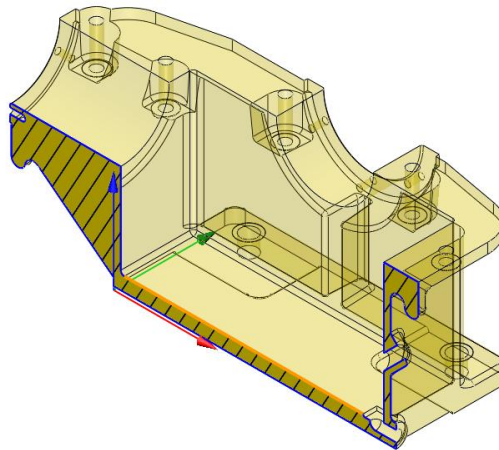
- ✓ Create PMI in the section view.



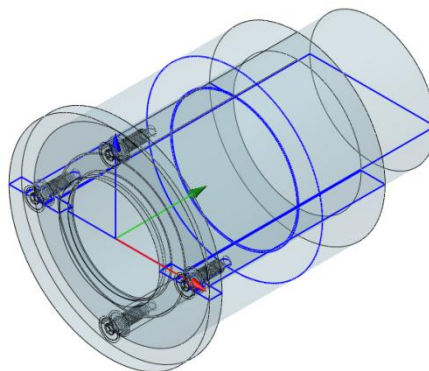
- ✓ Measure in the section view.



✓ When picking up a section element, its measurement value appears in the lower right corner.

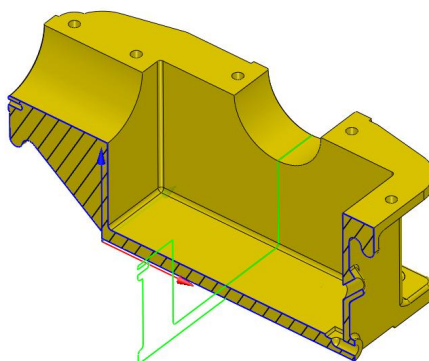


✓ Generate the intersection lines between sections and solids, and display the intersection results of multiple section views simultaneously.



✓ Display and save multiple section views with different curve colors, line widths,

and section line styles.



【Where is it】

[Inquire >>](#) [Inspect Model >>](#) [Section View](#)

2 Translator

2.1 Input Format Upgrade

We have upgraded both the default converter and the alternative converter to support the latest formats of mainstream 3D software on the market. The red parts indicate the updates in this release.

No.	Product	File Extension Name	Default Translator	Alternative Translator
1	Catia V4	.model, .exp, .session	4.1.9 - 4.2.4	4.1.9 - 4.2.4
2	Catia V5	.CATPart, .CATProduct, .CGR	V5R10 --- V5-6R2025	V5R8--- V5-6R2026
3	Catia V5_2D	.CATDrawing	V5R10 --- V5-6R2025	V5R8--- V5-6R2026
4	3DEXperience (CATIA V6)	.CATPart, .CATProduct	R2010x - R2025x	Exported from V6 R2026x and previous version
5	CGR	.cgr	Exported from V6 R2025x and previous version	Exported from V6 R2026x and previous version
6	NX(UG)	.prt	11 - NX 2506	11 - NX 2506
7	Creo(Pro/E)	.prt, .prt.*, .asm, .asm.*	16 - Creo 12.0	16 - Creo 12.0
8	Creo(Pro/E)_2D	.drw .drw.*	2000i - Creo 12.0	/
9	SolidWorks	.sldprt, .sldasm	98 - 2026 (only support 64-bit)	98 - 2026 (only support 64-bit)
10	SolidWorks_2D	.slddrw	2004 - 2026	2013- 2026 (only support 64-bit)
11	SolidEdge	.par, .asm, .psm	10 - 2026	18 - 2026
12	Inventor	.ipt, .iam	9 to 2026	.ipt (6 - 2026) .iam (11 - 2026)
13	ACIS	.sat, .sab, .asat, .asab	R1 - 2026 1.0	R1 - 2026 1.0
14	Rhino	.3dm	Version 2 - 8	Version 2 - 8
15	CADbro	.Z3D	——	——

【Where is it】

2.2 Supports DWG associative import

The DWX DWG import options introduce an **Associative import** function. During the model design process, when the referenced DWG file is modified, using this function enables synchronous updates, reducing the time required to update the 3D model.

【What users can do】

- ✓ Insert an associative import feature into the history tree.
- ✓ During updates, the incremental parts of geometric objects will be displayed in preview color in the drawing area, making it easy for users to quickly identify design changes in the DWG file.
- ✓ Support redefining the placement coordinates of imported objects, allowing one-click adjustment of the geometric position of referenced objects.

【Notes】

- Associative import only supports 2D geometry in DWG files; 3D models will be filtered out.

【Where is it】

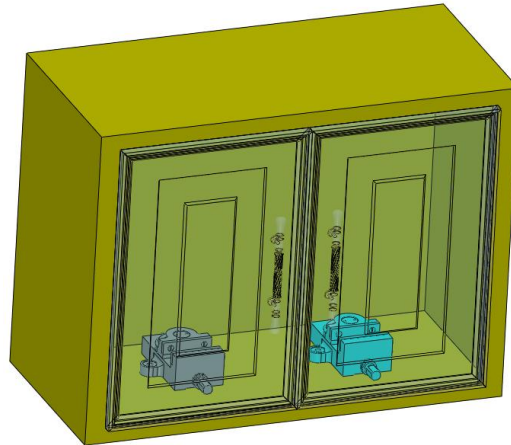
[Data Exchange >> Import >> DWG File >> Associative import](#)

2.3 Simplified Warp

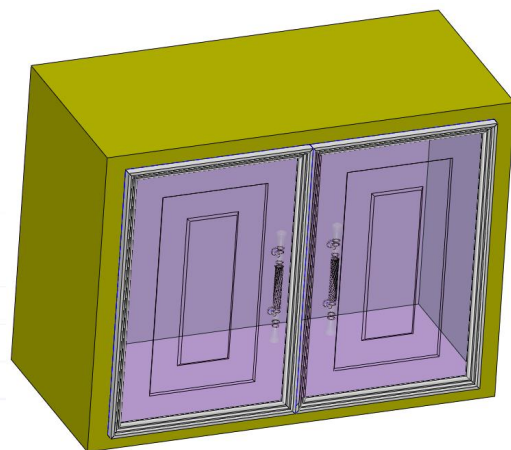
Simplified Warp supports creating the results by extracting only the external surfaces of the model.

【Example】

- 1) Open a cabinet assembly with some internal components that are obscured.



- 2) Create a new part, select this assembly to generate a Simplified Warp, and choose to extract external surfaces. This will ignore the cabinet's inner surfaces and internal obscured components.



【Where is it】

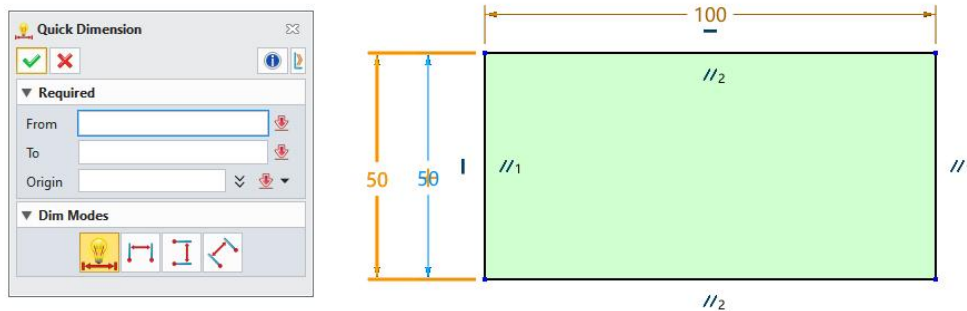
Data Exchange Ribbon >> Simplified Warp

3 CAD

3.1 Sketch & Wireframe Design

3.1.1 Sketch Dimension Optimization

The Sketch Dimension function has been optimized. When activating the dimension command, you can still edit existing dimensions.



【Notes】

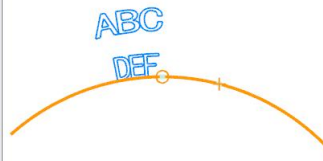
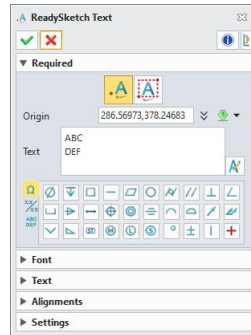
- When the command for editing dimension values is activated, editing existing dimensions is not yet supported.

【Where is it】

Sketch >> Dimension

3.1.2 Sketch Text Optimization

Ready Sketch Text function has been optimized. When Ready Sketch Text is associated with curves, it supports displaying multi-line text.

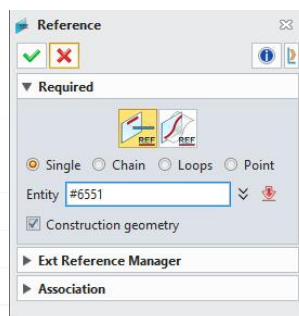


【Where is it】

Sketch >> Ready Sketch Text

3.1.3 Sketch Reference Optimization

Sketch **Reference** function has been optimized. It supports referencing **Ready Sketch Text** in other sketch.



【Notes】

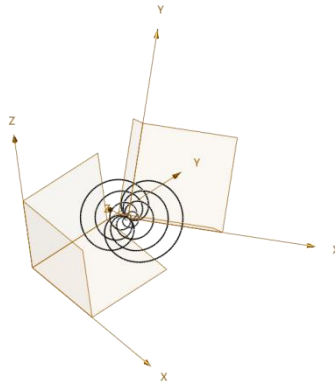
- When the content of the referenced **Ready Sketch Text** changes, the reference cannot locate the object.

【Where is it】

Sketch >> Reference

3.1.4 Wireframe Equation Curve Optimization

Wireframe **Equation Curve** function has been optimized. It supports setting custom reference coordinate systems.



【Notes】

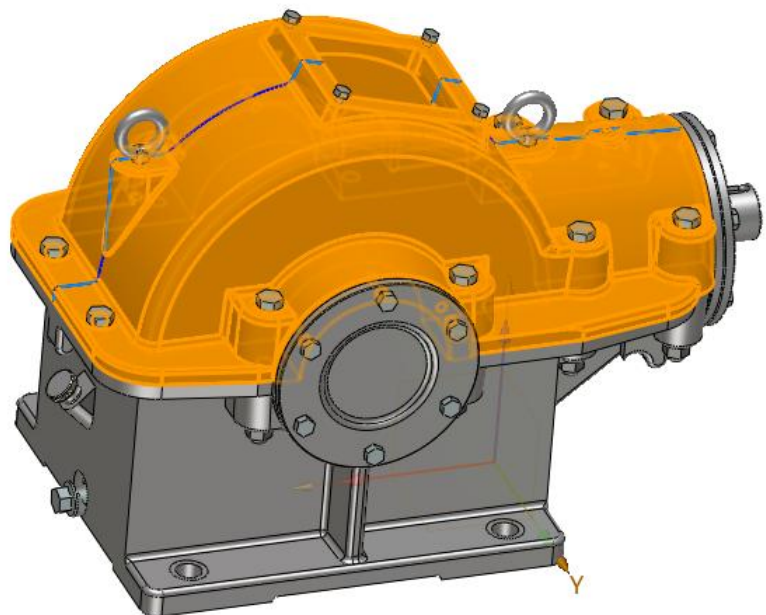
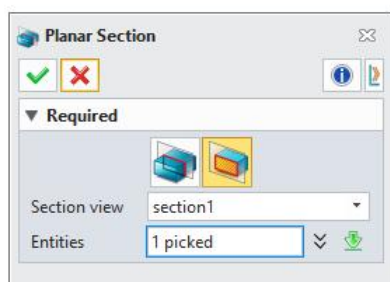
- If no coordinate system is selected, use the default LCS instead.

【Where is it】

Wireframe >> Equation Curve

3.1.5 Planar Section Supports Retrieval from the Section(ZW3D WuKong 2027)

The **Planar Section** command now includes a **From Section** mode, which generates cross-section curves for specified solid objects based on existing section views.

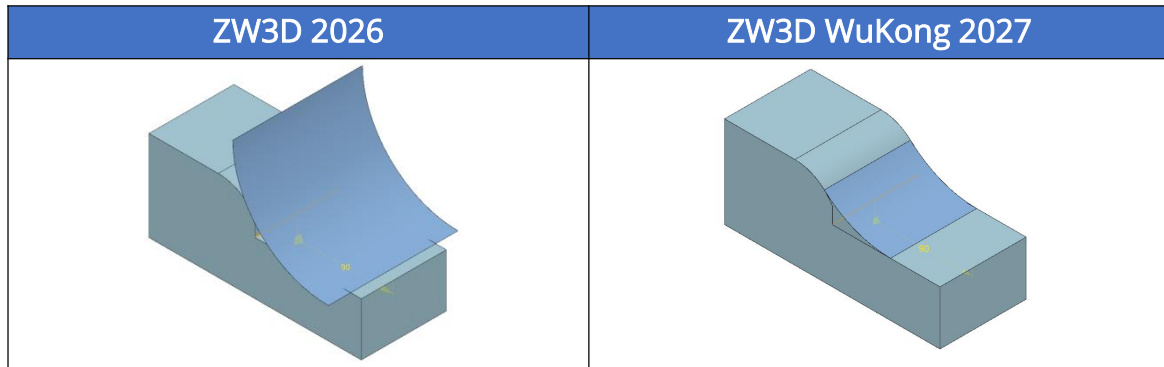


【Where is it】

3.2 Shape Design

3.2.1 Fillet Enhancement

The new version supports creating fillets across steps.



【Notes】

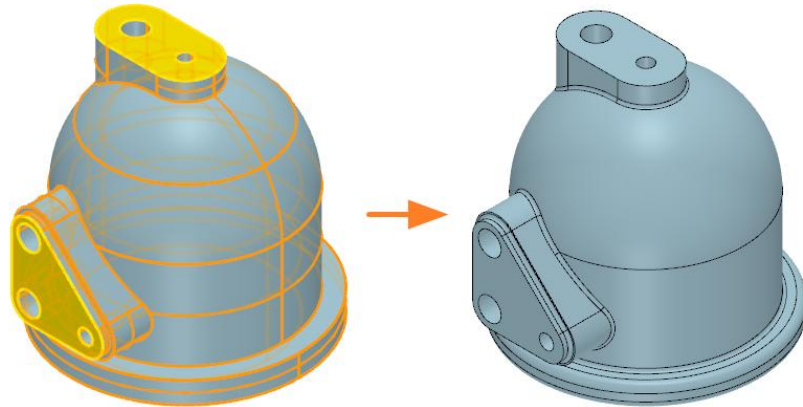
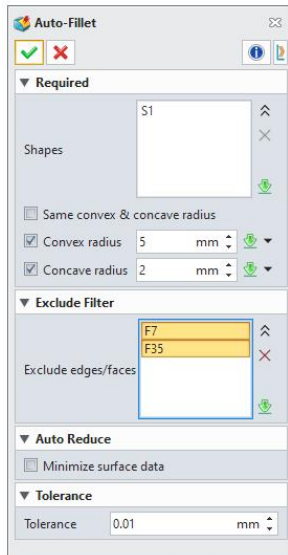
- The function works in the **Automatic** option of **Hold fillet**.

【Where is it】

Part/Assembly Environment >> Shape >> Fillet

3.2.2 ★New Auto Fillet

You can now create fillets quickly on the edges located on solids or surfaces with one step, greatly increase the filleting efficiency.



【What users can do】

- ✓ Convex and concave edges can be filleted with different radius.
- ✓ You can select the edges or faces to be the excluded objects (all edges on the selected faces will be excluded).

【Notes】

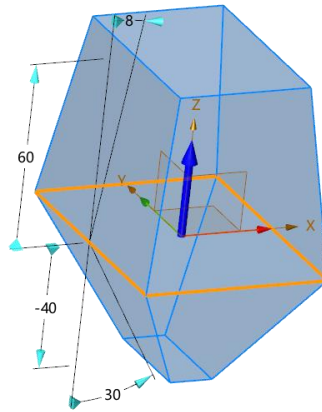
- Only the shape which has at least 1 edge can be create auto fillet.
- Auto fillet feature can't be patterned, mirrored or UDF.

【Where is it】

Part/Assembly Environment >> Shape >> Engineering Feature >> Auto Fillet

3.2.3 New Stationary Setting of Draft in Extrude

The **Extrude** command has added a **Stationary** option where you can set the position of the stationary face for the draft.



【What users can do】

- ✓ Use the starting face, profile plane, and ending face of extrude as the stationary face for drafting.
- ✓ Drafting by different angles on both sides of the profile.
- ✓ Keep the cap faces on both sides the same.

【Notes】

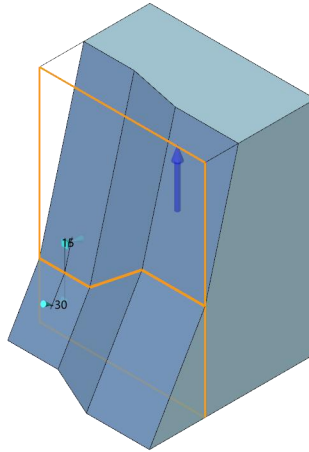
- When only the Start S and the End E are on both sides of the profile, **Profile-asymmetric, Profile-symmetric, Same cap as start** can be used.

【Where is it】

Part/Assembly Environment >> Shape >> Extrude

3.2.4 New Stationary Chain Mode in Draft

The **Draft** command has added **Stationary chain** mode. This mode ensures that draft can be applied to specified faces on the condition that the curve chain lies on the faces.



【What users can do】

- ✓ Draft faces by a curve chain.
- ✓ It can ensure that a continuous curve chain always remains on the draft surface or its extended surface.

【Notes】

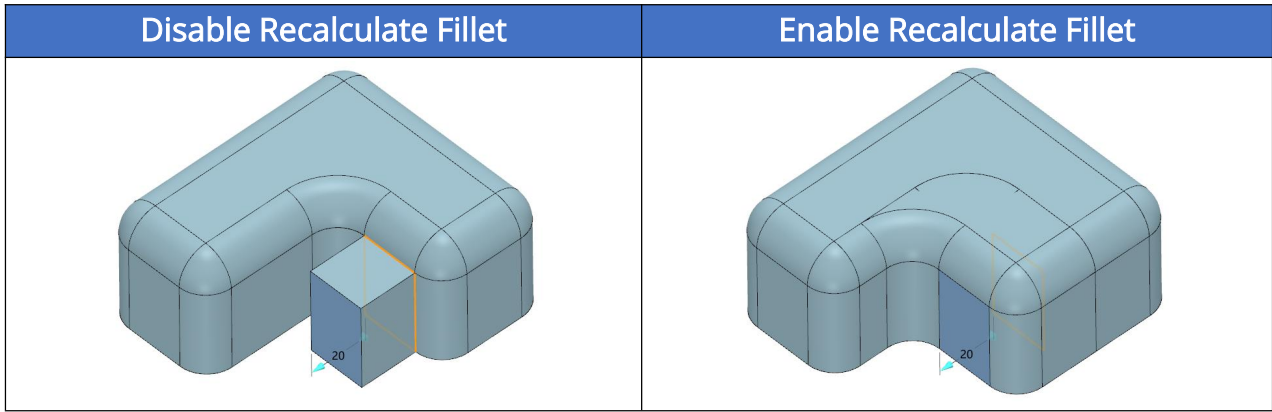
- The inputs of the stationary chain must be at least G0 continuous.
- Edge draft no longer supports curve input, and the same scene is achieved by stationary chain draft.

【Where is it】

Part/Assembly Environment >> Shape >> Draft

3.2.5 New Recalculate Fillet in Face Offset

The **Face Offset** command has added the option of **Recalculate fillet**. This function helps you automatically update adjacent fillets when performing face offset operations.



【Notes】

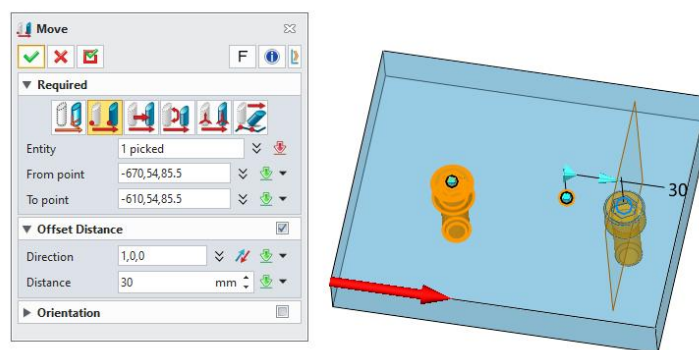
- The fillet faces will only be recalculated when they are adjacent. If the fillet surfaces are selected as the offset faces, They will not be recalculated because they will be treated as Offset rather than Recalculated.

【Where is it】

Part/Assembly Environment >> Shape >> Offset face

3.2.6 Move/Copy Function Improvement

The Point-to-point mode of the **Move/Copy** command adds an **Offset Distance** column, which allows designers to quickly move/copy objects to the offset Distance position in the specified direction.



【Where is it】

Part/Assembly Environment >> Shape >> Basic Editing >> Move/Copy

Part/Assembly Environment >> Free Form >> Basic Editing >> **Move/Copy**

Part/Assembly Environment >> Wireframe >> Basic Editing >> **Move/Copy**

Part/Assembly Environment >> Electrode >> Basic Editing >> **Move**

Assembly Environment >> Assembly >> Basic Editing >> **Move**

3.2.7 Align Move Supports Picking Datum/Axis/CSYS(ZW3D WuKong 2027)

Align Move command in new version supports picking datum plane/axis/coordinate system(CSYS) and wireframe object for movement operation.

【Where is it】

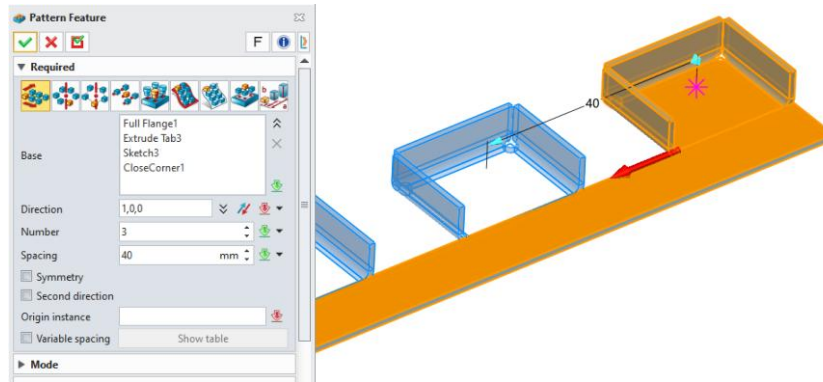
Part/Assembly Environment >> Shape >> Basic editing >> **Align Move**

3.2.8 Pattern/Mirror feature Supports More Feature(ZW3D WuKong 2027)

More features support to be added to pattern/mirror feature in new version. Including the following commands: Boolean operation (**Add/Remove/Intersect**), **Face offset**, **DE face offset**, **DE move**, **Sew**, **Close Corner**. Compared to pattern geometry, the biggest advantage of pattern/mirror feature is that they allow variable pattern, which can change instances' parameters and enable regular changes in the shape of pattern/mirror.

【Example】

- 1) When mirror/pattern newly supported features, their base features must also be included in current mirror/pattern. Take those features require boolean operation as an example, their parent features must also be added to mirror/pattern.



【Where is it】

Part/Assembly Environment >> Shape >> Basic editing >> **Pattern Feature**

Part/Assembly Environment >> Shape >> Basic editing >> **Mirror Feature**

3.2.9 At Pattern Mode Supports Define Original Instance(ZW3D WuKong 2027)

At **pattern** mode in pattern feature/geometry supports user to manually specify **original instance** in new version. The newly instance in pattern can be matched to the specified instance (The referenced pattern) according to user input.

【What users can do】

✓ Specify **original instance** to align the newly pattern instance so as to align with the referenced pattern according to the user's intention.

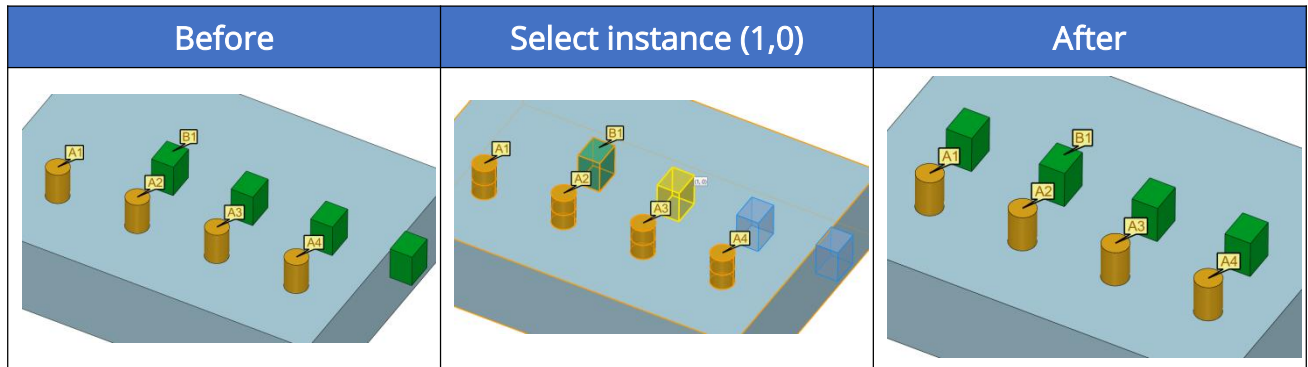
【Notes】

- Pattern may fail if specifying **original instance** manually when some instances are dangling.

【Example】

1) As shown in the figure below, feature Extrude(B1) aligns with the first instance(Cylindrical extrude A1) of the referenced pattern (A1~A4) before specified the **original instance**.

- 2) If you want to specify instance B1 align with 2nd instance(A2) of the referenced pattern. You need to pick a new **original instance** to select the 2nd newly created instance(B2) which displays as (1,0) in the pattern echo. Then the result of pattern will be adjusted to the referenced pattern better.



【Where is it】

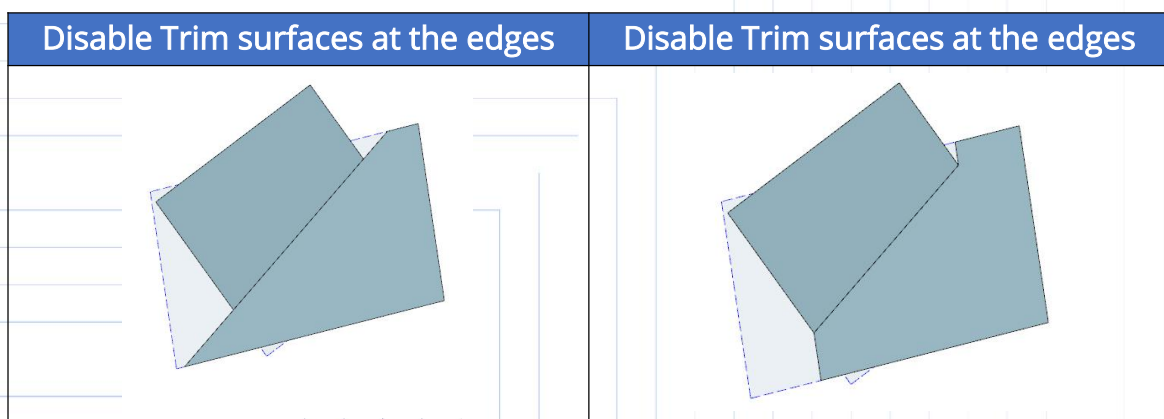
Part/Assembly Environment >> Shape >> Basic editing >> **Pattern Feature**

Part/Assembly Environment >> Shape >> Basic editing >> **Pattern Geometry**

3.3 Surface Design

3.3.1 New Trim Surface at Edges in Cross Trim

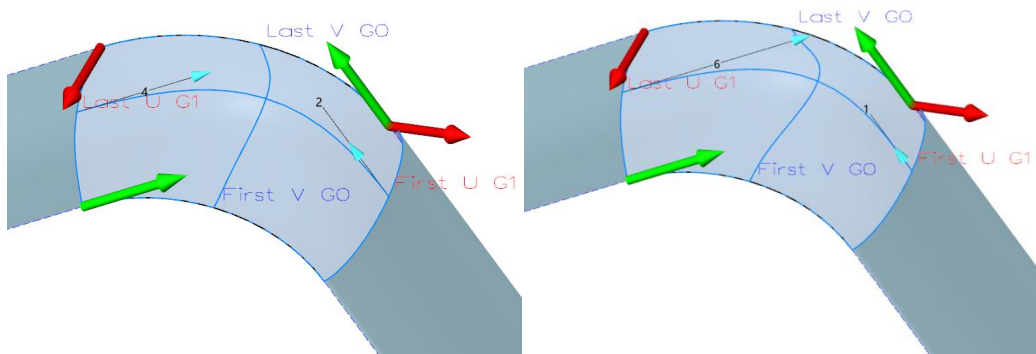
The **Cross Trim** command has added the option of **Trim surfaces at the edges**. This function can help designers perform truncation trimming near the edge and cross trimming on the surface when the surface cannot be completely trimmed to the edge.



【Where is it】

3.3.2 New Weight Control in U/V Surf

The **U/V Surf** command has added the option of **Weight**. This option can help you adjust the degree of influence of boundary continuity, thereby achieving a more aesthetically pleasing surface effect.



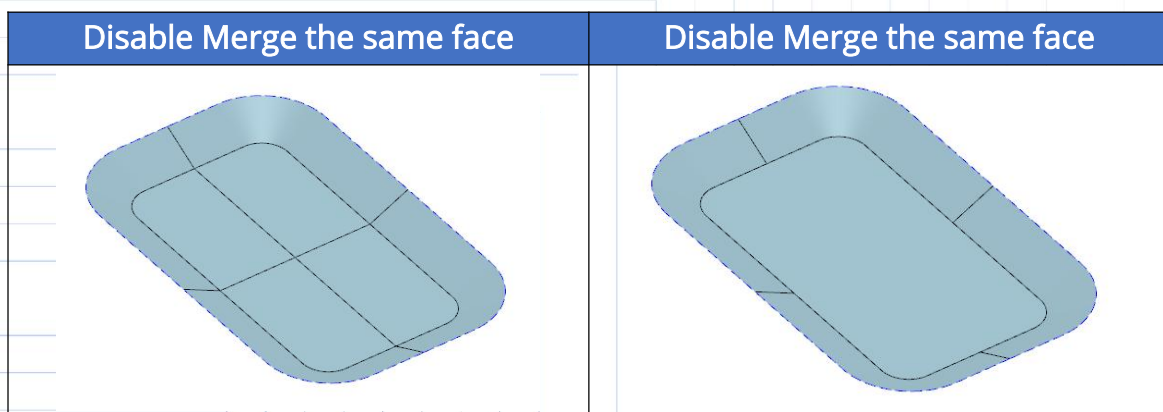
【Notes】

- The weight only takes effect when the boundary continuity is G1 or higher.

【Where is it】

3.3.3 New Merge The Same Face in Sew

The **Sew** command has added the option of **Merge the same face**. This option can help you effectively reduce surface breakage after Sew operations.



【Notes】

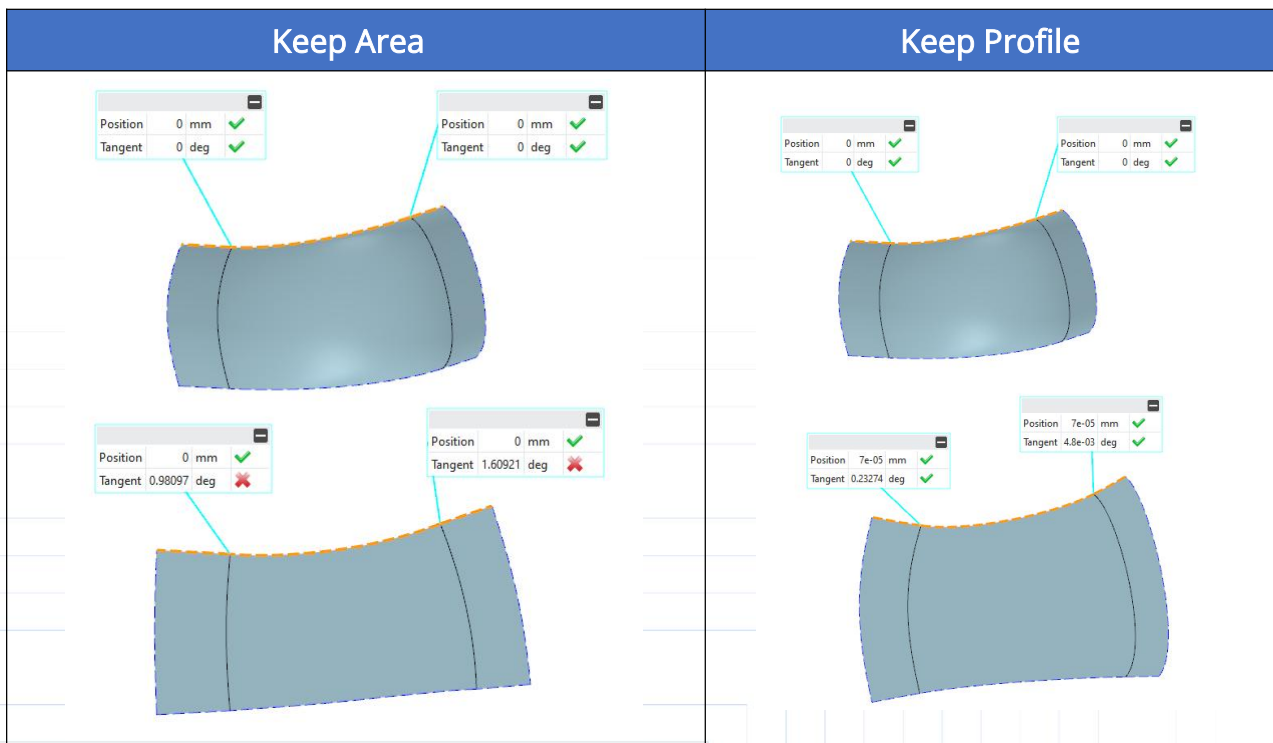
- Only co-planar planes, concentric cylindrical surfaces, and other surfaces of the same type will be merged.

【Where is it】

Part/Assembly Environment >> Free Form >> Sew

3.3.4 New Keep Profile Mode in Unfold Surf

The Unfold surf command has added Keep profile which will try to maintain the boundary quality of the contour as much as possible.



【Notes】

- For curved surfaces with large curvature, the deviation of the new mode area of keeping profile will be relatively large.

【Where is it】

Part/Assembly Environment >> Free Form >> Unfold Surf

3.4 Assembly Design

3.4.1 Layout-Based Top-Down Design

The **Master Layout** now allows you to freely select any external object as a reference according to your design intent. It will accommodate a wider range of application scenarios.

【What users can do】

- ✓ When creating features such as Reference, Copy Geom, External Part features in the Master Layout model, other parts can be selected.
- ✓ When creating modeling features such as Datum, Sketch, and Extrude, the Master Layout model can directly reference other parts.

【Notes】

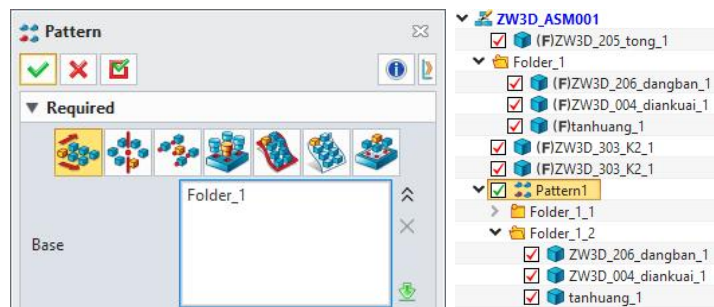
- Since there are no restrictions on reference selection, you need to be mindful of your actions when creating references to avoid unexpected regeneration results.

【Where is it】

[Assembly Environment >> New to Insert](#)

3.4.2 Assembly Pattern

The assembly pattern has optimized the generation results when the input is a component folder. After patterning, the internal features will also manage components in the form of folders.

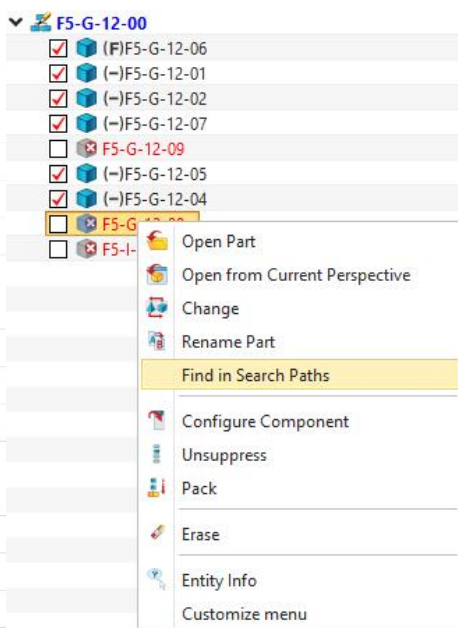


【Where is it】

Assembly Environment >> Pattern

3.4.3 Component Lookup Path Optimization(ZW3D WuKong 2027)

In assembly design, for missing components, you can directly right-click on them, select "Find in Search Paths" from the context menu, and add the file path to retrieve them.



【Where is it】

Assembly Node

3.5 Drawing Sheet

3.5.1 Optimization of the automatic drawing function

The **Auto Drafting** function adopts a brand-new UI design. Its interactive interface and operation workflow have been fully optimized, making it more intuitive and user-friendly. Meanwhile, dimensioning effects are further improved to better meet user requirements.

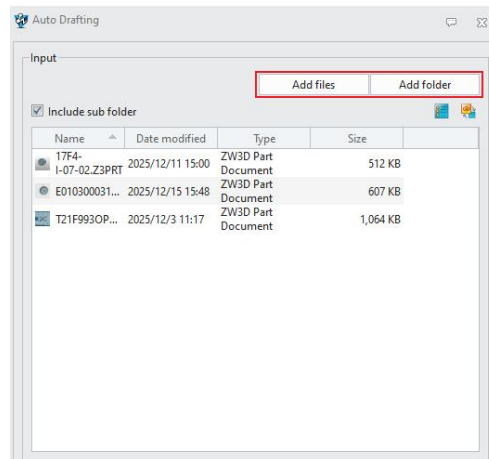
【What users can do】

- ✓ Supports setting different templates for part and assembly models, which are automatically matched during AutoDrafting.
- ✓ Supports various dimension types including linear, baseline, continued and ordinate dimensions.
- ✓ Supports quickly AutoDrafting for single files and batch files. For batch processing, files can be added in batches via folders.
- ✓ Added dimensioning support for parametric chamfers, fillets and circular hole patterns.
- ✓ Automatically generates a section view passing through the axis for revolved bodies.
- ✓ Supports invoking section views in the 3D environment to automatically generate drawing section views.
- ✓ For parts not aligned with the coordinate system, it supports automatic alignment of 3D models to generate standard views.
- ✓ Supports automatic generation of sheet metal flat patterns for unfoldable sheet metal parts.
- ✓ Supports multi-view projection and automatic removal of duplicate dimensions.

【Examples】

Brief introduction to the automatic drawing generation workflow for batch files:

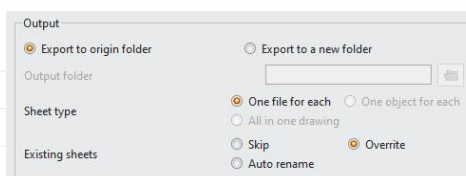
- 1) Enter the part environment, launch the Auto Drafting function, and add files to be drafted.



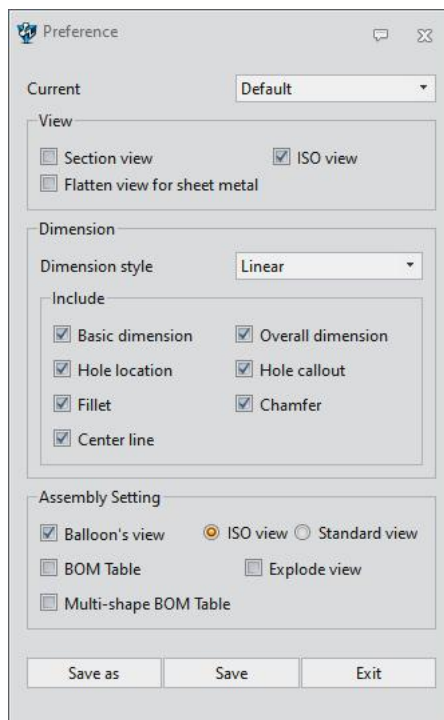
- 2) Configure drawing templates for part and assembly files respectively.



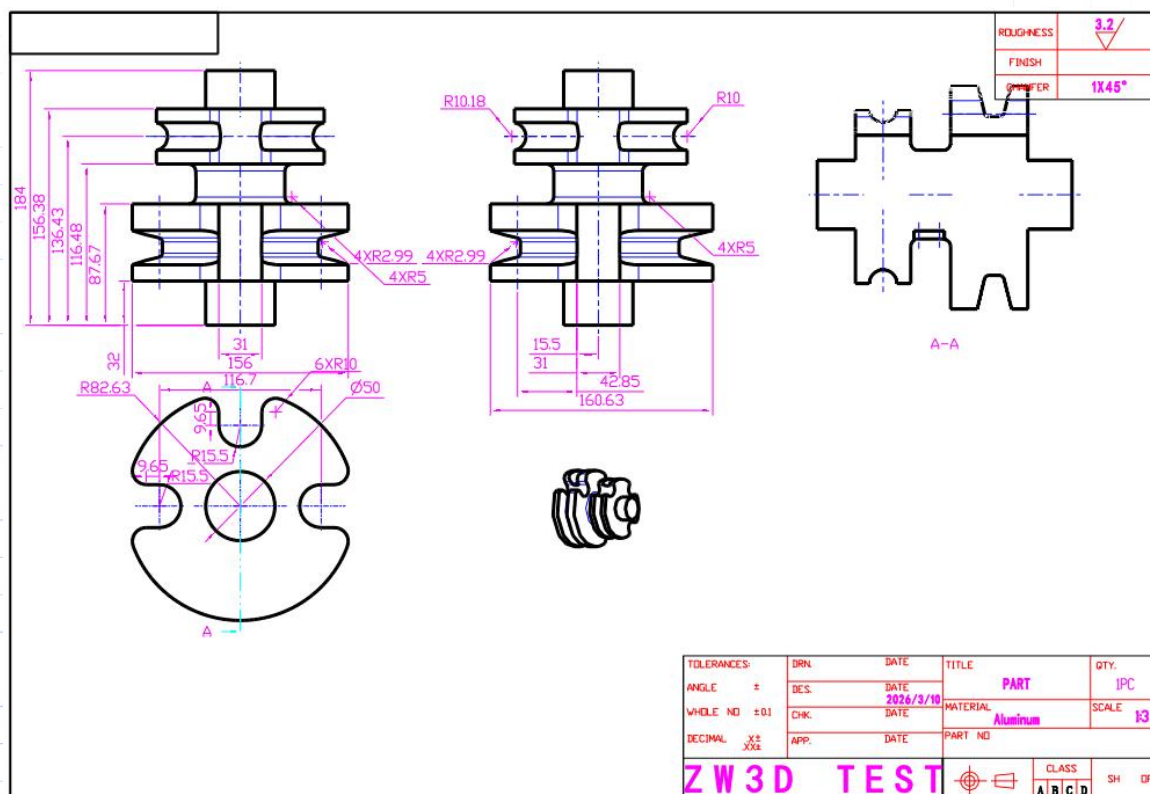
- 3) Set the output folder and drawing generation mode.



- 4) Configure dimension elements, dimension styles, special views and tables.

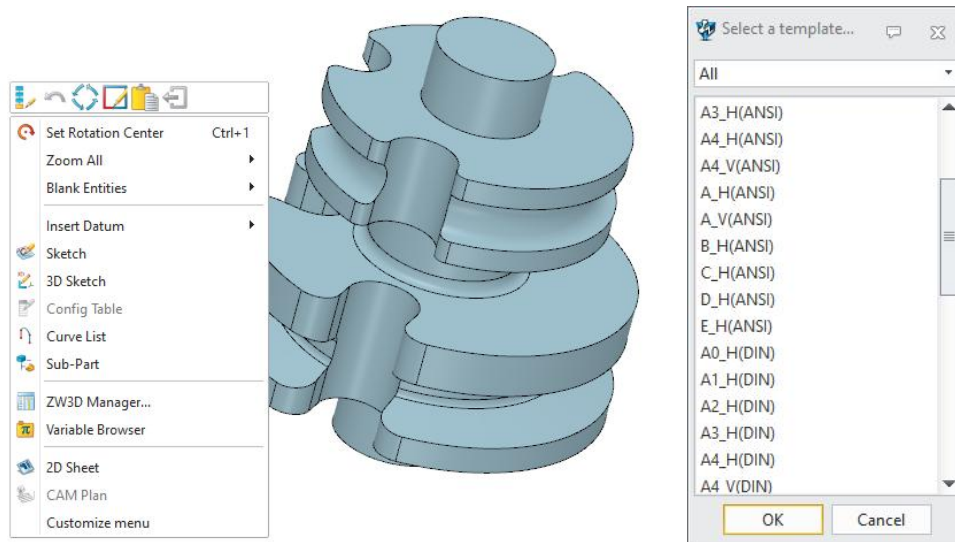


- 5) Confirm to realize automatic drafting generation. The output drawings include views in multiple directions. A section view passing through the axis will be automatically generated for revolved bodies, and all views will be dimensioned automatically.



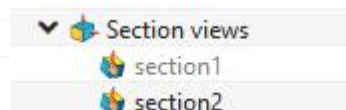
Brief overview of the fast automatic drawing generation workflow for single files:

- 1) In the part environment, right-click the blank area to directly launch automatic drawing generation for the current model. Select a template and the drawing can be completed quickly.

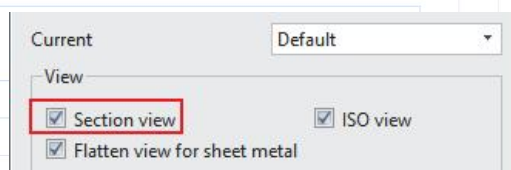


Invokes sections in the 3D environment to automatically generate section views:

- 1) Create multiple section views in the 3D environment.



- 2) Select section views in the Auto Drafting settings.



- 3) Execute automatic drawing generation, and corresponding drawing section views will be created for all 3D sections.



【Notes】

- View layout area, BOM table placement position and other parameters can all be configured in the drawing template.
- View layout, projection direction and drawing scale settings in the Style Manager are not applicable to the AutoDrafting function. The system automatically matches fixed view layouts according to the number of projections, intelligently selects projection directions based on model features, and automatically calculates the drawing scaling ratio according to the number of views and the view range defined in the template.
- Fillets and chamfers are only applicable to parametric models. Dimensioning of non-parametric holes is recognized and generated based on the built-in hole dimension command.
- Dimensions are only supported for horizontal and vertical plane projections. Angular dimension generation is not currently supported (except for pattern features).
- Only unfoldable sheet metal parts are supported for automatic generation of

sheet metal flat patterns.

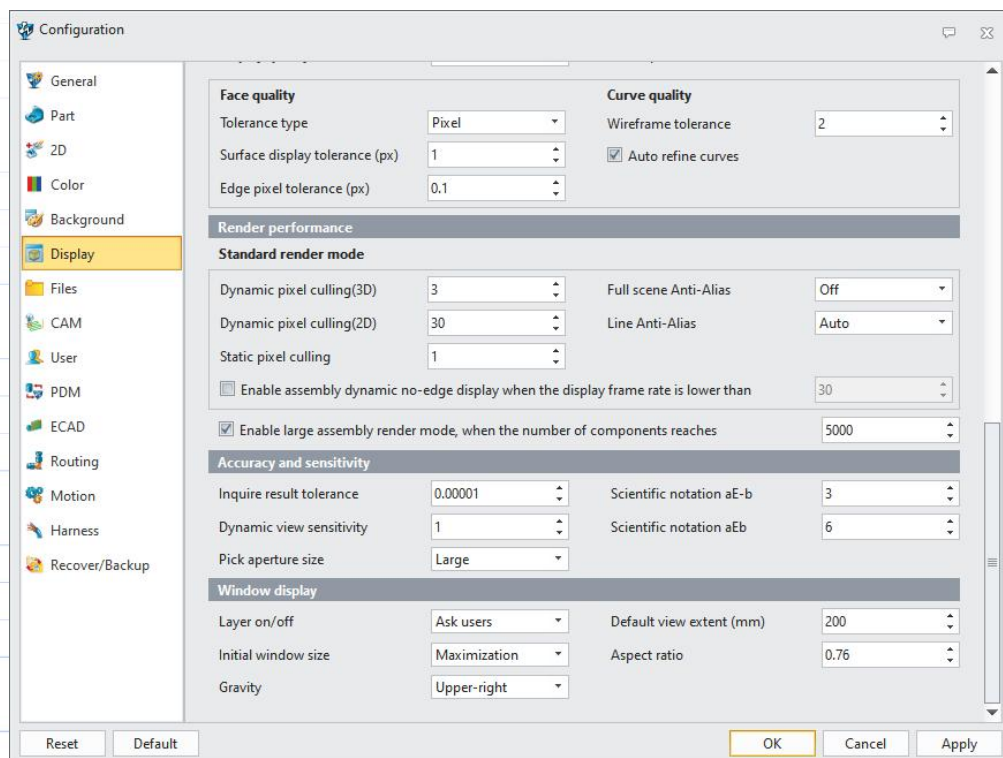
- Only revolved parts and 3D models with predefined section views support automatic section view generation.
- Individual drawing generation requires a separately specified template and does not inherit the batch drawing template. All other drawing settings remain consistent with batch settings.

【Where is it】

Part & Assembly>>Tools>> Insert>>Auto Drafting

3.5.2 New Dynamic Pixel Culling(2D)

Dynamic pixel culling(2D) has been added to the configuration, which supports setting the dynamic culling pixel value for the drawing module separately. By configuring this option, operations such as panning and zooming in complex views can be made smoother.

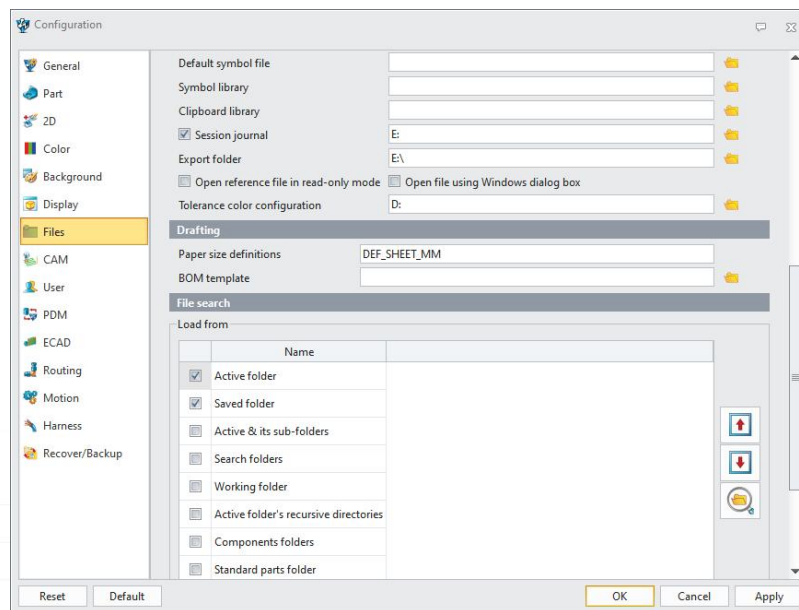


【Where is it】

Configuration >> Display >> Render performance >> Dynamic pixel culling(2D)

3.5.3 Symbol Library Optimization

The **Symbol Library** now supports configuring a default path. In conjunction with function **Insert Block**, it can manage all blocks in multiple files within that path simultaneously.



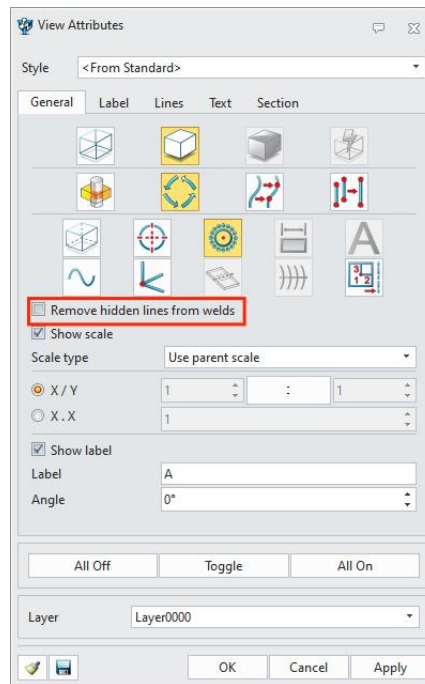
【Where is it】

Configuration >> Files >> Default symbol file

Configuration >> Files >> Symbol library

3.5.4 Weld Display Optimization

The **View Attributes** has add a new option of **Remove hidden lines from welds**, allowing for more precise control over the display of welds.

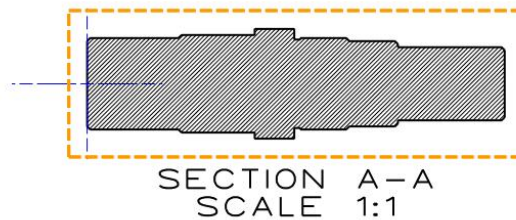
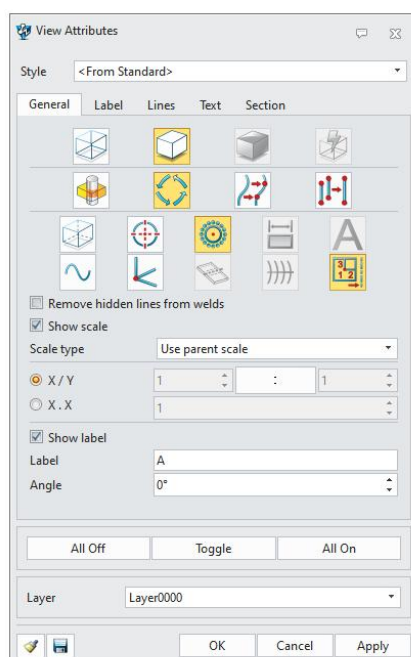


【Where is it】

Drawing Environment >> View Attributes >> Remove hidden lines from welds

3.5.5 Section View Optimization

Section views support the display of projection datum and can automatically inherit the projection datum display properties from their parent view.

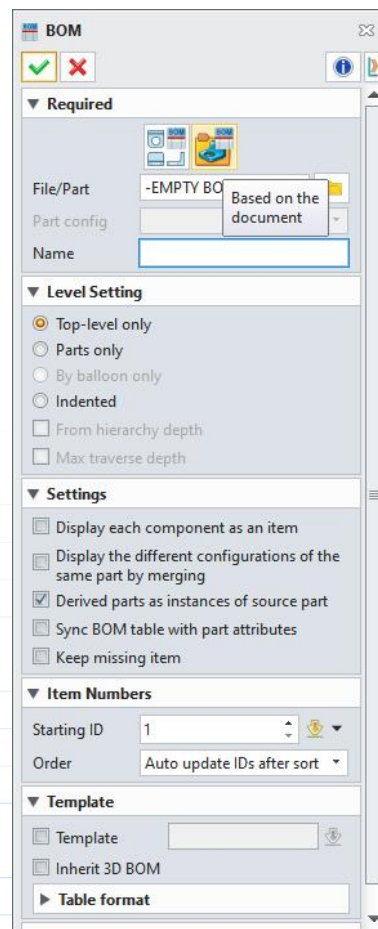


【Where is it】

Drawing Environment >> View Attributes >> Show projected datum

3.5.6 BOM Table Optimization

The BOM table has added **Based on the document** mode, which supports selecting a 3D model file to create a BOM table directly. With this mode, a BOM table can be created directly without any views.

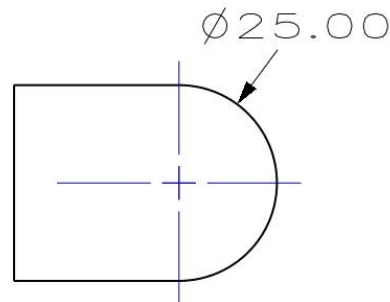
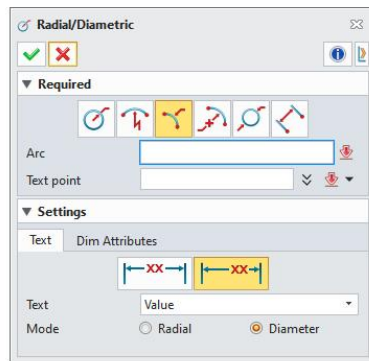


【Where is it】

Drawing Environment >> Layout >> Table >> BOM

3.5.7 Radial/Diametric Dimension Optimization

Radius/Diameter dimensions can be directly created in a diameter style without extension lines.

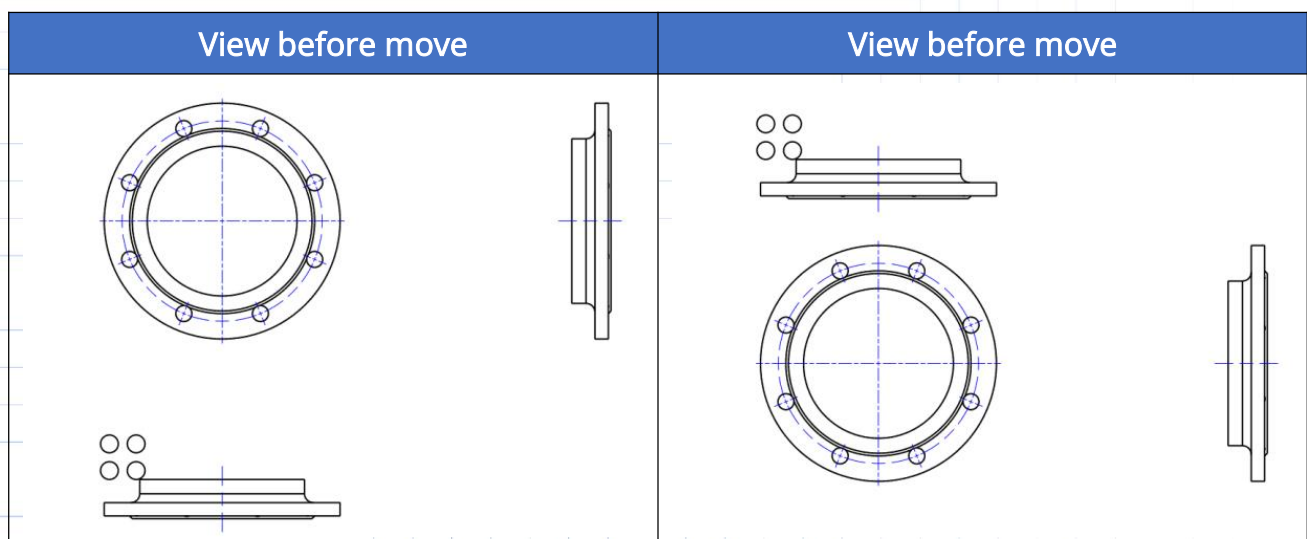


【Where is it】

Drawing Environment >> Dimension >> Radial/Diametric >> Large radial dimension >> Radial/Diametric

3.5.8 Optimization of Entity Association(ZW3D WuKong 2027)

Primitive entities associated with a view, as well as all primitive objects generated from them via operations such as array, copy, mirror, scale, and rotate, shall be associated with the view in their entirety.

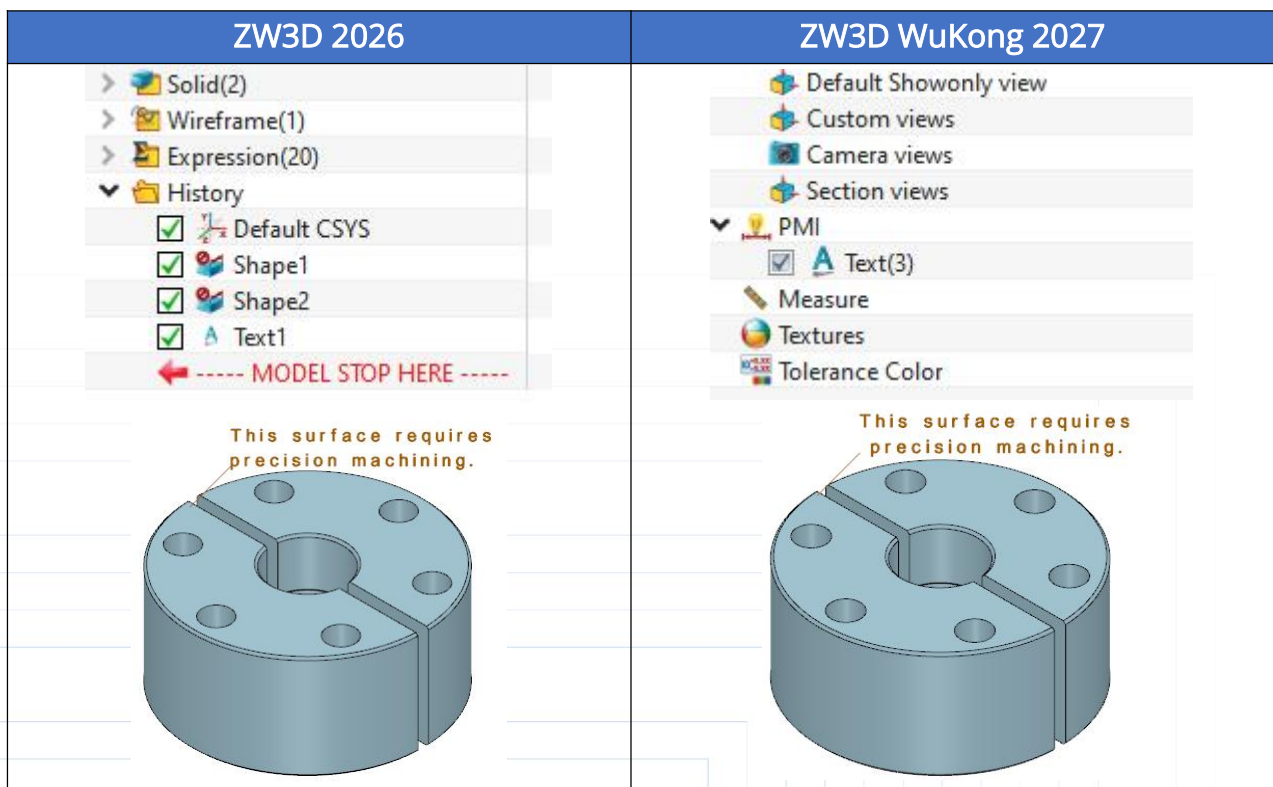


【Where is it】

3.6 PMI

3.6.1 Text, Text balloon Optimization

The new version optimizes the generation node of PMI module **Text** and **Text Balloon** on the manager. The display has been adjusted from the history tree under the original History Manager to the PMI tree under the View Manager, significantly enhancing the efficiency of redefining PMI **Text** and **Text Balloon**.



【Where is it】

PMI>> Text >>Text

PMI>> Text >>Text Balloon

3.6.2 Modify PMI Behavior Optimization

The new version optimizes the modification behavior of **PMI linear, Symmetry, Angular, Radial/Diametric, Chamfer, Arc Length, Coordinate, Note, Datum Feature, Datum Target, Feature Control**, etc. You can double-click on the above objects to achieve PMI modifying and editing, which greatly improves the efficiency.

【Where is it】

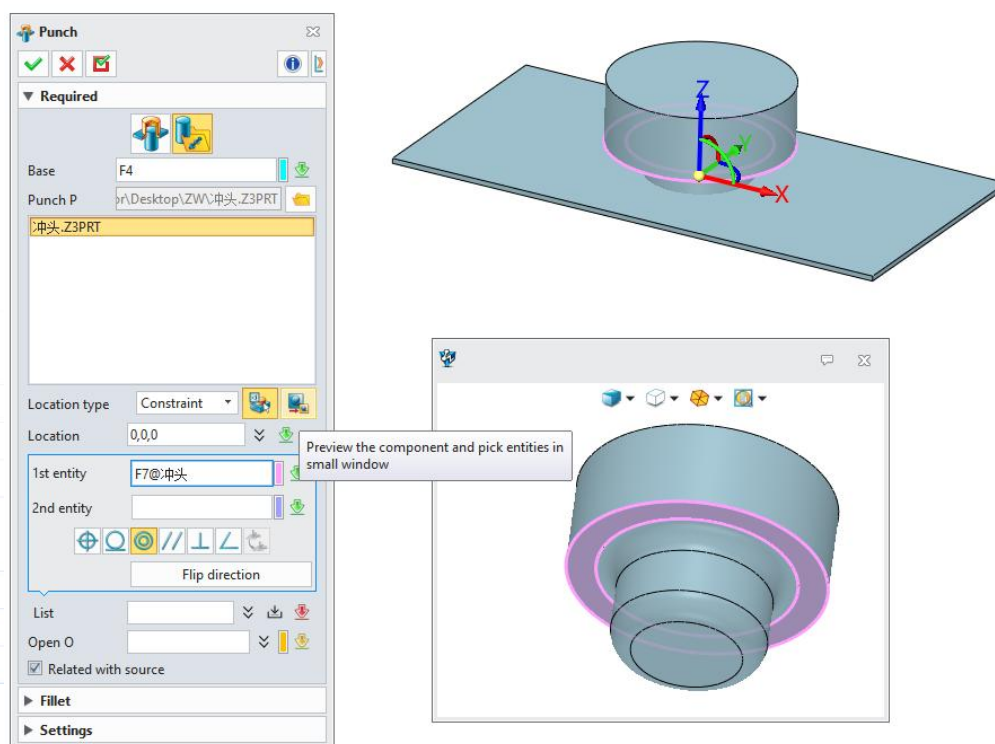
ZW3D drawing area>> **double-clicking PMI objects**

4 Industry Applications

4.1 Sheet Metal Design

4.1.1 Punch Optimization

During the Punching process, when applying **Constraint** positioning to the external punch model, the new version provides a small window preview function and allows users to select entities for constraints, thereby significantly enhancing operational convenience.

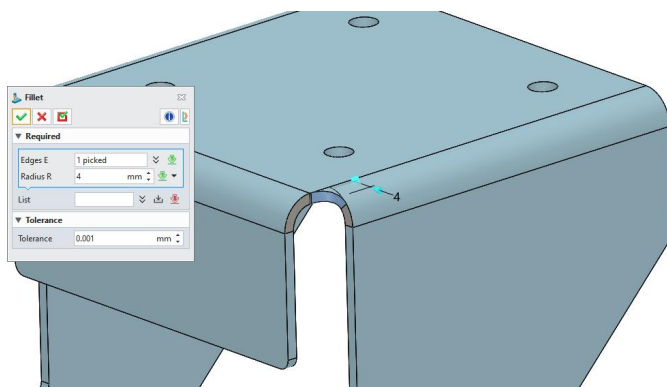


【Where is it】

Sheet Metal >> Form >> Punch

4.1.2 New SMT Fillet

The new version has added a sheet metal-specific **Fillet** function, supporting unfolding or folding operations with fillets.

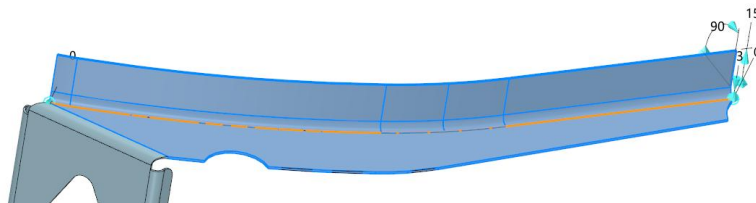


【Where is it】

Sheet Metal >> Basic Editing>> Fillet

4.1.3 Add Flange To Spline Edge(ZW3D WuKong 2027 EA2)

The "Flange" and "Full Flange" features have been enhanced. You can now add flanges to spline edges and unfold or fold the added flanges

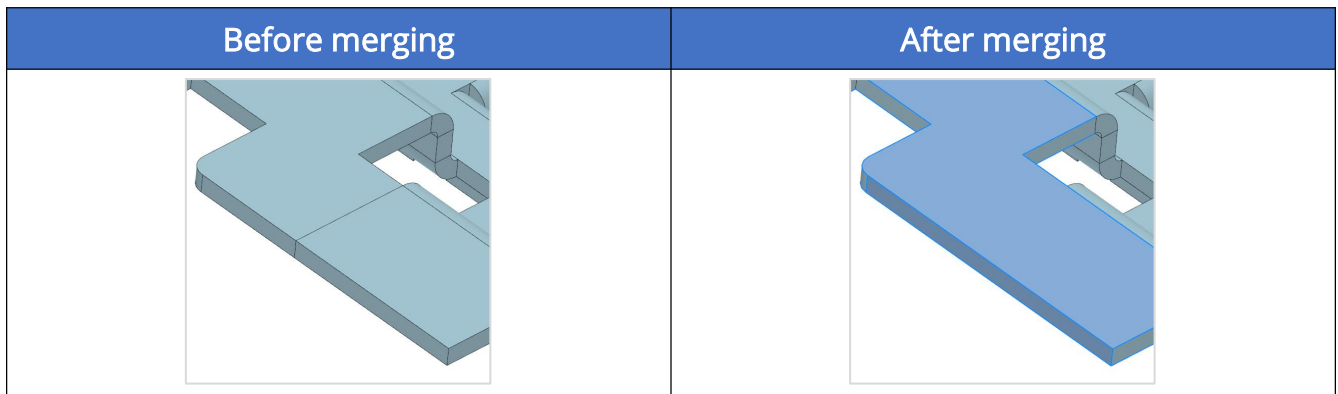


【Where is it】

Sheet Metal >> Flange >> Flange/Full Flange

4.1.4 New Merge Flange(ZW3D WuKong 2027 EA2)

Add the "Merge Flange" feature to combine two or more separate tangent flanges or bends.

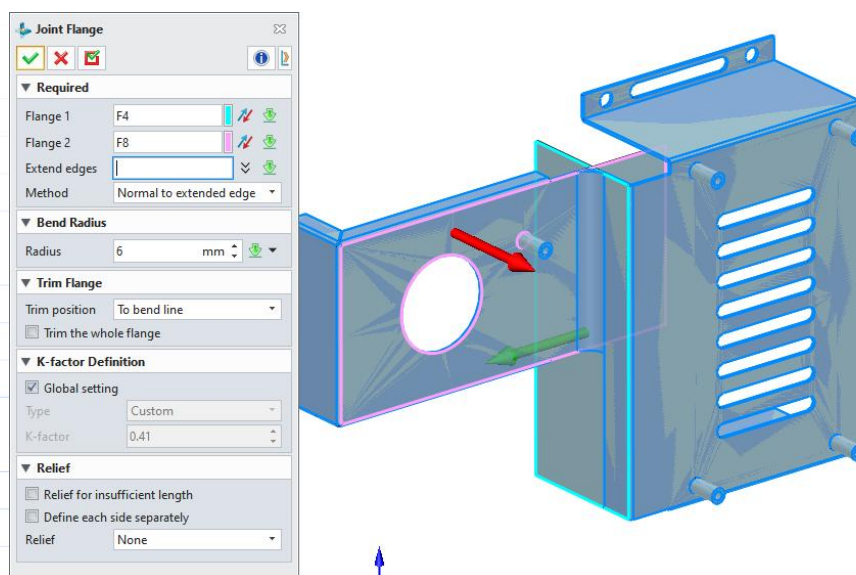


【Where is it】

Sheet Metal >> Edit >> Merge Flange

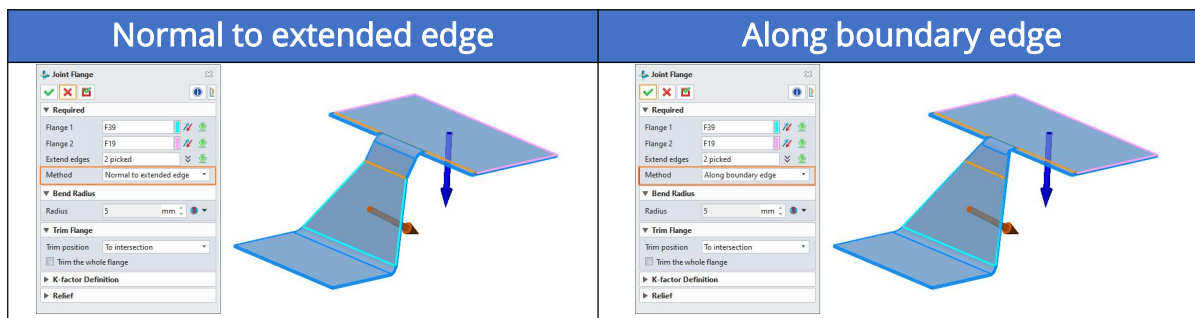
4.1.5 New Joint Flange

Add the "Joint Flange" function to join separated flanges and add bends and relief at the connection points. You can connect flanges that intersect directly or after extension, making sheet metal design more flexible.

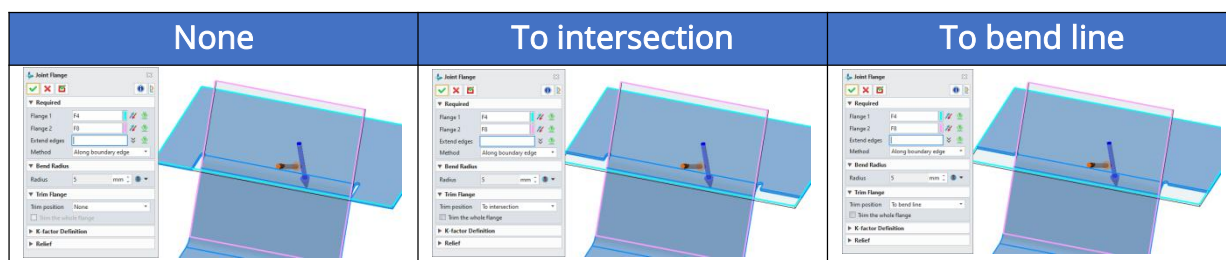


【What users can do】

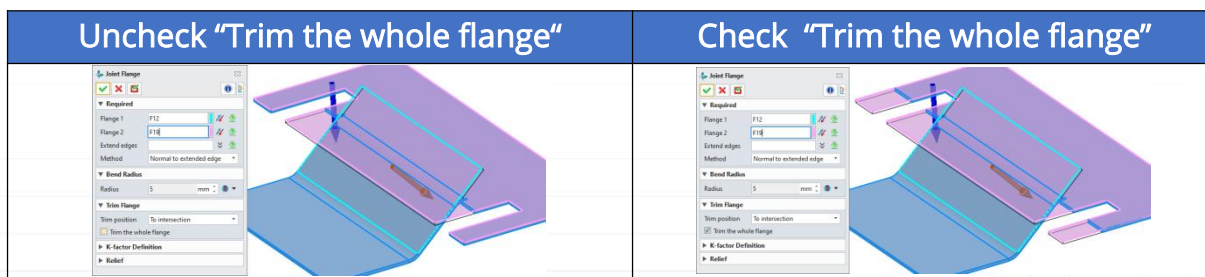
When flanges do not intersect directly, they can be extended to meet each other, with two extension methods available: "Normal to extended edge" and "Along boundary edge".



✓ Support trimming non-intersecting flange sections with two methods: “To intersection” and “To bend line”.



✓ Trim all non-intersecting sections along the trim edge.



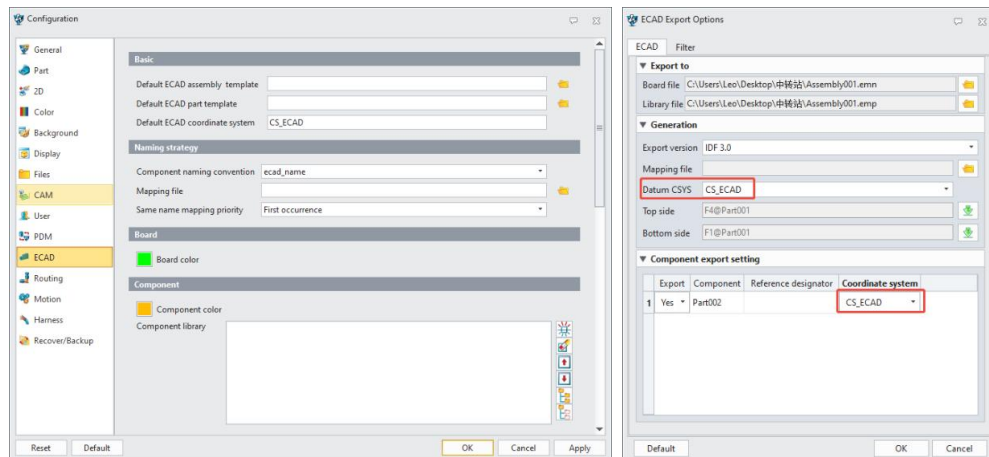
【Where is it】

Sheet Metal >> Edit >> Joint Flange

4.2 ECAD

4.2.1 ECAD Configuration Optimization

In the new version, after setting the **Default ECAD coordinate system** parameters in the **Configuration >> ECAD** section, these parameters will be applied to both the **Datum CSYS** and the component **Coordinate System** in the **ECAD Export Options**. This will enhance user convenience during the EMN/EMP export process and reduce the likelihood of coordinate setting errors.



【Where is it】

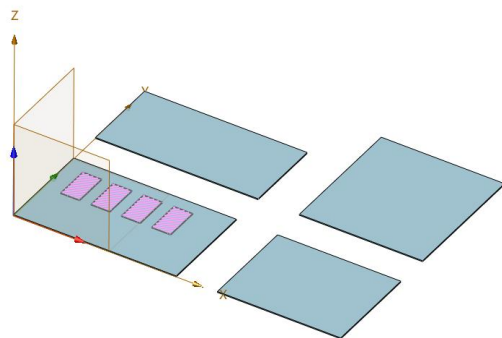
Configuration >> ECAD >> Basic

4.2.2 Export EMN/EMP File Optimization

The new version has optimized the EMN/EMP file **Export** function, supporting for exporting multiple unconnected boards or boards with loop regions.

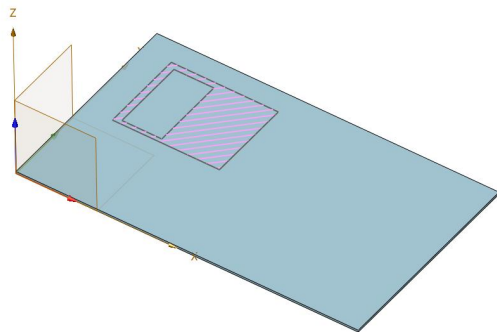
【Example】

1) Support for exporting multiple boards.



```
.HEADER
BOARD_FILE 3.0 "中望3D 悟空 2027 先锋版 Beta3 x64 "
"Part001" MM
.END_HEADER
.BOARD_OUTLINE ECAD
1.000000
0 201.429773 128.730111 0.000000
0 323.192291 128.730111 0.000000
0 323.192291 257.429779 0.000000
0 201.429773 257.429779 0.000000
0 201.429773 128.730111 0.000000
1 0.000000 128.730111 0.000000
1 160.714279 128.730111 0.000000
1 160.714279 205.949921 0.000000
1 0.000000 205.949921 0.000000
1 0.000000 128.730111 0.000000
2 0.000000 0.000000 0.000000
2 160.714279 0.000000 0.000000
2 160.714279 94.581284 0.000000
2 0.000000 94.581284 0.000000
2 0.000000 0.000000 0.000000
3 201.429773 0.000000 0.000000
3 323.192291 0.000000 0.000000
3 323.192291 94.581284 0.000000
3 201.429773 94.581284 0.000000
3 201.429773 0.000000 0.000000
.END_BOARD_OUTLINE
```

2) Support for exporting loop regions.



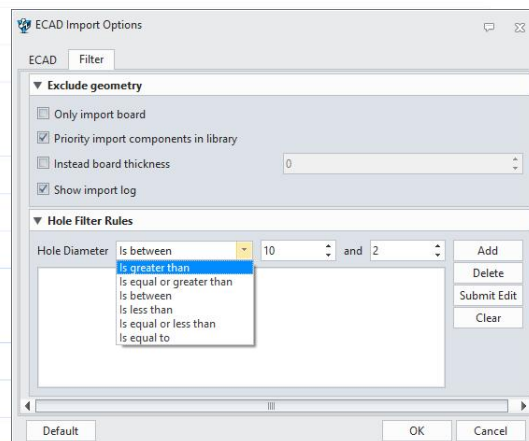
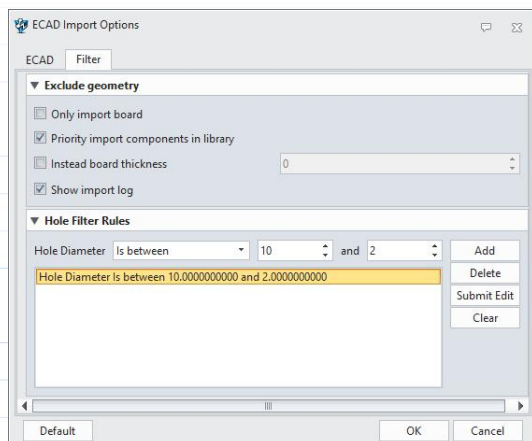
```
.HEADER
BOARD_FILE 3.0 "中望3D 悟空 2027 先锋版 Beta3 x64 "
"Part001" MM
.END HEADER
.BOARD_OUTLINE ECAD
1.000000
0 0.000000 0.000000 0.000000
0 160.714279 0.000000 0.000000
0 160.714279 94.581284 0.000000
0 0.000000 94.581284 0.000000
0 0.000000 0.000000 0.000000
.END_BOARD_OUTLINE
#MULTIPLE_LOOP_REGION
.ROUTE_OUTLINE MCAD
TOP
0 17.179796 81.527092 0.000000
0 35.899014 81.527092 0.000000
0 35.899014 46.551727 0.000000
0 17.179796 46.551727 0.000000
0 17.179796 81.527092 0.000000
.END_ROUTE_OUTLINE
.ROUTE_OUTLINE MCAD
TOP
0 14.000949 84.254379 0.000000
0 64.255348 84.254379 0.000000
0 64.255348 44.692402 0.000000
0 14.000949 44.692402 0.000000
0 14.000949 84.254379 0.000000
.END_ROUTE_OUTLINE
#END_MULTIPLE_LOOP_REGION
```

【Where is it】

Configuration >> ECAD >> Basic

4.2.3 Import EMN/EMP File Optimization

When importing EMN/EMP files, you can filter out unwanted holes through multiple methods to improve import efficiency. These methods include **Is greater than**, **Is greater than or equal to**, **between**, **less than**, **Is equal or less than**, and **Is equal to**.

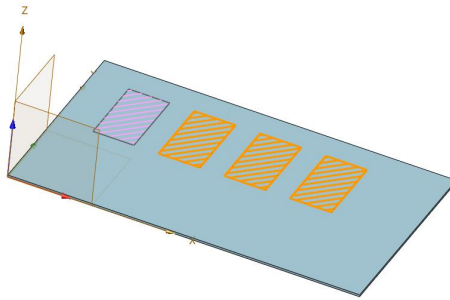


【Where is it】

ECAD >> Data Exchange >> Import

4.2.4 Support Pattern Region Feature

When creating multiple regions with the same shape and type, you can utilize pattern features to generate them quickly, thereby significantly improving design efficiency.



【Where is it】

ECAD >> Pattern Feature

5 CAM(ZW3D WuKong 2027)

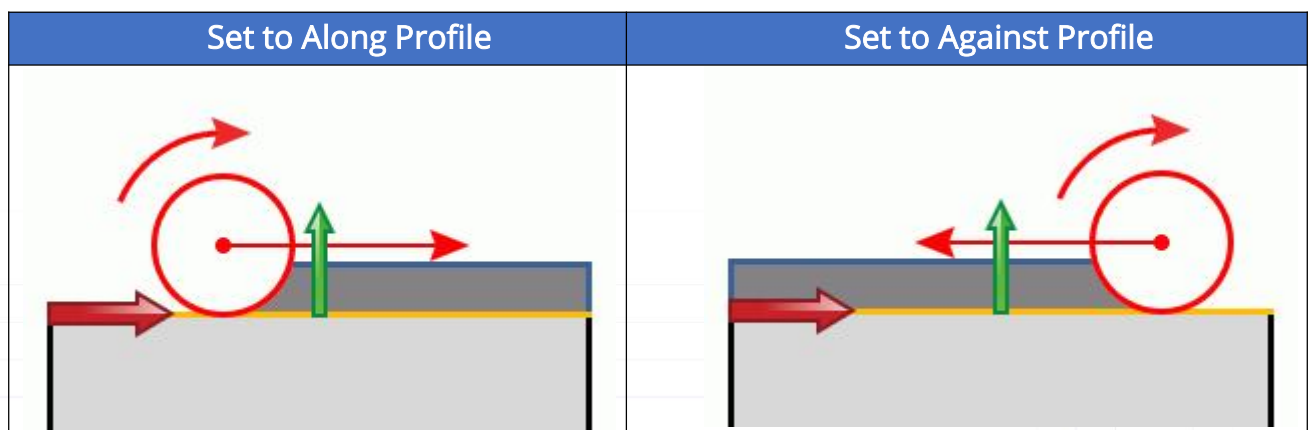
5.1 2x Mill

5.1.1 Profile Cut Enriches Cutting Direction Options

2-axis Profile Cut's cutting direction option is supplemented with **Along Profile** and **Against Profile** options, which enable the tool to perform cutting either in alignment with the profile feature direction or in the reverse direction.

【What users can do】

- ✓ Control the tool feed direction by setting **Along Profile** or **Against Profile**.



【Notes】

- Climb/conventional milling is not considered when using **Along Profile** and **Against Profile**.
- Only 2-axis Profile Cut supports the two cutting directions of **Along Profile** and **Against Profile**.

【Where is it】

CAM Plan >> 2x Mill >> Profile >> Path Setting >> Cut Direction

5.1.2 Corner Round Adds The Functions of Toolpath Extension And Shortening

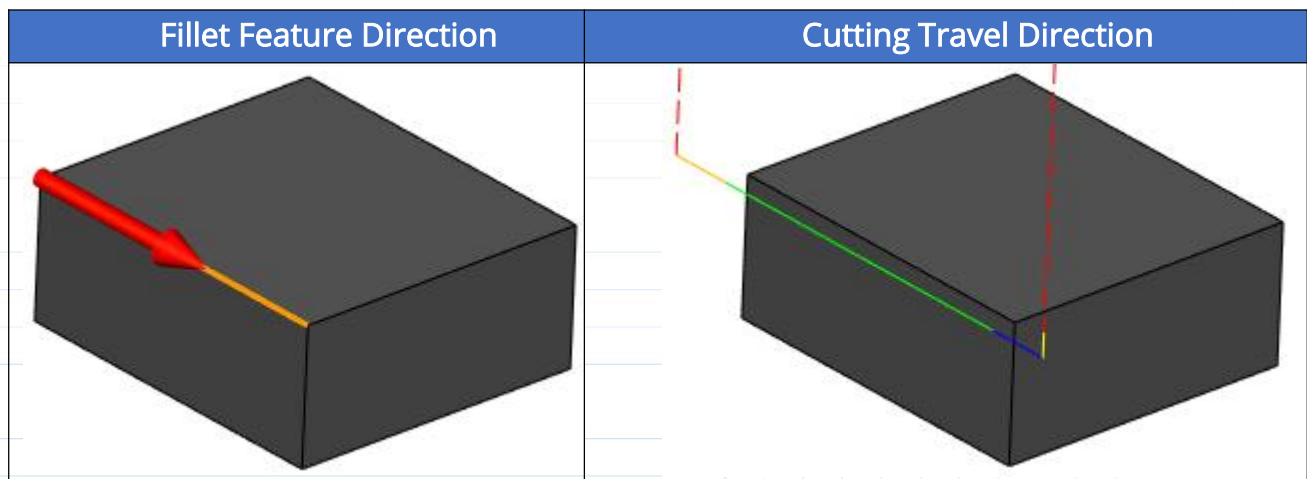
The 2-axis Corner Round operation supports independent extension or shortening of both ends of the toolpath, ensuring complete part machining or avoidance of critical surfaces. Specific parameter adjustments shall be made according to actual machining requirements.

【What users can do】

✓ It supports input of absolute values or percentages of tool diameter to control the extension or shortening distance of the toolpath.

【Notes】

- Extension and shortening are based on the start and end positions of the selected feature, rather than the approach and retract positions of the toolpath.



【Where is it】

CAM Plan >> 2x Mill >> Corner Round >> Limiting >> Profile Start/End Location

5.2 3x Quick

5.2.1 Add 3x Chamfer Debur Operation

The **3x Chamfer Debur** Operation uses a fixed tool axis for edge deburring, capable of processing curved edges, beveled edges and straight edges, making it ideal for deburring complex parts.

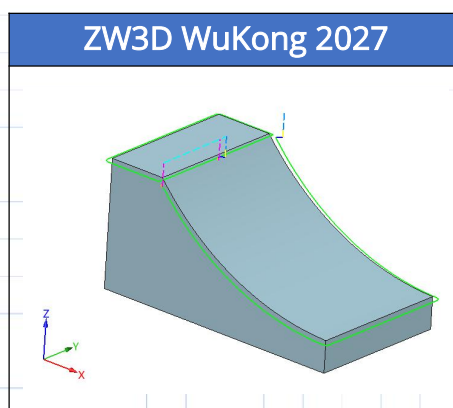
【What users can do】

- ✓ Edges to be machined can be automatically detected from solids or manually selected.
- ✓ Filter Tool Paths.
- ✓ Supports two deburring modes: width-based and depth-based.

【Notes】

- Supports only ball mills, chamfer tools, lollipop mills, and dovetail cutters.
- Recommended for edges without modeled chamfer and fillet.

【Example】



【Where is it】

CAM Plan >> 3x Quick >> 3x Chamfer Debur

5.2.2 Offset 3D Operation Supports the Function of Setting The Tool Path Start Point

Allows specifying the start position of the cutting path and plunging at an appropriate position, effectively avoiding tool entry in hazardous areas of the tool path.

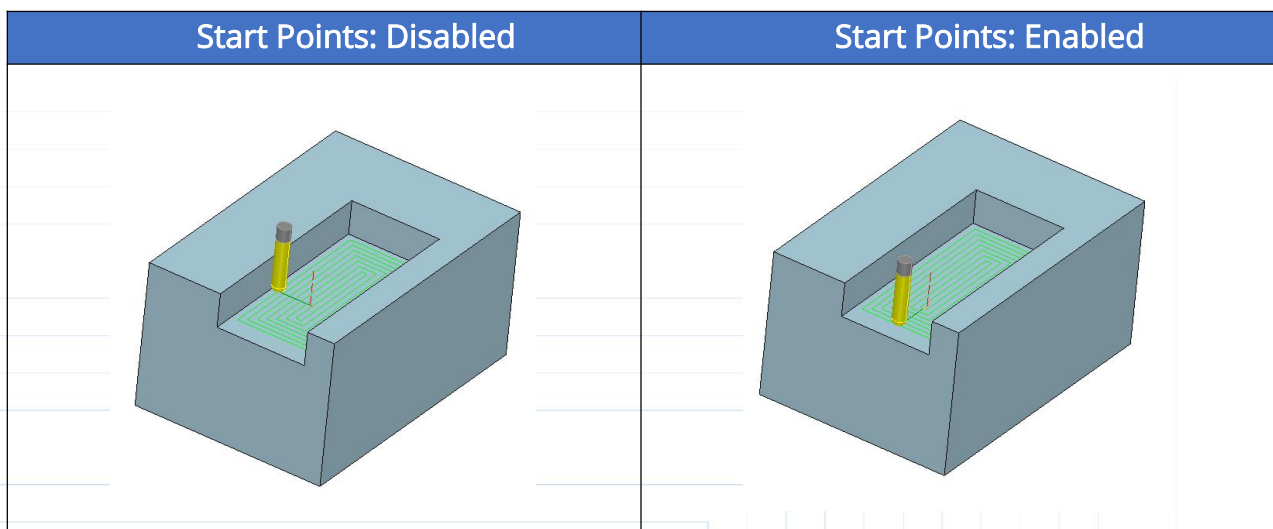
【What users can do】

✓ The process automatically references the specified plunge point to generate an appropriate tool path start point.

【Notes】

- The actual tool path start point of the Operation only references the specified point.

【Example】



【Where is it】

CAM Plan >> 3x Quick >> Offset 3D >> Path Setting >> Cutting Control >> **Start Points**

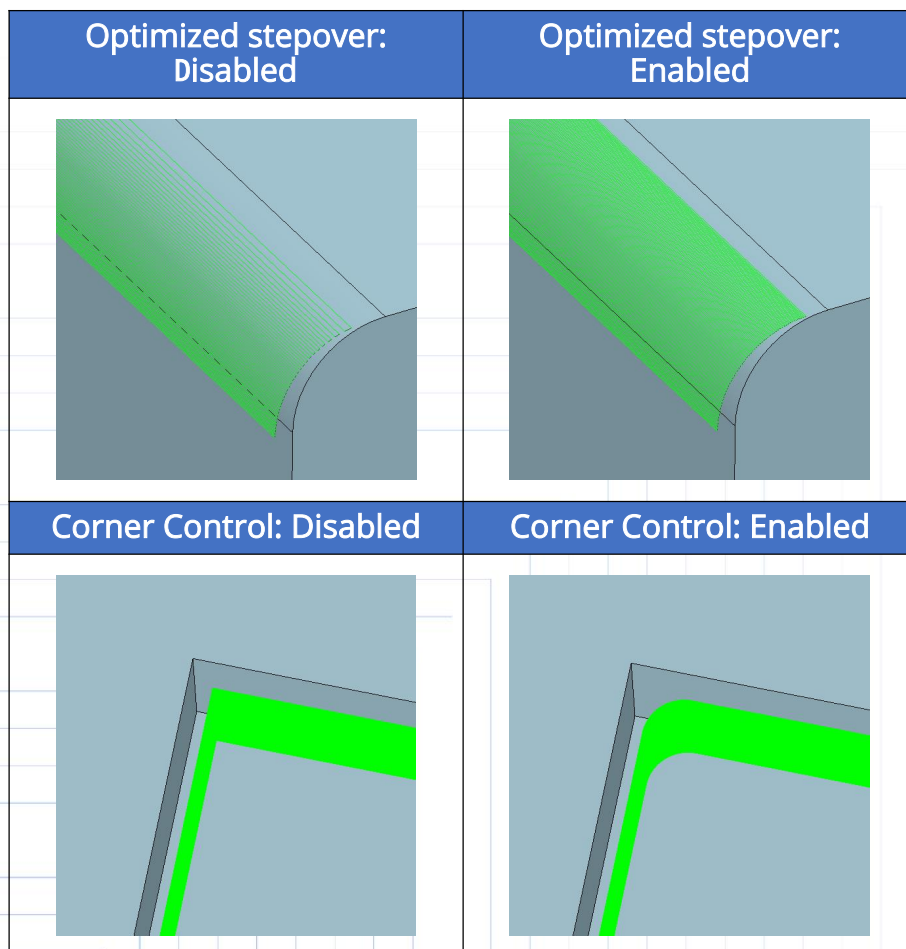
5.2.3 Z Constant Operation Supports the Optimized Stepover Function and the Corner Smoothing Function

1. The optimized stepover function of the Z Constant Operation generates a more uniform stepover on surfaces, achieving better machining results.
2. The corner smoothing function of the Z Constant Operation generates smooth tool paths at internal corners, enabling a smoother tool transition at corners and delivering superior machining results.

【Notes】

- The optimized stepover function cannot generate tool paths on planes normal to the tool axis;

【Example】



【Where is it】

CAM Plan >> 3x Quick >> Z Constant >> Machining Strategy >> Cutting Layers >> Layer >> **Cut Levels**

CAM Plan >> 3x Quick >> Z Constant >> Machining Strategy >> Path Setting >> **Corner Control**

5.2.4 Z Constant Layer Function Optimization

The Z Constant layer function has been optimized, supporting direct editing of the layer depth and stepdown in the layer list for more convenient operation.

【What users can do】

- ✓ Direct editing of layer depth and stepdown in the layer list is supported.

【Notes】

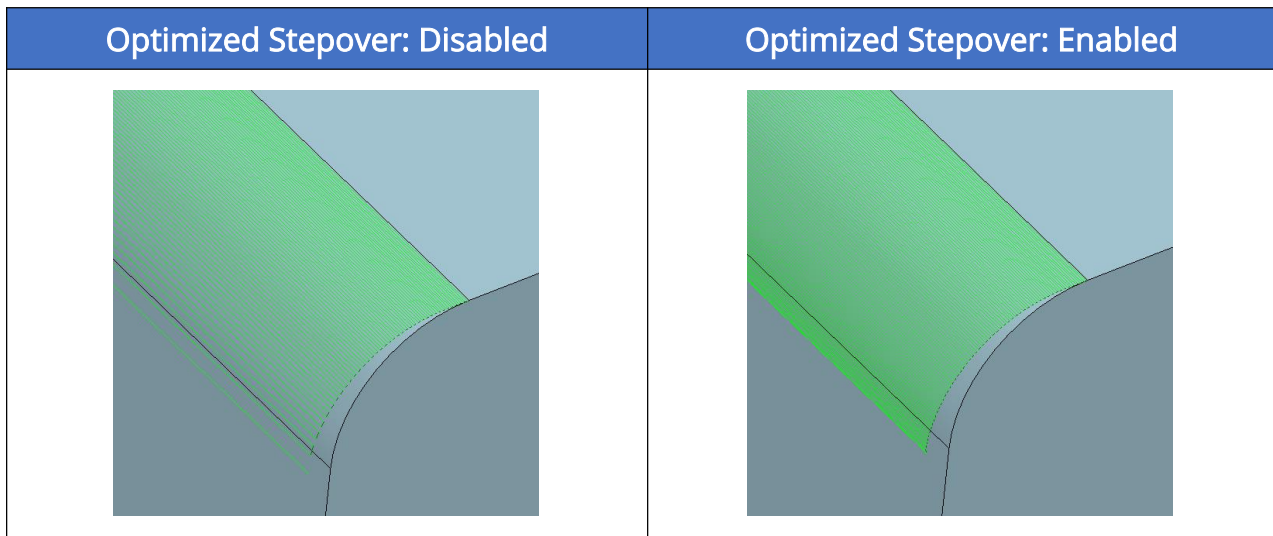
- To create a new layer, you need to click "**New**" first, then edit the layer depth and stepdown to finalize the new layer.

【Where is it】

CAM Plan >> 3x Quick >> Z Constant >> Machining Strategy >> Cut Layers >> **Layer Definition**

5.2.5 The Region Mill Operation Supports the Optimized Stepover Function

The optimized stepover function of the Region Mill Operation can generate a more uniform stepover on curved surfaces, achieving better machining results.



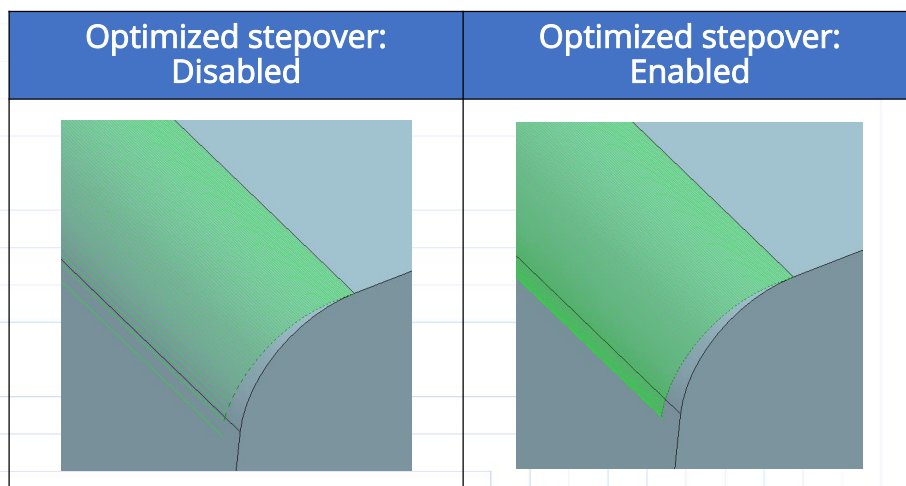
【What users can do】

- ✓ The optimized stepover function can generate a more uniform stepover.

【Notes】

- The optimized stepover function can only be set for shallow areas.

【Example】



【Where is it】

CAM Plan >> 3x Quick >> Region Mill >> Machining Strategy >> Region Control >> Shallow Regions >> Stepover Type

5.2.6 Z Level Supports Tool Compensation

Tool Compensation is newly added to Z Level. This allows users to adjust the machine tool's cutter compensation parameters to achieve precise part machining when insufficient machining occurs due to tool wear during actual production. It ensures dimensional accuracy, eliminates the need for repeated NC program adjustments, and improves production efficiency.

【What users can do】

✓ Control whether to output NC code with cutter compensation activation and deactivation information via the **Tool Compensation** function.

【Notes】

- When the **Tool Compensation** function is enabled, only horizontal arc approach/retract is allowed. In addition, a linear extension tool path must be set before the arc approach path and after the arc retract path for cutter compensation activation and deactivation.
- When the **Tool Compensation** function is enabled, if **Zigzag** is selected for the cutting direction, Add Approach/Retract to Short Links in the linking settings must be set to **Yes**.

【Example】

Tool Compensation = None

Tool Compensation = Offset

% OZlevel 1 N1 G90 G17 G49 G00 N2 T0 M06 N3 S1000 M03 N4 G05.1 Q1. N5 G90 G00 X-17.3 Y-51.106 N6 G43 Z100. H0 M08 N7 Z39.463 N8 G01 Z37.464 F150. N9 X-12.174 Y-50.997 F250. N10 G17 G03 X-9.301 Y-48. I-.127 J2.997 N11 G01 Y-2.99 N12 X-9.305 Y-2.95 N13 X-9.338 Y-2.938 N14 X-44. N15 X-44.127 Y-2.941 N16 G03 X-46.997 Y-5.81 I.127 J-2.997 N17 G01 X-47.106 Y-10.937 N18 G00 Z100. N19 G05.1 Q0 N20 G91 G28 Z0 M05 N21 G91 G28 X0 Y0 M09 N22 M30 %	% OZlevel 2 N1 G90 G17 G49 G00 N2 T0 M06 N3 S1000 M03 N4 G05.1 Q1. N5 G90 G00 X-17.3 Y-51.106 N6 G43 Z100. H0 M08 N7 Z39.463 N8 G01 Z37.464 F150. N9 G41 X-12.174 Y-50.997 F250. D0 N10 G17 G03 X-9.301 Y-48. I-.127 J2.997 N11 G01 Y-2.99 N12 X-9.305 Y-2.95 N13 X-9.338 Y-2.938 N14 X-44. N15 X-44.127 Y-2.941 N16 G03 X-46.997 Y-5.81 I.127 J-2.997 N17 G40 G01 X-47.106 Y-10.937 N18 G00 Z100. N19 G05.1 Q0 N20 G91 G28 Z0 M05 N21 G91 G28 X0 Y0 M09 N22 M30 %
--	---

【Where is it】

CAM Plan >> 3x Quick >> Z Level >> Path Setting >> Tool Compensation

5.2.7 Intelligent Engage and Retract

For different machining features, the intelligent Engage and Retract function can automatically identify closed and open regions, select appropriate methods, and achieve both efficiency and safety.

【What users can do】

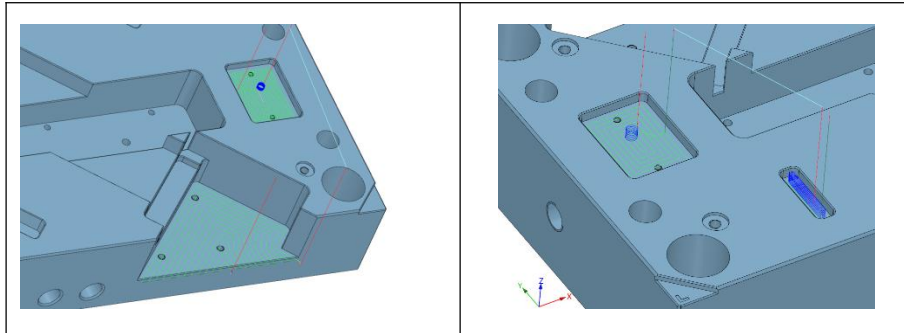
- ✓ Automatically adapts Engage and Retract based on features;
- ✓ Prevent tool center squeezing function;
- ✓ Allows set first engage and final retract patterns;

【Notes】

- Only supports the Offset 2D Roughing operation .

【Example】

ZW3D WuKong 2027 Intelligent Engage and Retract



【Where is it】

CAM Plan >> 3x Quick >> Offset 2D >> [Link and Lead](#)

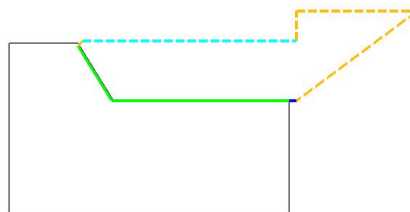
5.3 Turning

5.3.1 Finish Turn Toolpath optimization

Optimize the Finish Turn toolpath: during precision turning, the tool feeds directly from the starting point to the approach point in accordance with the approach motion type, minimizing non-cutting moves and improving machining efficiency.

【What users can do】

✓ The tool moves directly from the starting point to the approach start point according to the approach motion type.



【Notes】

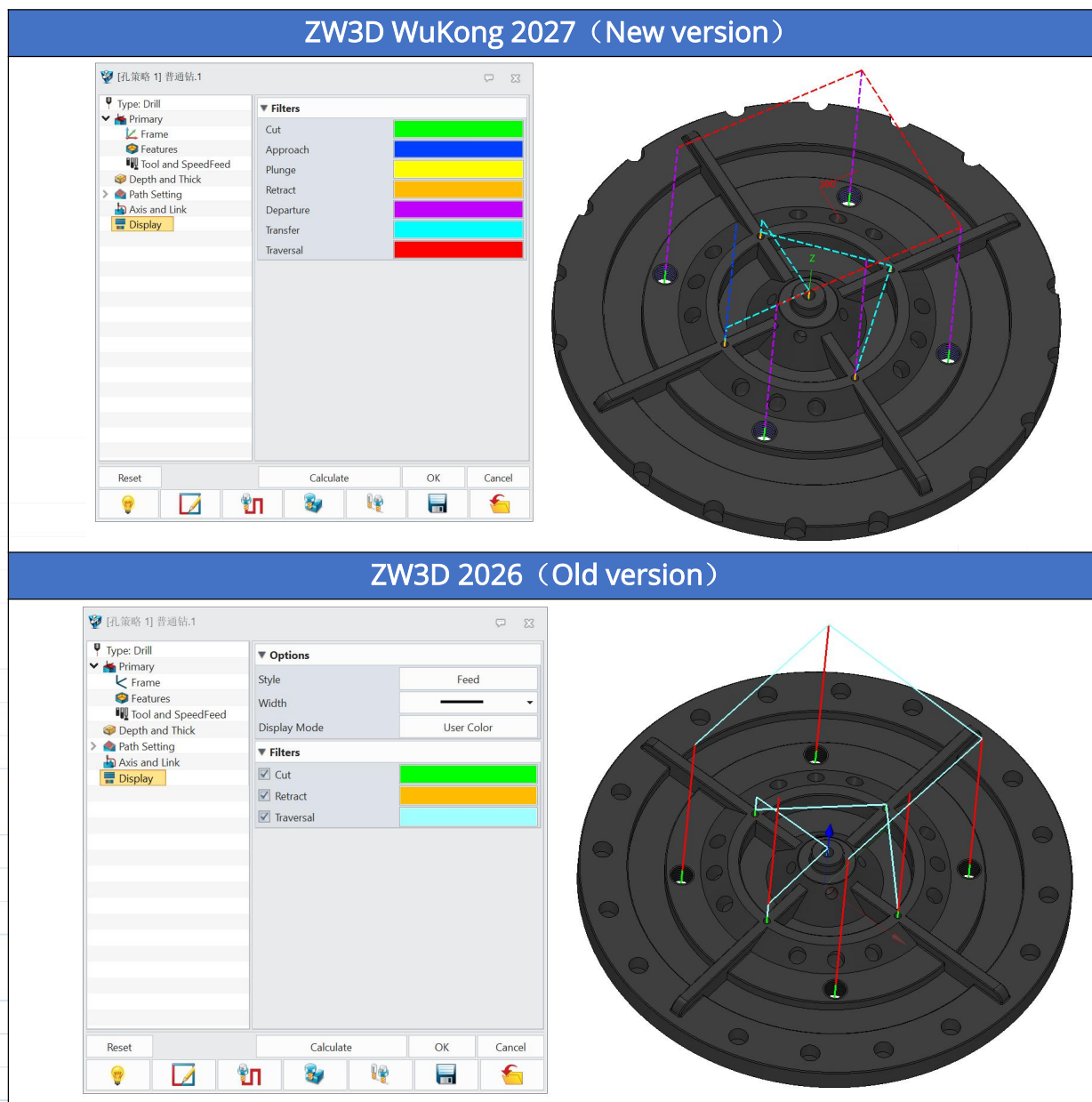
- This optimization is applied only to the approach motion of Finish Turn.

【Where is it】

5.4 Drilling

5.4.1 Add New Toolpath Type in Drilling Module

The drilling operations has added types of approach, plunge, departure, transfer and traversal of the tool path.



【What users can do】

- ✓ Set the display color for each type of tool path, which helps to visually distinguish

the types of tool paths.

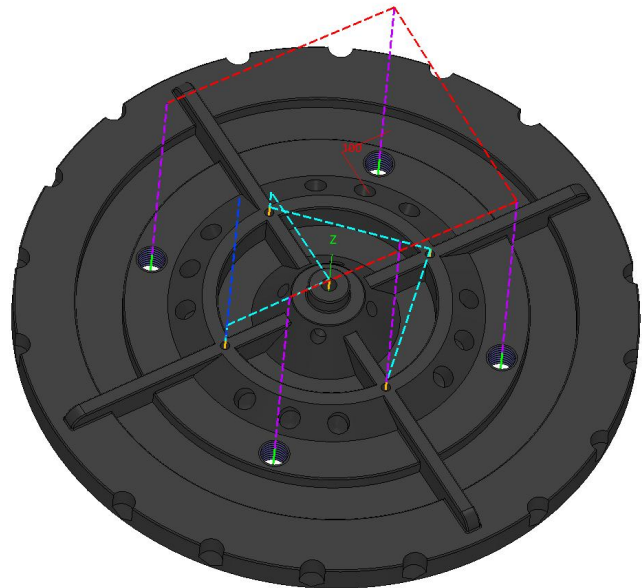
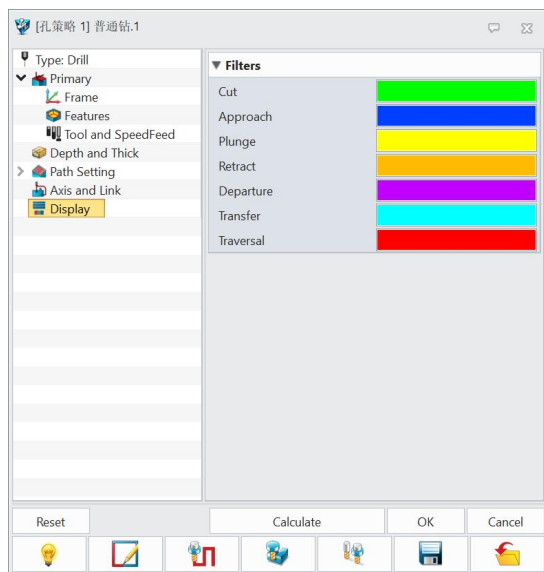
- ✓ Set different feed rates for each type of tool path.

【Notes】

- Approach, departure, transfer type of tool path are only distinguished in the drilling operation.

【Example】

- 1) When adding the Drill operation, the tool path is divided into different types such as Cut, Approach, Plunge, Retract, Departure, Transfer and Traversal. Users can customize the display colors of various tool paths in the display parameter page of the Drill operation.



【Where is it】

CAM Plan >> Drill >> Drill Operation >> Display

5.4.2 The Drilling Operation Supports Single-Step Output

In the drilling operation, an "Output Mode" function has been added. Users can control the output form of the drilling tool path, and they can choose to output it in the drilling cycle mode or in the G01 instruction mode.

Set Output Mode to Cycle Output in Drill operation

Set Output Mode to Single Output in Drill operation

【What users can do】

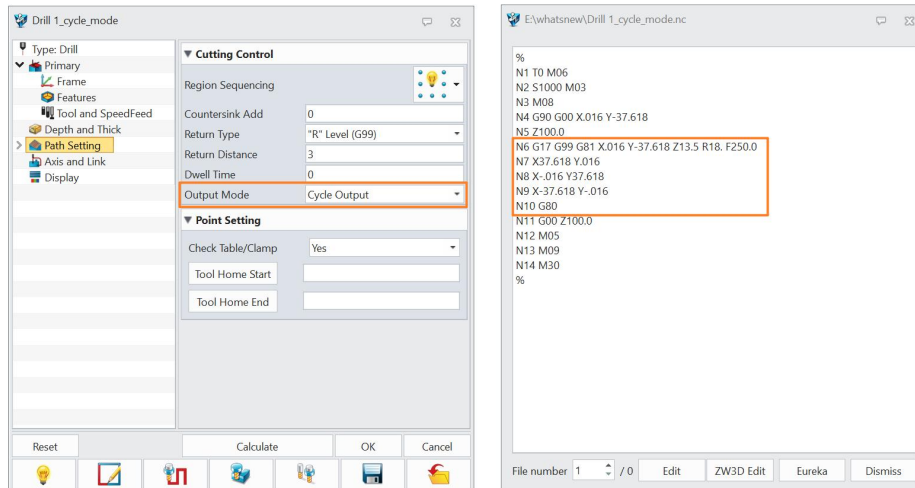
- ✓ Output NC codes in the drilling cycle mode.
- ✓ Output NC codes in G01 instruction mode.

【Notes】

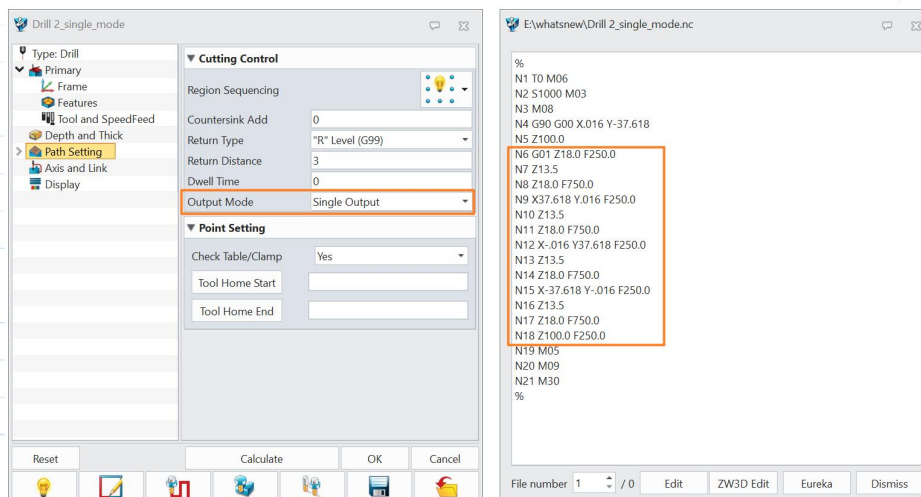
- The Tap operation does not support this function.

【Example】

- 1) Create a Drill operation, On the Path Setting page of the operation dialog, set the output mode to "Cycle Output". Then generation tool path and output NC file. The drilling tool path will output the NC code in a drilling cycle form.



- 2) Create a Drill operation, On the Path Setting page of the operation dialog, set the output mode to "Single Output". Then generation tool path and output NC file. The drilling tool path will output the NC code in G01 instruction form.



【Where is it】

CAM Plan >> Drill >> Drill Operation >> Path Setting >> Output Mode

5.5 Platform Functions

5.5.1 New Electrode Automatic Programming Function

The Electrode Automatic Programming function is dedicated to efficient programming for mold electrode machining. It integrates five core functions: Electrode Machining Programming, Electrode Batch Programming, Electrode Configuration, Process Parameter Library and Template Saving, covering the entire electrode programming process. It automates the whole workflow from feature recognition and parameter configuration to NC code and program sheet output, and supports template reuse and parametric management, adapting to the requirements of standardized electrode machining.

【What users can do】

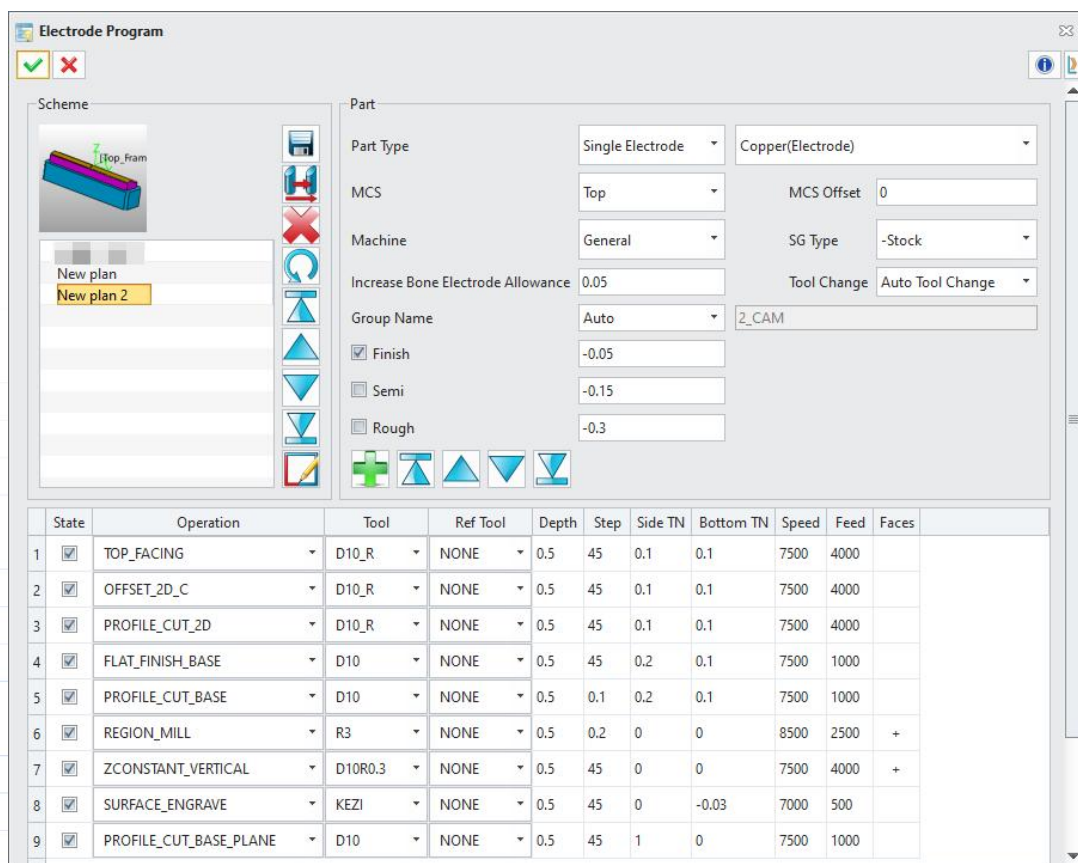
- ✓ The **Electrode Program** function enables one-click automatic programming for a single electrode, supporting intelligent feature recognition, automatic blank creation, template calling, parameter memory, and NC/program sheet output;
- ✓ The **Electrode Auto Batch Program** function realizes one-click batch programming for multiple electrodes, supporting custom part parameters and automatic file saving/closing after programming;
- ✓ The **Electrode Configuration** function pre-configures automatic programming links, including custom post-processing machines, blank dimensions, initial programming parameters and engraving specifications;
- ✓ The **Process Parameter Library** pre-configures tools and cutting parameters, supporting tool retrieval, addition, modification, deletion and consistent parameter display;
- ✓ The **Save Plan** function writes operation information to the user directory for direct calling and process reuse.

【Notes】

- The **Electrode Program** function only supports automatic programming for single electrodes within a single file;
- The **Electrode Program** function only supports automatic programming for electrodes with a cuboid base.

【Example】

- 1) Use the **Electrode Program** function and click Confirm to perform automatic electrode programming.



- 2) One-click automatic electrode programming is completed; you may check if the programming results meet the process programming requirements.

>		✓	TOP_FACING 1	✓
>		✓	OFFSET_2D_C 1	✓
>		✓	PROFILE_CUT_2D 1	✓
>		✓	FLAT_FINISH_BASE 1	✓
>		✓	PROFILE_CUT_BASE 1	✓
>		✓	REGION_MILL 1	✓
>		✓	ZCONSTANT_VERTICAL 1	✓
>		✓	SURFACE_ENGRAVE 1	✓
>		✓	PROFILE_CUT_BASE_PLANE	✓

【Where is it】

CAM Plan >> Setup >> Electrode

5.5.2 Add Post Editor

The Post Processor Editor allows editing post-processor files visually, effectively reducing the difficulty and improving the efficiency of post-processor file editing.

【What users can do】

- ✓ View, edit, and customize post-processor files.
- ✓ Supports three types of CNC systems: FANUC, HEIDENHAIN, and SIEMENS.
- ✓ Supports post-processor editing for 2-3 axis and multi-axis Operations.

【Notes】

- Only supports post processing for milling machining.
- Only supports generating post-processor files with the .zwpost suffix.

【Where is it】

CAM Plan >> Output >> ZW3D PostEditor

5.5.3 New Analyze Toolpath function

Analyze Toolpath provides robust safeguards for machining safety. It supports dual verification of toolpath overcut and collision analysis, enabling accurate localization and visual display of abnormal toolpaths caused by collision or overcut. It also offers optimization features including toolpath splitting, overlap processing and filtering, along with tool editing capability, allowing rapid adjustment of unsafe toolpaths and greatly enhancing the safety of toolpath programming.

【What users can do】

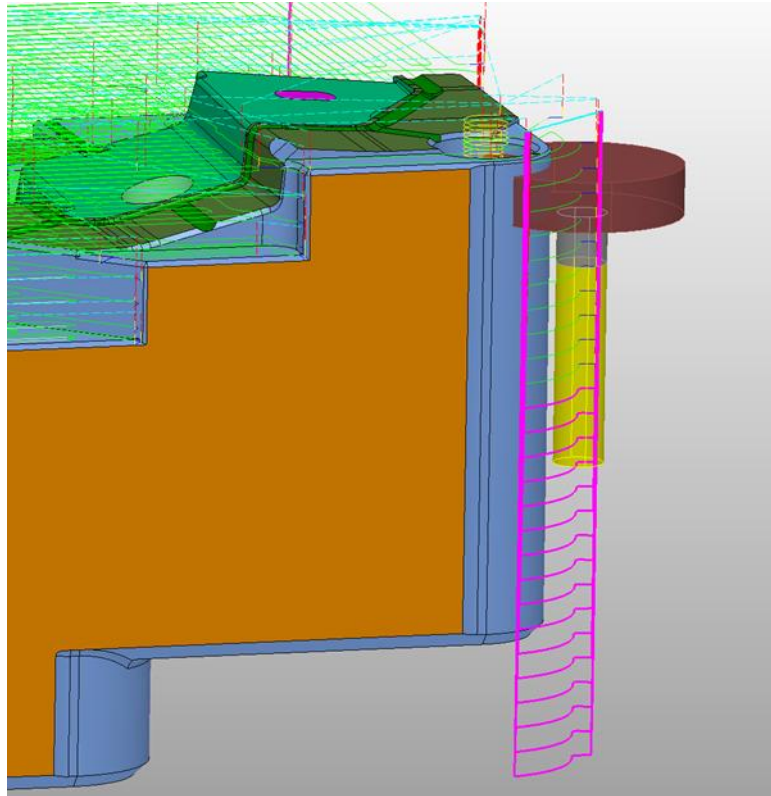
- ✓ Supports simultaneous toolpath collision detection and overcut detection.
- ✓ Enables accurate localization of hazardous locations such as collisions and overcuts.
- ✓ Allows toolpath splitting to separate unsafe toolpaths from safe ones.
- ✓ Supports tool adjustment to convert unsafe toolpaths into safe ones.

【Notes】

- The new Analyze Toolpath function features static stock removal detection for workpiece objects, and is not adapted to the In-Process Workpiece (IPW) dynamic stock removal scenario.

【Example】

- 1) Select a part and create a roughing operation.
- 2) Perform an Analyze Toolpath operation on the toolpath, check the analysis results, and any collisions in the toolpath will be accurately displayed.



【Where is it】

CAM Plan >> Right-click the selected operation >> [Analyze Toolpath](#)

5.5.4 New Toolpath Transform function

The New Toolpath Transform function integrates four core toolpath adjustment capabilities: move, pattern, mirror and scale. Independent function icons are set on the Ribbon bar, and clicking an icon quickly brings up its dedicated operation interface. It enables efficient toolpath reuse and customized adjustment, adapting to the diverse toolpath optimization needs in mold machining.

【What users can do】

✓ Move supports four modes: point-to-point, specified increment direction, axis rotation and coordinate system to coordinate system; Pattern includes two modes: rectangular and circular; Mirror supports the specified plane mode; Scale supports the specified origin mode.

✓ Toolpath transformation results can be set to three states: do not retain the source object, associate with the source object, and be independent of the source object. Values for Move and Pattern support the setting of both total value and incremental value modes.

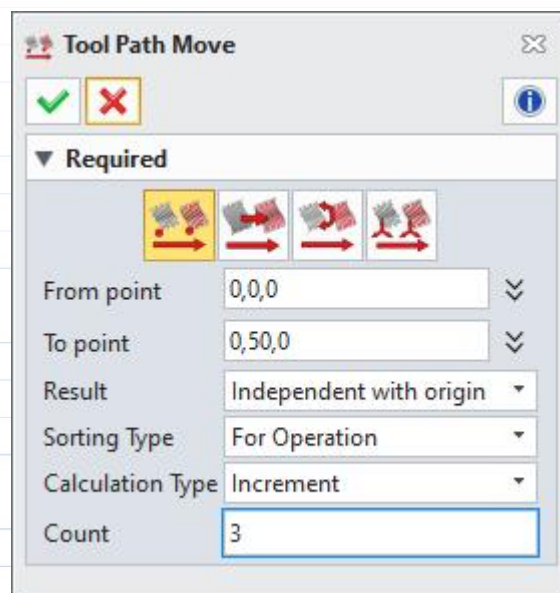
✓ The Toolpath Transform function supports the corresponding display of parent-child relationships in the Navigator. Transformed operations allow one-click removal of transformations and support flexible modification of operation parameters and machining areas.

【Notes】

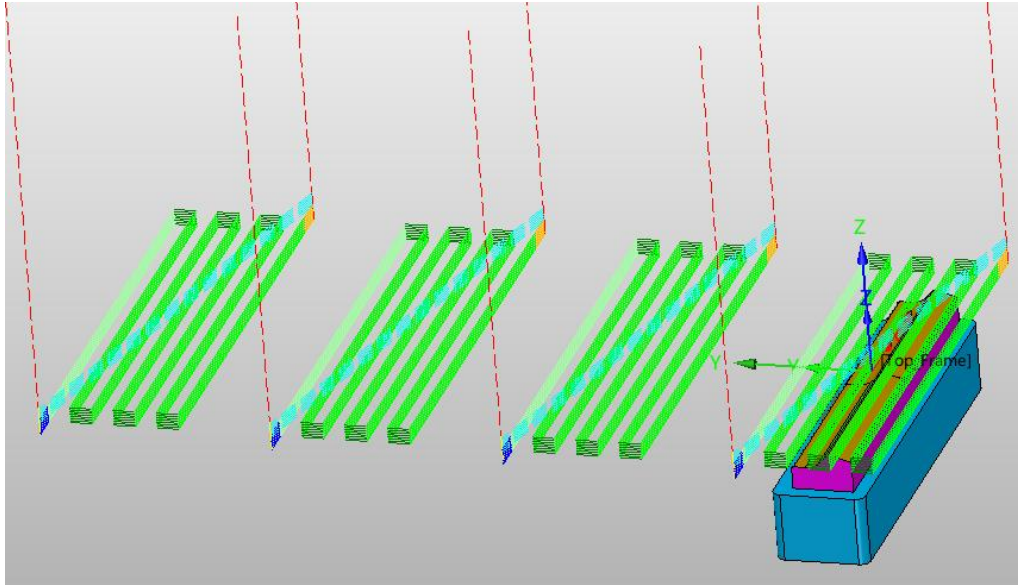
- The operation sequence requires selecting the target operation first, then launching the corresponding Toolpath Transform function.

【Example】

- 1) Select the target operation, activate the Move Toolpath function, and set the parameters as shown in the image .



- 2) Generate three identical new toolpaths at the set distance on the canvas.



【Where is it】

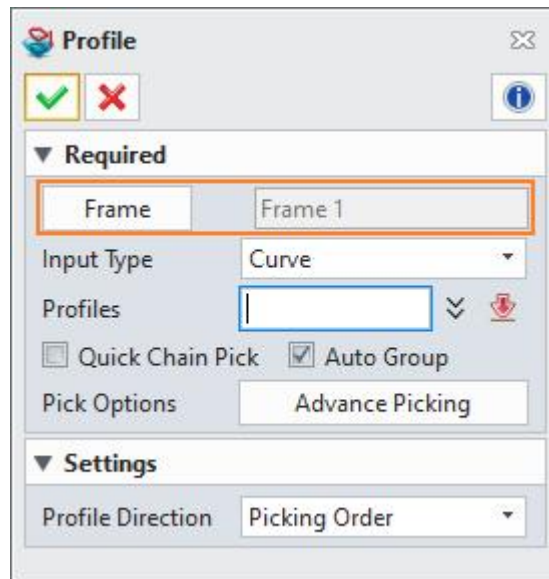
[CAM Plan >> Tool Path Editor >> Transform](#)

5.5.5 Enhanced capabilities for profile features

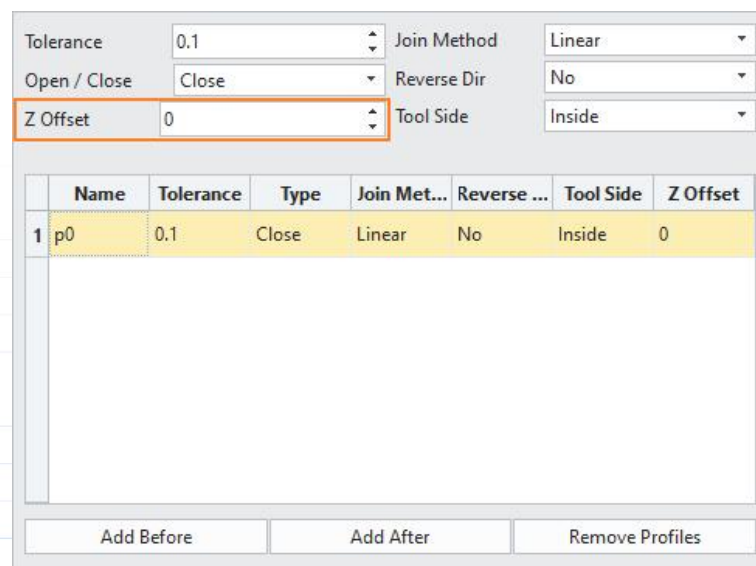
The profile feature interaction is optimized and its capabilities are enhanced, delivering improved visualization and broader application scenario coverage.

【What users can do】

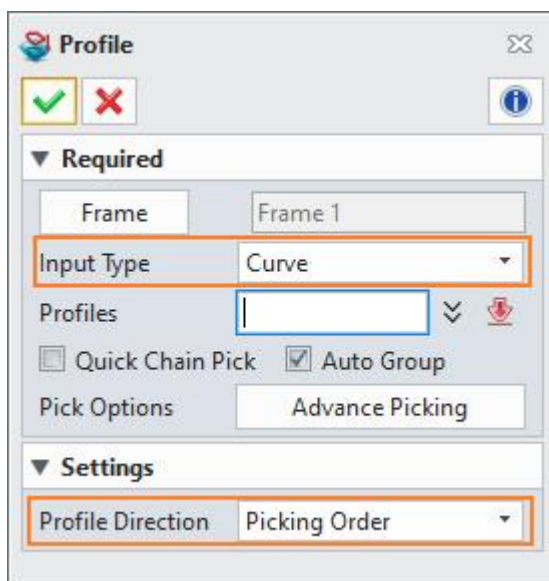
- ✓ The profile feature is associated with the coordinate system, and the tool side direction is displayed correctly according to the selected coordinate system.



✓ A **Z Offset** attribute is newly added to profile feature for adaptation to 2.5-axis machining scenarios.



✓ When profile features are selected via the **curve** input type, the profile feature direction can be defined according to the selection order.

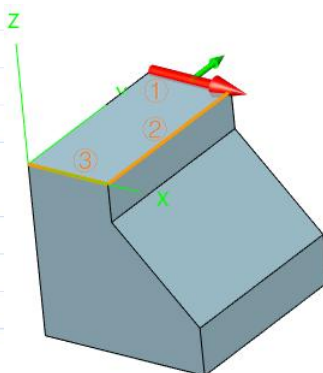


【Notes】

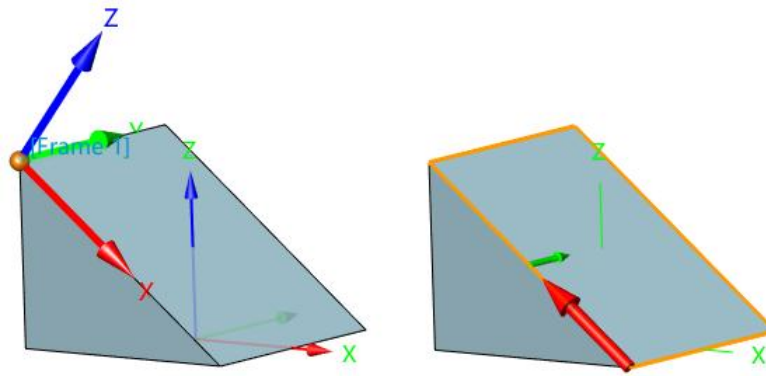
- Profile direction judgment rules are only supported when the **Input Type** is set to **Curve**. It is recommended to use the point selection method when determining the profile feature direction according to the selection order.

【Example】

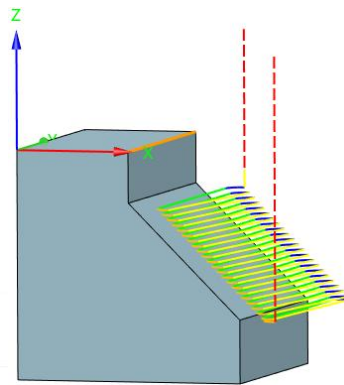
- 1) Set **Profile Direction** to **Picking Order** to determine the profile feature direction according to the selection order.



- 2) Display the profile feature tool side based on the specified coordinate system Frame 1.



- 3) Set the offset value and draft angle for 2.5-axis machining. As shown in the figure, select the top surface edge when the actual machining requirement is a beveled surface.



【Where is it】

CAM Plan >> Setup >> Profile

5.5.6 Optimization of User Define tool interaction

A brand-new interactive interface for User Define tool, with enhanced ease of use in user define tool creation and improved design efficiency.

【What users can do】

- ✓ Edit parameters directly in the table with real-time preview on the right upon completion;
- ✓ Adjust the flute length.

【Notes】

- Click the **Add** button first before editing the tool parameters.

Tool Len(L) 0
Flute Len(FL) 0
Flutes(F) 4

LL LA +SwA StA AR -SwA

Add Delete

Type	Line Length(LL)	Angle(LA/StA)	Arc Radius(AR)	Sweep Ang(SwA)
------	-----------------	---------------	----------------	----------------

- Adjust the addition type via the drop-down box: **Line** or **Arc**.

Tool Len(L) 0
Flute Len(FL) 0
Flutes(F) 4

LL LA +SwA StA AR -SwA

Add Delete

Type	Line Length(LL)	Angle(LA/StA)	Arc Radius(AR)	Sweep Ang(SwA)
1 Line	5	0		

Line
Arc

【Example】

Create an end mill with a diameter of 10, a corner radius of 1, a flute length of 20 and an overall tool length of 50 via the user define tool function.

1) Add a **Line** and set its length to 4 and start angle to 0.

Tool Len(L) 0
Flute Len(FL) 0
Flutes(F) 4

LL LA +SwA StA AR -SwA

Add Delete

	Type	Line Length(LL)	Angle(LA/StA)	Arc Radius(AR)	Sweep Ang(SwA)
1	Line	4	0		

2) Add an **Arc** and set its start angle to 0, radius to 1 and sweep angle to 90.

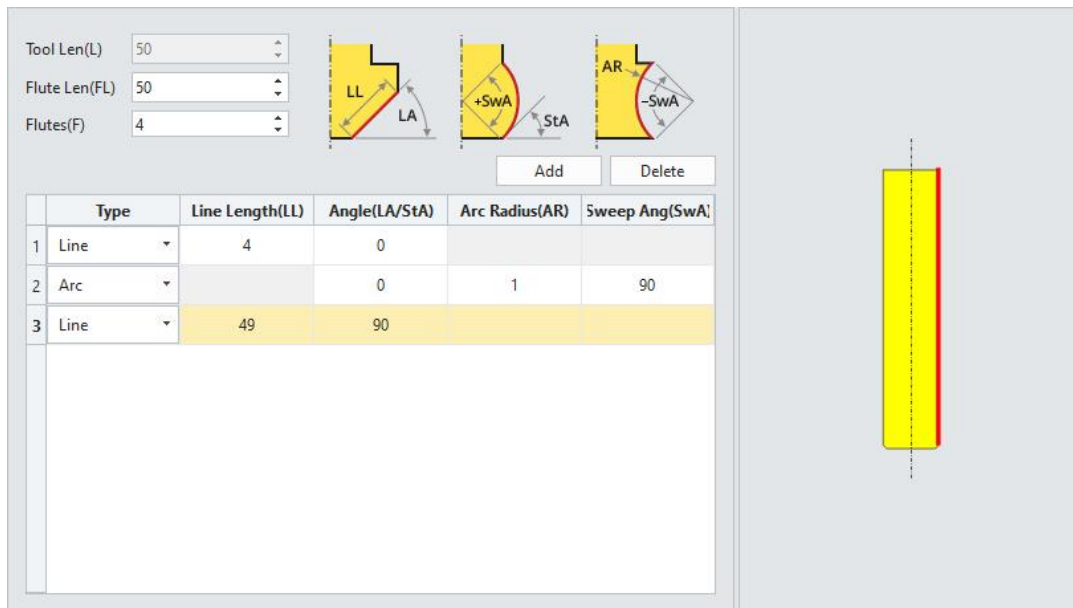
Tool Len(L) 1
Flute Len(FL) 1
Flutes(F) 4

LL LA +SwA StA AR -SwA

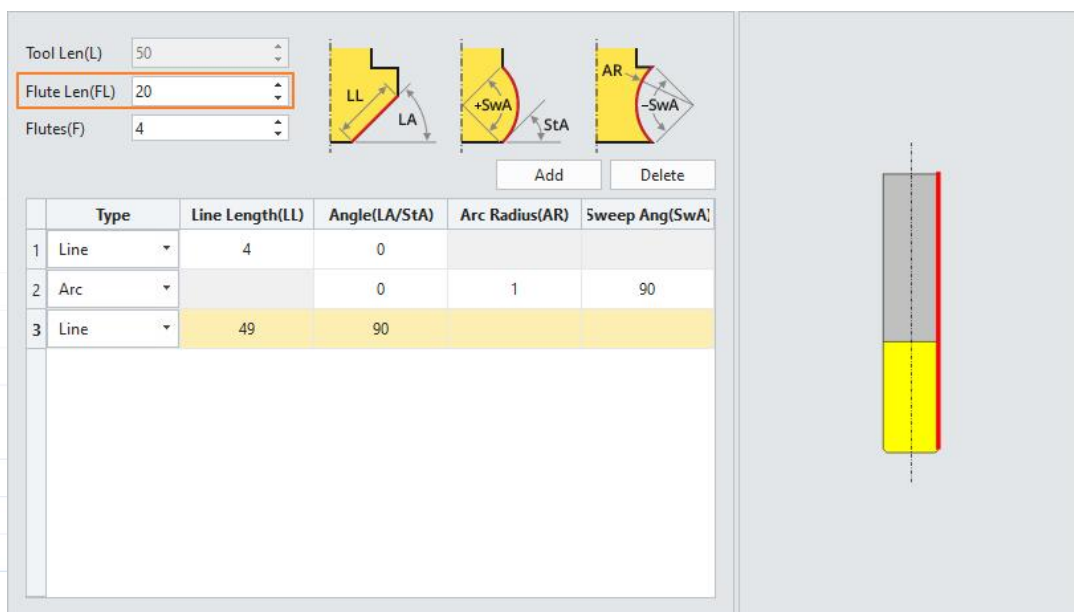
Add Delete

	Type	Line Length(LL)	Angle(LA/StA)	Arc Radius(AR)	Sweep Ang(SwA)
1	Line	4	0		
2	Arc		0	1	90

3) Add a **Line** and set its length to 49 and start angle to 90.



4) Set the **Flute len(FL)** to 20 and complete the creation of the target tool.



【Where is it】

CAM Plan >> Setup >> Tool >> Mill >> User Define Tool

5.5.7 Add Dovetail Tool

Add a dovetail tool creation method to improve the dovetail cutter creation efficiency. You can create a dovetail tool directly without the need to create it via the user define tool function.

【Notes】

- The neck diameter must not exceed the upper flute diameter;
- The dovetail tool only supports 2-axis Profile operation and 3-axis Chamfer Deburring operation.

【Where is it】

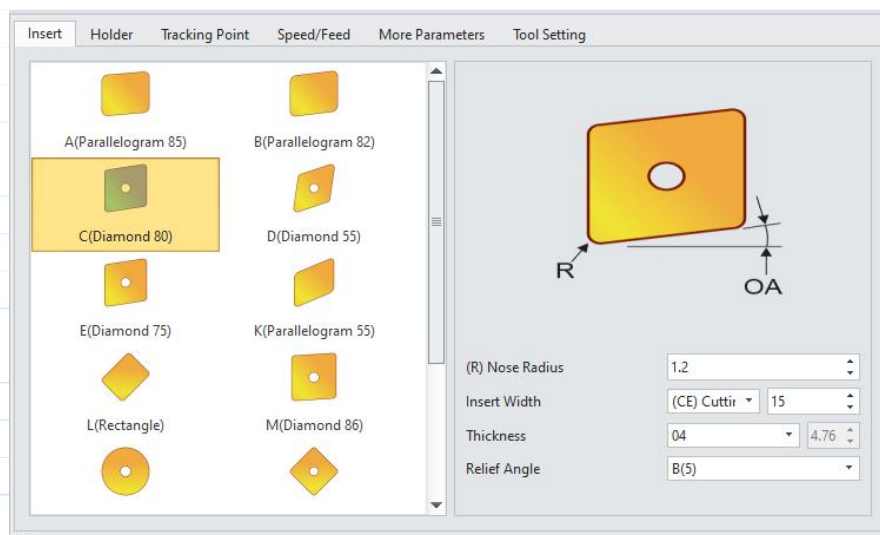
CAM Plan >> Setup >> Tool >> Mill >> Dovetail

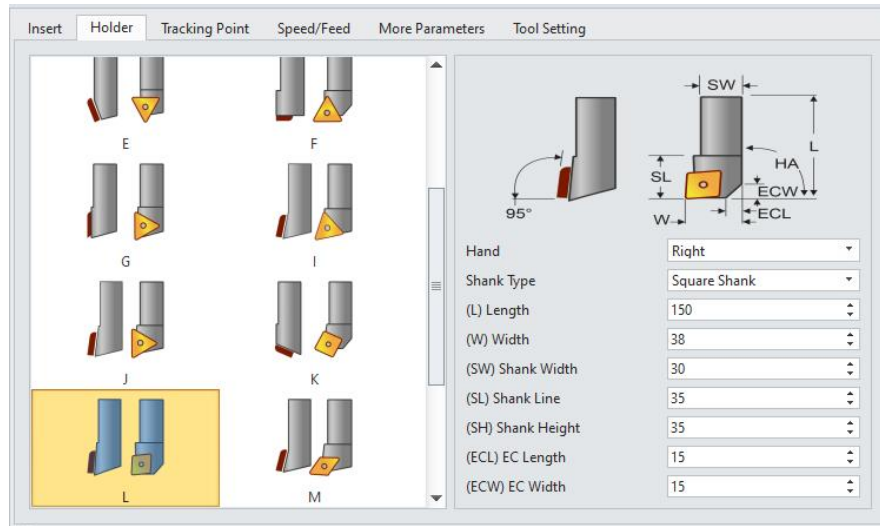
5.5.8 Lathe Tool Interaction Optimization

All-new lathe tool interface interaction and lathe tool creation workflow, delivering greater intuitiveness and ease of use.

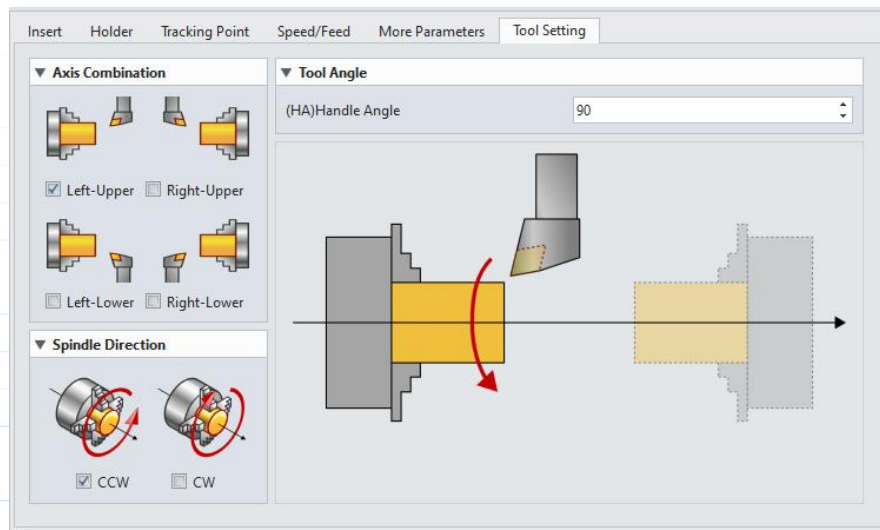
【What users can do】

- ✓ The insert and tool holder selection interface features a tiled layout for intuitive and clear visualization of creation objects.





- ✓ Support the definition of spindle and tool post positions for greater alignment with actual machining scenarios.
- ✓ Automatically determine the insert position based on the **Axis Combination** and **spindle direction** to ensure cutting with the leading edge at all times.



【Notes】

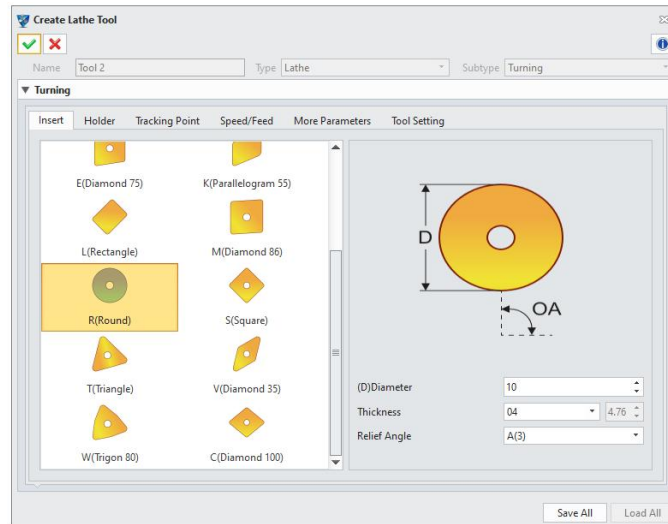
- Remove the original **insert position** parameter; the insert position is automatically determined based on the **Axis Combination** and **spindle Direction**.

【Where is it】

CAM Plan >> Setup >> Tool >> Lathe

5.5.9 Add Circular Insert For Lathe Tools

Circular insert are added for turning, enabling more comprehensive coverage of turning machining scenarios.



【Where is it】

CAM Plan >> Setup >> Tool >> Lathe

5.5.10 Optimize Add Stock Function

Optimize the "Add Stock" function, add support for generating the minimum bounding box in the Z direction of a machining coordinate system, support locking specific X, Y or Z limits to prevent changes to the limit values when adjusting the stock model limits, and support quickly creating a machining coordinate system at the 27 key points of the stock boundary.

【What users can do】

- ✓ Support the generation of the minimum bounding box based on the Z-axis of a machining coordinate system.
- ✓ Support locking specific X, Y or Z boundaries to prevent the values from being changed when adjusting the boundaries of the Stock model.

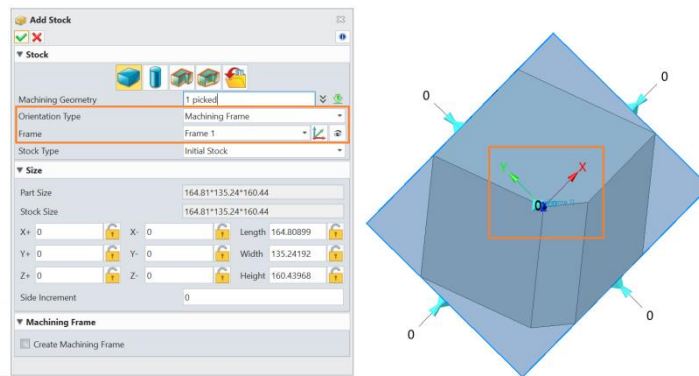
✓ Support the quick creation of machining coordinate systems at 27 key points on the stock boundary.

【Notes】

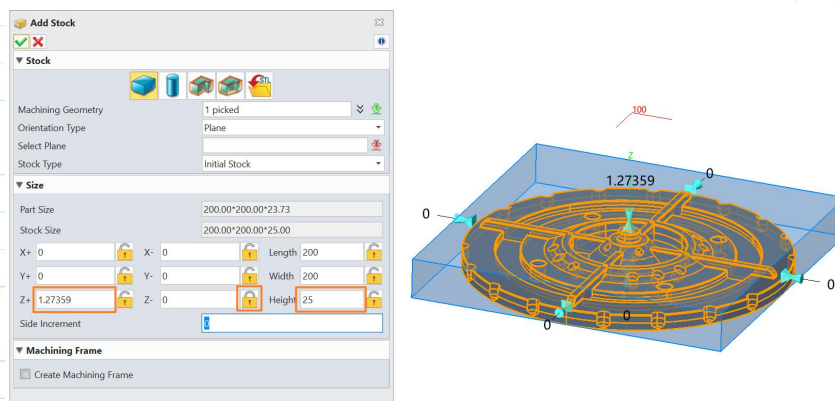
- The function of generating the minimum bounding box based on the Z-axis of a machining coordinate system is only supported on Block Stock type.

【Example】

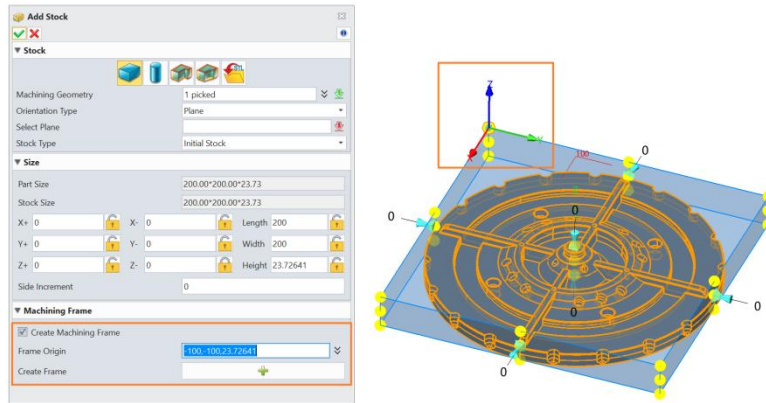
1) Generate the minimum bounding box along the Z-axis of a machining coordinate system.



2) Lock the limits of Z-, then output the height of the stock. The system will automatically calculate the limit value of Z+.



3) Activate the Machining Frame function, select one of the key points on the stock, and quickly create the machining coordinate system.



【Where is it】

CAM Plan >> Setup >> Add Stock

5.5.11 Show/Hide Interaction Optimization

Grayed-out states have been added for operations and folder icons to indicate the hidden status of operations and folders. The icons can also serve as toggle buttons to show or hide operations. You can quickly identify and switch the show/hide status of operations, thereby improving programming efficiency.

【What users can do】

- ✓ Left-clicking the icons of operations and folders allows you to quickly toggle the show/hide status of the objects.
- ✓ The icons of operations and folders can display the show/hide status of the objects in real time.

【Notes】

- Only supports process folders and processes;
- A operation folder is set to the show status if at least one process within the folder is in the show status.

【Example】

ZW3D 2026	ZW3D WuKong 2027
 Operation Folder 1  Region Mill 1	 Operation Folder 1  Region Mill 1

【Where is it】

[CAM Plan >> Browser Manager >> NC Sequence View](#)

5.5.12 Add non-orthogonal 5-axis output capability

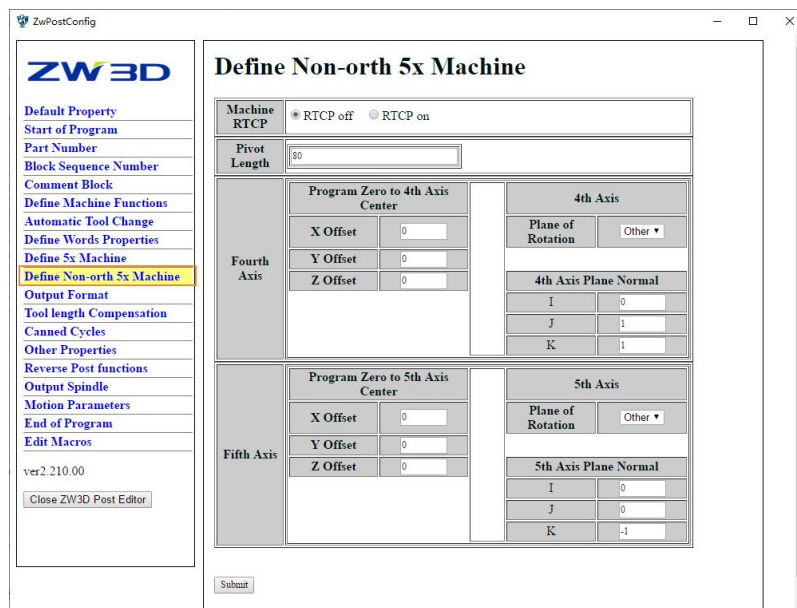
Delivers non-orthogonal 5-axis output capability, enabling more comprehensive coverage of multi-axis machining application scenarios.

【What users can do】

- ✓ Provides default post-processing files for non-orthogonal 5-axis output.

ADV_ZW_FANUC_5X_BC_HeadB_TableC_Non_orthogonal.znc
ADV_ZW_FANUC_5X_BC_TableC_on_TableB_Non_orthogonal.znc

- ✓ Provides the capability for editing and customizing non-orthogonal 5-axis post-processing files.



【Notes】

- Only supports the editing and customization of the provided default post-processing files for non-orthogonal 5-axis output;
- The following specifications apply to the 4th and 5th axes: For a single turntable + single tilting head configuration: the 4th axis is the tilting head, and the 5th axis is the turntable .For a double tilting head configuration: the 4th axis is the driven axis, and the 5th axis is the driving axis .For a double turntable configuration: the 4th axis is the driving axis, and the 5th axis is the driven axis.

【Where is it】

[CAM Plan >> Output >> ZW3D Post >> Web Edit](#)

5.5.13 New CAM Global setting

In the system configuration, a new CAM global settings function has been added. This function provides options for setting the line type, line width, and color of the tool path, and also supports the configuration of tool display transparency and tool path verify speed.

【What users can do】

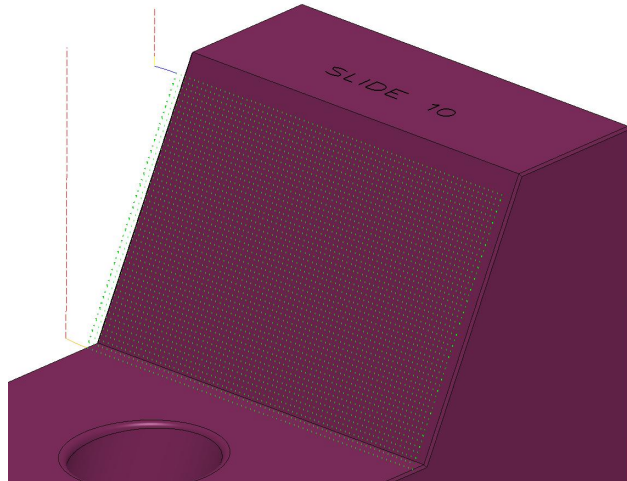
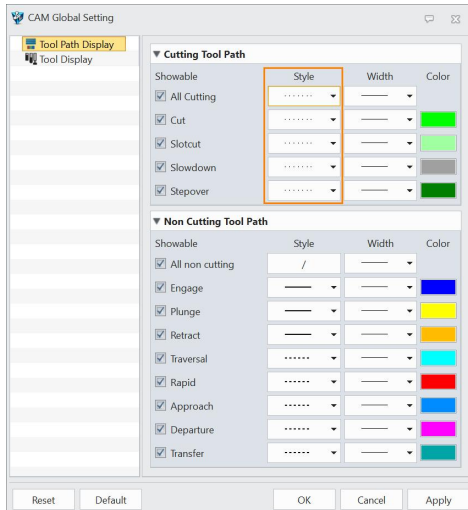
- ✓ Set the line type, line width and color of the tool path in the system configuration.
- ✓ Set the transparency of tool display in the system configuration. This setting will be applied to the tool display command in Setup's Ribbon bar.
- ✓ Set the toolpath verify speed in the system configuration. This setting will be applied to the tool path animation function in Setup's Ribbon bar.

【Notes】

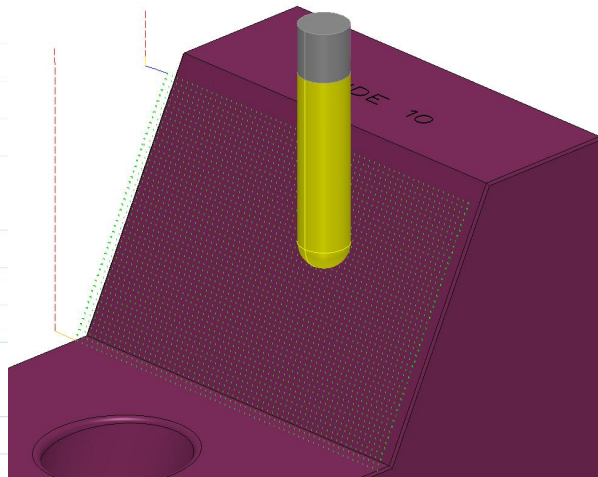
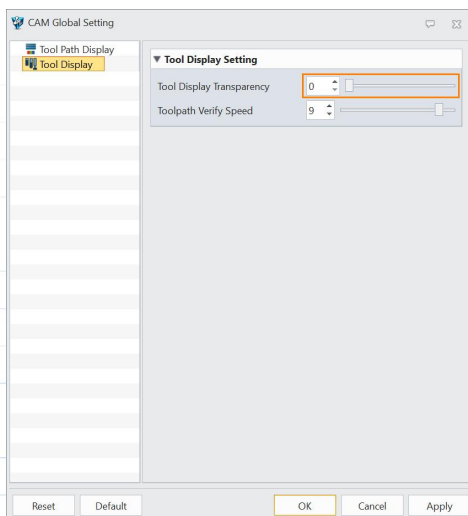
- The transparency setting for tool display is only applied to the tool display commands in Setup's Ribbon bar and will not affect the tool path verify function provided by the right-click menu in the operation.
- The tool path verify speed setting only applies to the tool path animation function in Setup's Ribbon bar, and will not affect the tool path verify function provided by the right-click menu in the operation.

【Example】

- 1) In the CAM Global settings, change the line type of the cutting tool path from "solid line" to "dot pattern" and confirm.
- 2) After creating a "Region Mill" operation and generating the tool path, the cutting part of the tool path will change from the original "solid line" style to a "dot pattern" style.



- 3) In the CAM Global settings, set the transparency of the tool display page to 0 and confirm the settings.
- 4) After creating a "Region Mill" operation and generating the tool path, use the tool path animation function in Setup's Ribbon bar. Currently, the displayed state of the tool is opaque.



【Where is it】

Utilities >> Configuration >> CAM >> Other setting

5.5.14 Add Display Function And Optimize Simulation Function

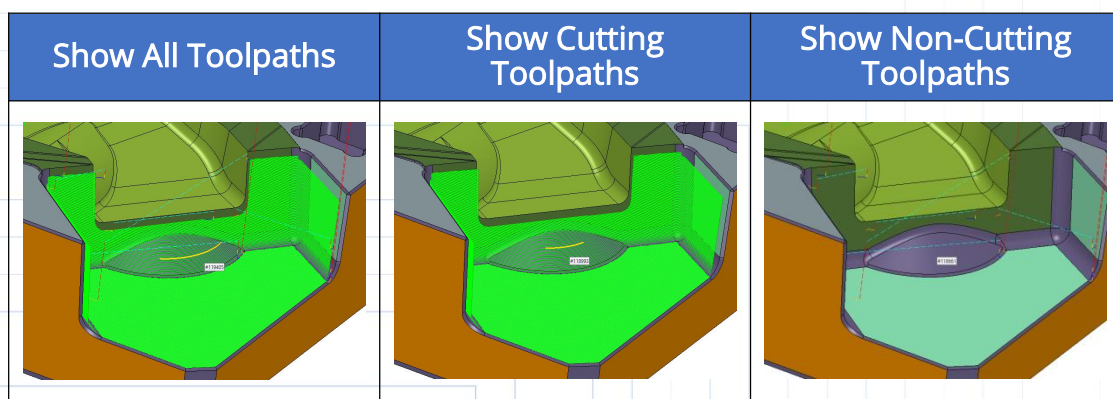
Provide the display capability of cutting toolpath, non-cutting toolpath, tool, tool tip point and tool axis direction, as well as the toolpath simulation capability without entering the simulation environment.

【What users can do】

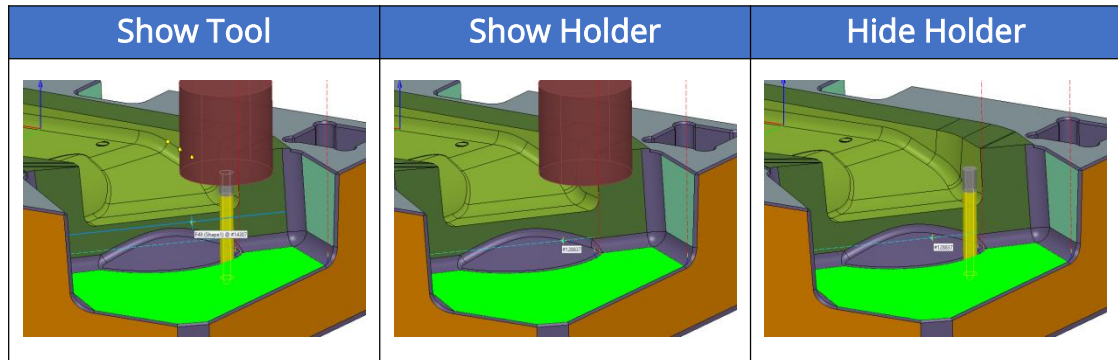
- ✓ Globally control the display and hide of cutting toolpaths and non-cutting toolpaths. Hidden tool paths are displayed when the operation is directly selected;
- ✓ Control the display of different parts of the tool; the tool is displayed directly when the tool path is clicked;
- ✓ Control the display and hide of tool tip points and tool axis directions of the tool path;
- ✓ Perform toolpath simulation directly without entering the simulation environment.

【Example】

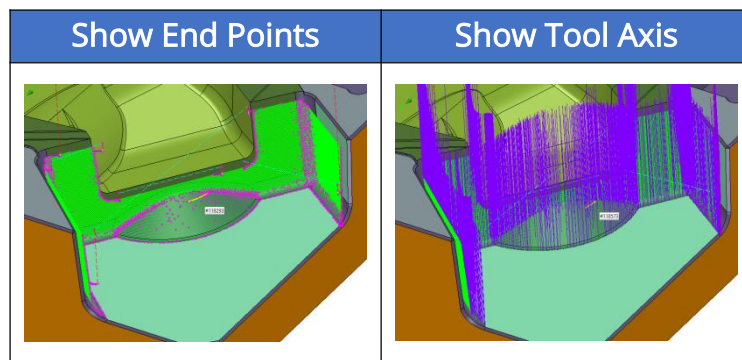
Toolpath display



Tool display



End Point and Tool Axis Display



【Where is it】

CAM Plan >> Setup >> Display/Tool Path Animation

6 Teammate CAD

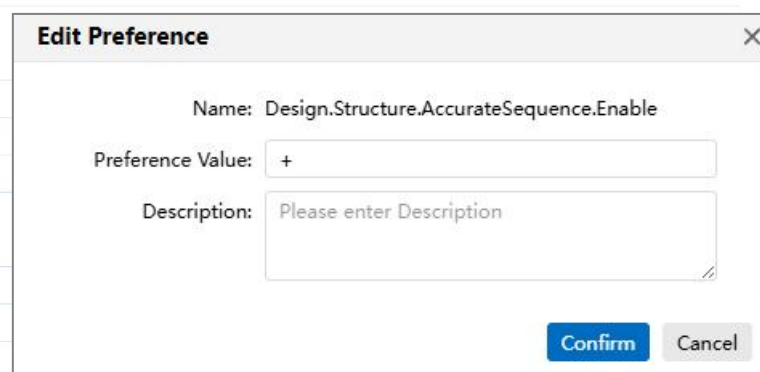
6.1 Teammate CAD

6.1.1 ★ New Assembly Version Rule Management Feature

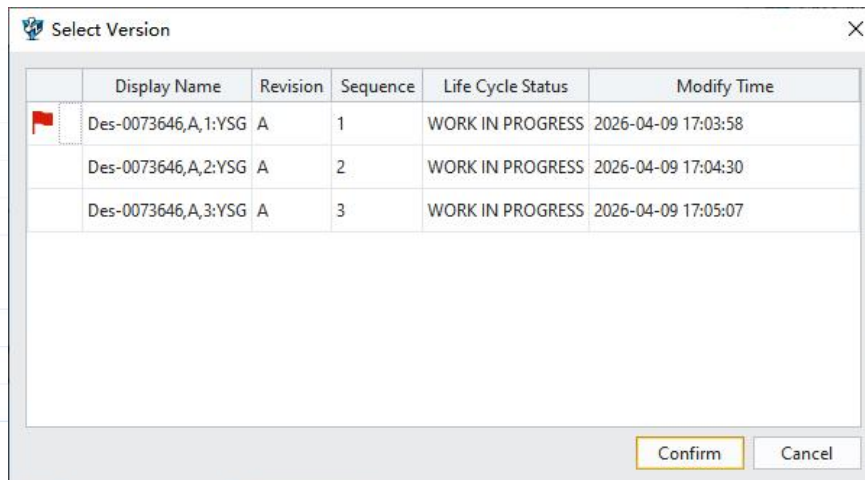
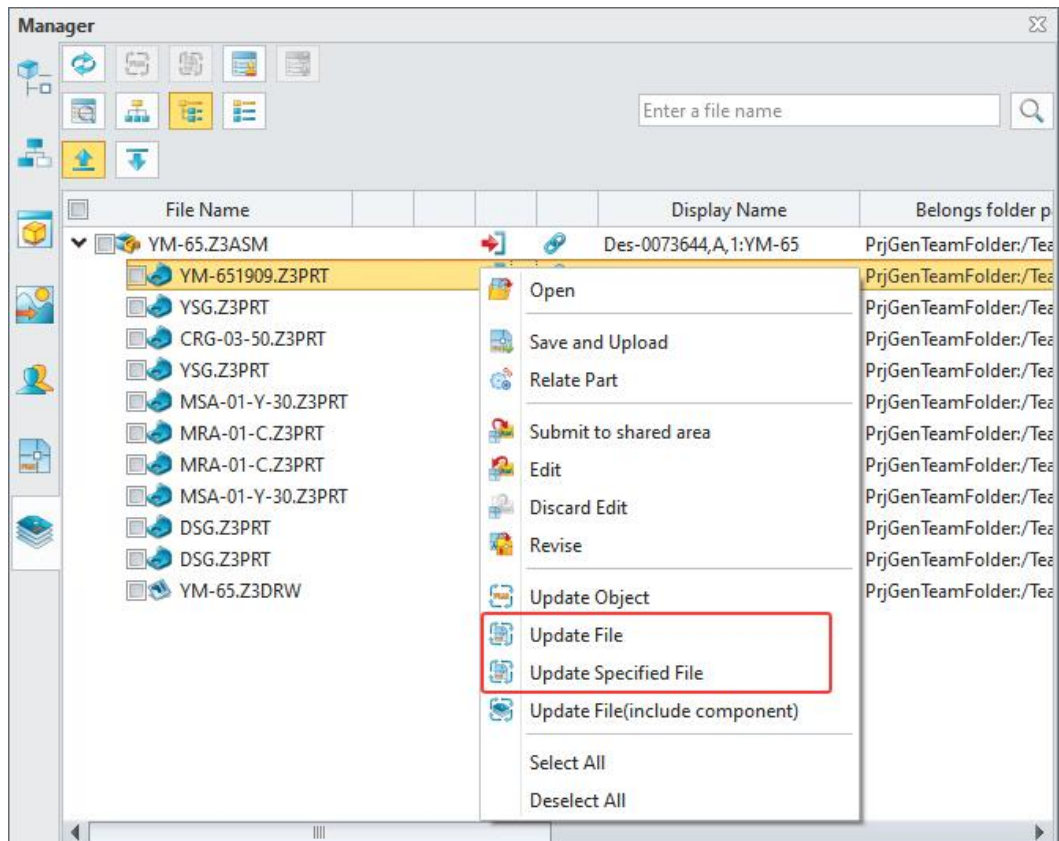
During collaborative design processes, assembly components can be loaded using two common management mechanisms: either based on the last saved version or the latest version. The system preferences control the loading method for new versions. Additionally, a component version update feature is provided to enable engineers to evaluate the most suitable assembly version.

【What users can do】

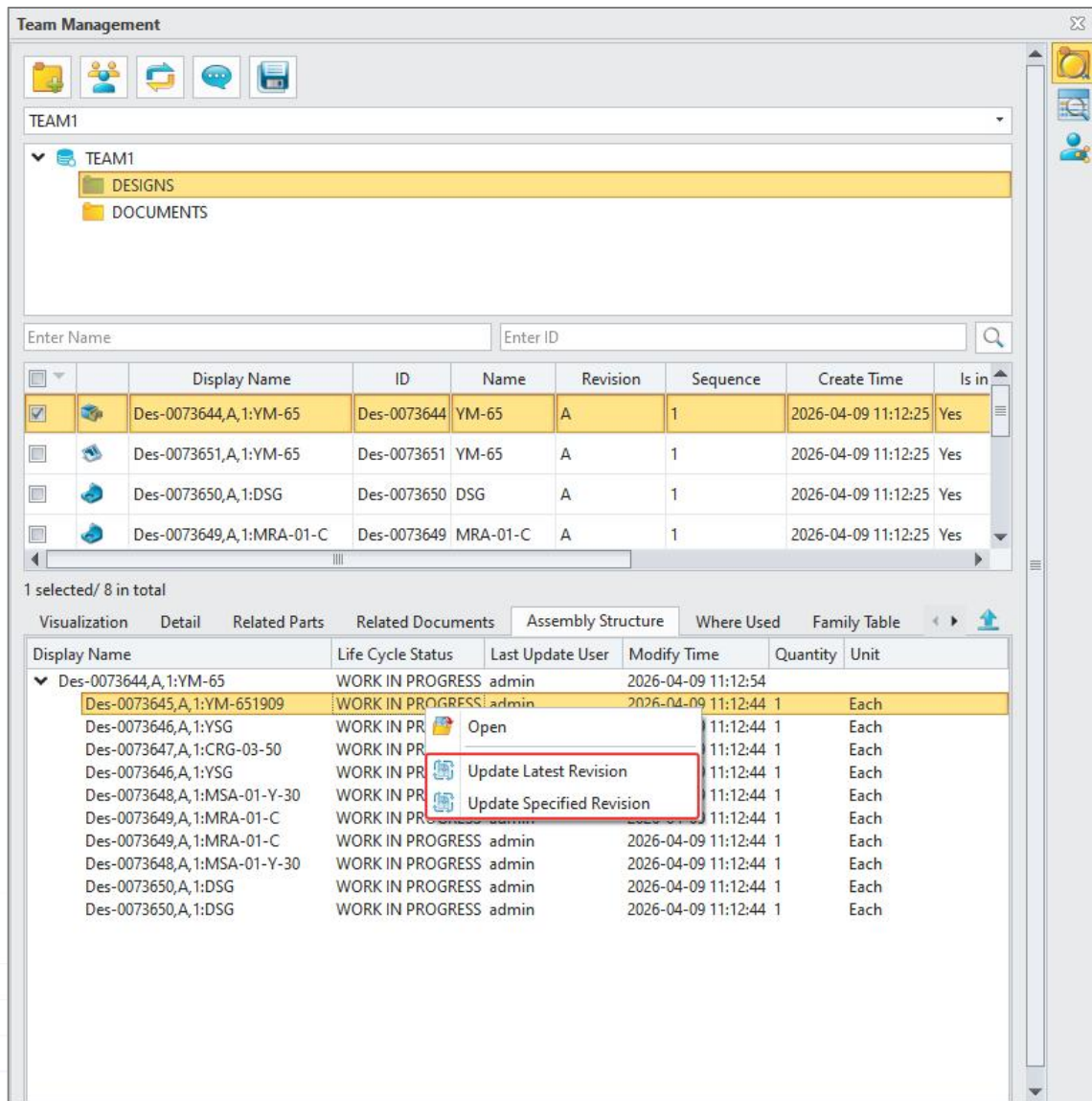
- ✓ Through the system preference Design.Structure.AccurateSequence.Enable supports the configuration version rule. When the preference value is "+", the assembly will load components according to the version saved last time; when the preference value is "-", the assembly will load components according to the latest version.



- ✓ For assemblies opened in ZW3D, you can update the component loading version content in the design structure tree, either to the latest version or a specified version.



- ✓ You can update the version of assembly components directly in the system, either to the latest version or a specified version.



【Where is it】

Web Management Client >> System >> System Extensions >> Preference Config

Manager >> Design Structure >> Load Latest Version

Manager >> Design Structure >> Load Specified Version

Manager >> Design Structure >> Load Latest Sub-Files

Manager >> Design Structure >> Load Last Submitted Sub-Files

Manager >> Design Structure >> Return to Last Loaded Sub-Files

Zwteammate >> Project Collaboration Area >> Assembly Structure >> Update

Latest Revision

[Zwteammate >> Project Collaboration Area >> Assembly Structure >> Update](#)

Specified Revision

[Zwteammate >> Project Collaboration Area >> Assembly Structure >> Update All](#)

Sub-File

6.1.2 ★ New Third-Party PLM Import Function

The new version utilizes Teammate as middleware for ZW3D's enterprise-level PLM integration, enabling seamless model import from third-party PLM systems into ZW3D to facilitate design reuse or borrowing.

【What users can do】

- ✓ The new version now supports integration with Teamcenter. You can enter component IDs to import Teamcenter models with matching IDs into Teammate.
- ✓ The new version now supports integration with Windchill. Users can enter a model ID to import identical models from Windchill into Teammate.

【Where is it】

[ZWTeamate >> Project Collaboration Area >> Import from Teamcenter/Windchill](#)

6.1.3 ★ New Checkin Third-Party PLM Function

The new version utilizes Teammate as middleware for ZW3D's enterprise-level PLM integration, enabling ZW3D models to be submitted to third-party PLM systems via Teammate.

【What users can do】

- ✓ The new version now supports integration with Teamcenter, enabling complete submission of Teammate shared area models to Teamcenter.

- ✓ The new version now supports integration with Windchill, enabling complete submission of Teammate shared area models to Windchill.

【Where is it】

[ZWTeammate >> Project Collaboration Area >> Checkin Teamcenter/Windchill](#)

6.1.4 Optimize Large Assembly Upload/Download Performance

Managing large assemblies is often a key challenge affecting collaborative design efficiency. The new version effectively reduces upload and download times for large assemblies by optimizing attribute writing timing, enhancing batch processing capabilities, and improving local cache retrieval mechanisms.

【What users can do】

- ✓ The time required to retrieve assembly models from Teammate and open them in ZW3D has decreased by approximately 20% compared to the previous version.
- ✓ The time required to upload a brand-new assembly model from local to Teammate has been reduced by approximately 20% compared to the previous version.

【Where is it】

[ZWTeammate >> Project Collaboration Area >> Open as Read-Only](#)

[ZWTeammate >> Project Collaboration Area >> Open Editing](#)

[ZWTeammate >> Search For Design >> Open as Read-Only](#)

[ZWTeammate >> Search For Design >> Open Editing](#)

[ZWTeammate >> Submit to Shared Area](#)

6.1.5 New Automatic Upload Mechanism

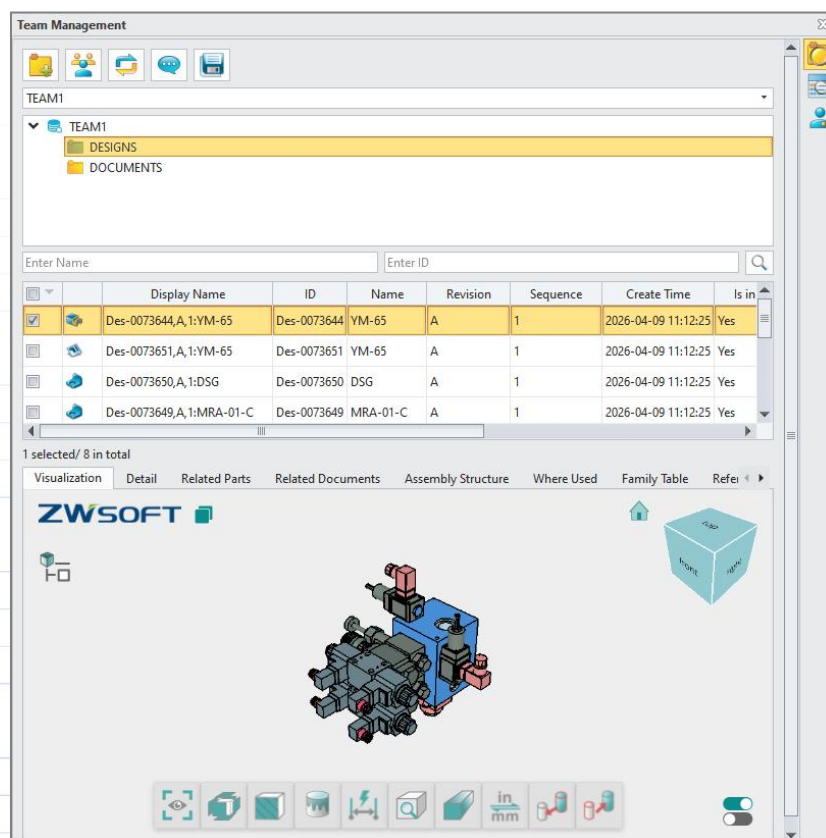
For designs in the workspace, the latest files can be retrieved from the local cache and automatically uploaded to the server. The upload interval can be configured in the PLMconf.xml file.

【Where is it】

Web Management Client >> System >> System Extensions >> System Settings >> ZW3D Config File

6.1.6 Optimize ZW3D Model Preview Function

The new version upgrades the ZW3D model preview from thumbnail to HTML format, delivering clearer visual previews.



【Where is it】

ZWTeamate >> Project Collaboration Area >> Visualization

[ZWTeammate >> Search For Design >> Visualization](#)

6.1.7 New Forced Discard Edit Function

When submitting a design to the shared area, if another user is editing it, the current user can force the editing status to cancel without retaining the edited content, provided permissions allow.

【Where is it】

[ZWTeammate >> Submit to Shared Area >> Force Discard Edit](#)

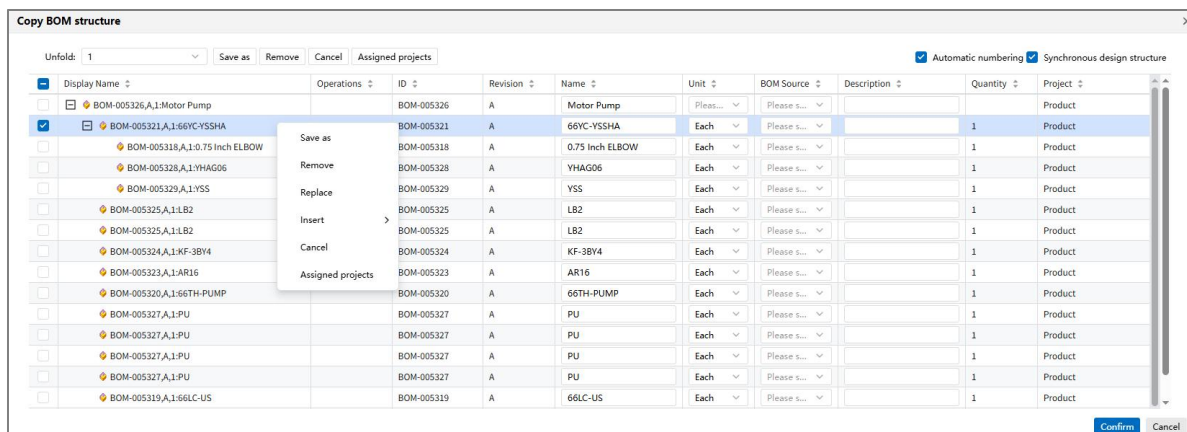
6.2 Teammate Web

6.2.1 ★New Copy BOM Function

In series production, variant design, and top-down development frameworks, rapidly replicating a standalone new BOM from existing mature BOM can significantly reduce redundant BOM creation efforts.

【What users can do】

- ✓ Create a new BOM by replicating the existing one.
- ✓ The replication process supports editing operations for the new BOM. Users can: generate new materials by referencing existing BOM rows through "Save As"; insert new materials under BOM rows via "Insert New Part"; add system-existing materials to BOM rows using "Insert Existing Part"; remove BOM rows from the new BOM with "Remove"; or replace BOM row materials with alternative system materials through "Replace". Newly added materials in BOM rows feature an automatic coding mechanism and can be assigned to projects.
- ✓ When copying the BOM, the assembly structure can also be copied synchronously.



【Where is it】

Web User Client >> Part >> BOM >> Copy BOM

Web User Client >> NAV >> BOM >> Copy BOM

6.2.2 Optimize Bom Drives ZW3D Model Function

The new version has removed the "Drive Structure" attribute from BOM rows, with deleted content including interaction and data control rules. Additionally, a new attribute "Controlled by ZW3D" has been introduced for materials. When executing BOM-driven ZW3D assembly structures, for materials controlled by ZW3D, if no model type design exists, the system will automatically create an empty model file and place it at the origin of the assembly space.

The BOM-driven ZW3D assembly structure function has been modified to execute asynchronously in the background.

【Where is it】

Web User Client >> Part >> BOM >> BOM Drives ZW3D Model

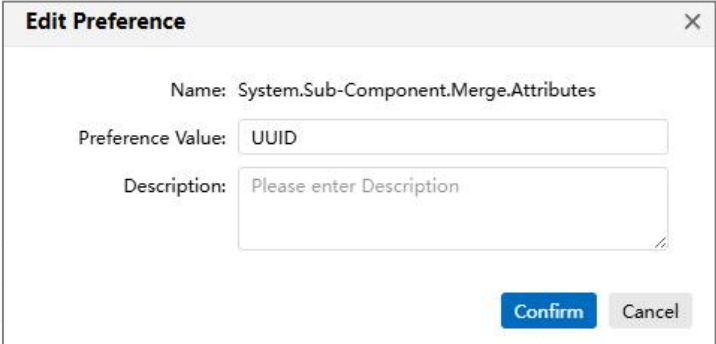
Web User Client >> NAV >> BOM >> BOM Drives ZW3D Model

6.2.3 Optimize Merge Same Children Function

Add the system preference System.Sub-ComponentMergeAttributes. This preference allows you to configure merge rules for the number of BOM rows.

【What users can do】

- ✓ Write the attribute names into this preference. Separate multiple attribute names with commas. When merging BOM row data, determine whether the number of BOM rows can be summed based on whether these attribute values are identical. For example, merge quantities only if configuration numbers are the same, or merge quantities only if configuration numbers, names, and versions are identical, etc.



【Where is it】

Web Management Client >> System >> System Extensions >> **Preference Config**

Web User Client >> Parts >> BOM >> **Merge Same Children**

Web User Client >> NAV >> BOM >> **Merge Same Children**

6.2.4 New Control Rule When BOM Editing

New constraints: When a BOM line is controlled by ZW3D, modifying the quantity of materials in that line is not permitted.

Basic Information		BOM	Related Designs	Related Documents	Replacement Parts	Where Used	Related Changes	History	Related Workflows
Please enter Name		Please enter ID		1					
									
☐ Display Name	Name	Life Cycle Status	Quantity	Unit	BOM Line Number				
☐  BOM-699129,A,1:Motor Pump-1	Motor Pump-1	WORK IN PROGRESS							
☐  BOM-005332,B,1:SR-15-TH	SR-15-TH	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005333,A,1:DH-25-46	DH-25-46	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-824533,A,1:66LC-US-1	66LC-US-1	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005325,A,1:LB2	LB2	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005325,A,1:LB2	LB2	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005324,A,1:KF-3BY4	KF-3BY4	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005323,A,1:AR16	AR16	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005327,A,1:PU	PU	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005327,A,1:PU	PU	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005327,A,1:PU	PU	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005327,A,1:PU	PU	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  BOM-005319,A,1:66LC-US	66LC-US	WORK IN PROGRESS	1	Each	Please enter BOM Li...				
☐  SSD-0072051,A,1:66TH-PUMP-2	66TH-PUMP-2	WORK IN PROGRESS	1	Each	Please enter BOM Li...				

【Where is it】

Web User Client >> Part >> BOM

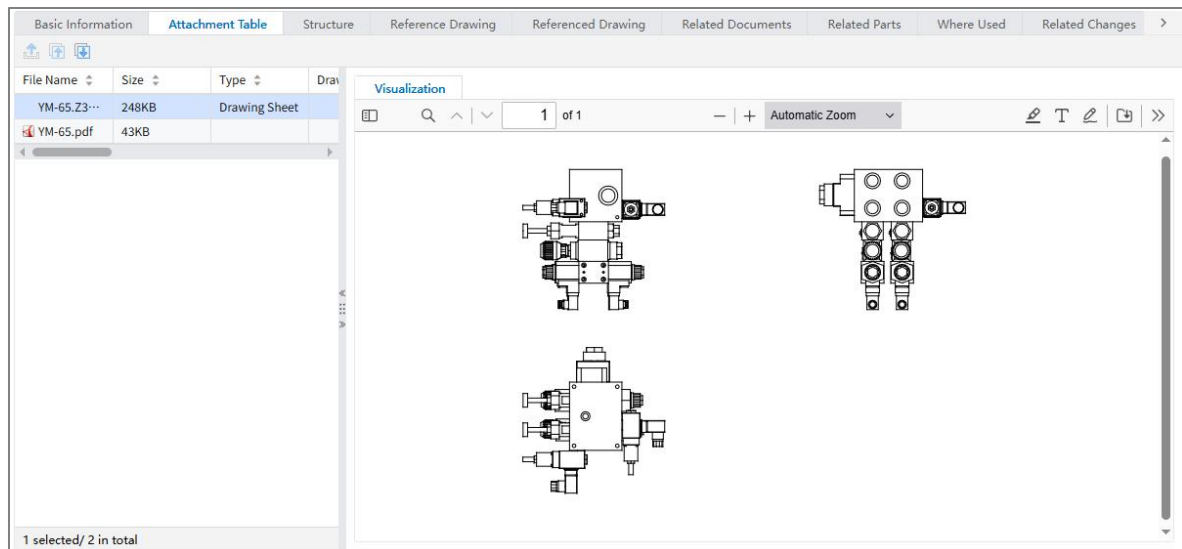
Web User Client >> NAV >> BOM

6.2.5 New ZW3D Drawing Sheet Preview

Supports online viewing of Z3DRW files.

【What users can do】

- ✓ You can open the lightweight file of ZW3D Drawing Sheet in a browser to view the content.
- ✓ You can add comments.



【Where is it】

Web User Client >> Design >> **Online Preview**

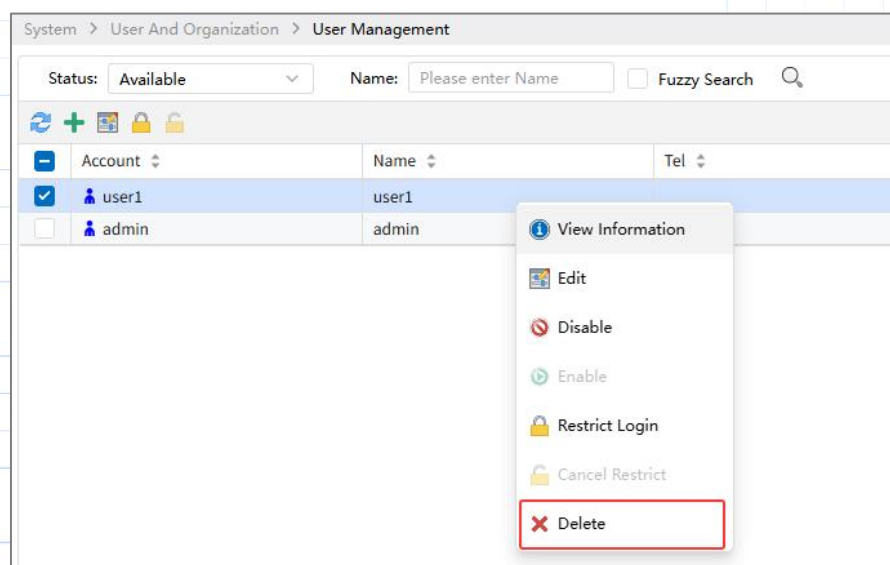
Web User Client >> Design >> Attachment Table >> **Online Preview**

Web User Client >> NAV >> **Online Preview**

Web User Client >> NAV >> Attachment Table >> **Online Preview**

6.2.6 New Delete User Function

You can delete system users, and this is a physical deletion.



【What users can do】

- ✓ Delete system users and their accounts
- ✓ Sync and delete user tasks;
- ✓ Sync and delete the user's favorites.

【Notes】

- Users cannot delete private business data.

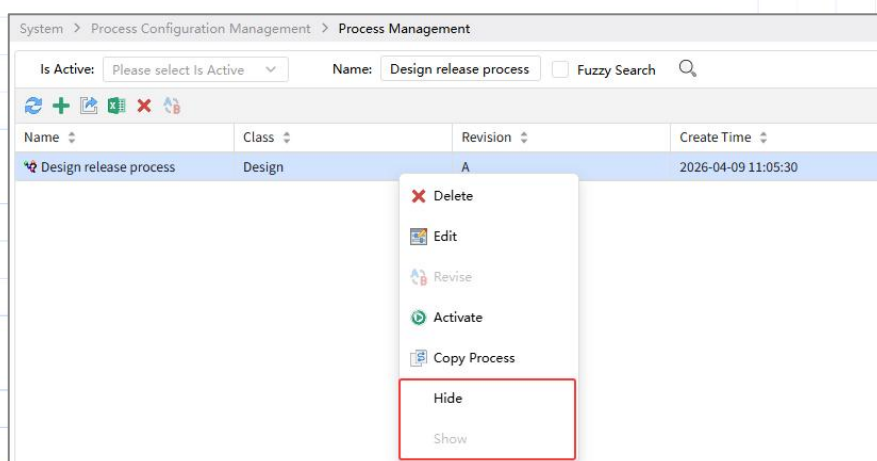
【Where is it】

Web Management Client >> System >> User and Organization >> User Management >> Delete

6.2.7 New Workflow Hide and Show Function

You can set workflows to hidden status, which prevents users from selecting and using them.

Supports setting workflows to display mode, where users can select and use workflows.



【Where is it】

Web Management Client >> System >> Process Configuration Management >>

[Process Management >> Hide](#)

[Web Management Client >> System >> Process Configuration Management >>](#)

[Process Management >> Show](#)

6.2.8 Add Annotation Flag

Include comment flags in the attachment list. Attachments with comments are automatically marked as "Yes".

Basic Information Attachment Table Structure Reference Drawing Referenced Drawing Related Documents Related Parts Where Used Related Changes >											
File Name	Size	Type	Drawing Su...	Auth Tool	Auth Tool Ve...	Is Major	Modify Time...	Check Out U...	Is Annotated...	Sequence	Is In Vault
YM-65.Z3...	248KB	Drawing Sheet		ZW3D	2027	Yes	2026-04-09 ...	admin	No	1	Yes
YM-65.pdf	43KB			ZW3D		No	2026-04-09 ...	admin	Yes	1	Yes

【Where is it】

[Web User Client >> Design >> Attachment Table](#)

[Web User Client >> NAV >> Attachment Table](#)

6.2.9 New Search Error Code Function

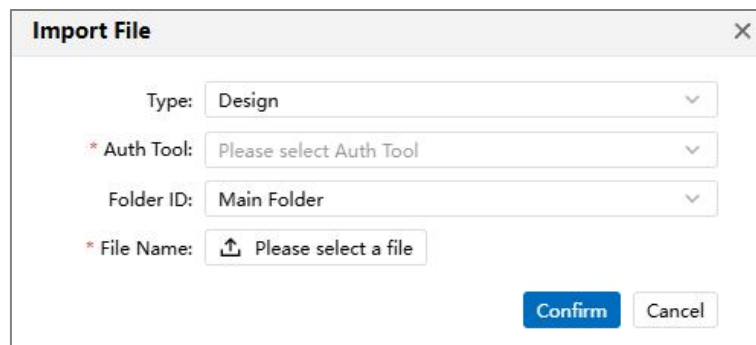
Supports retrieving specific system error information through error codes.

【Where is it】

[Login user >> Search error code](#)

6.2.10 New File Import Function

You can import multiple files from local storage to the system at once.



Import File

Type: Design

* Auth Tool: Please select Auth Tool

Folder ID: Main Folder

* File Name: Please select a file

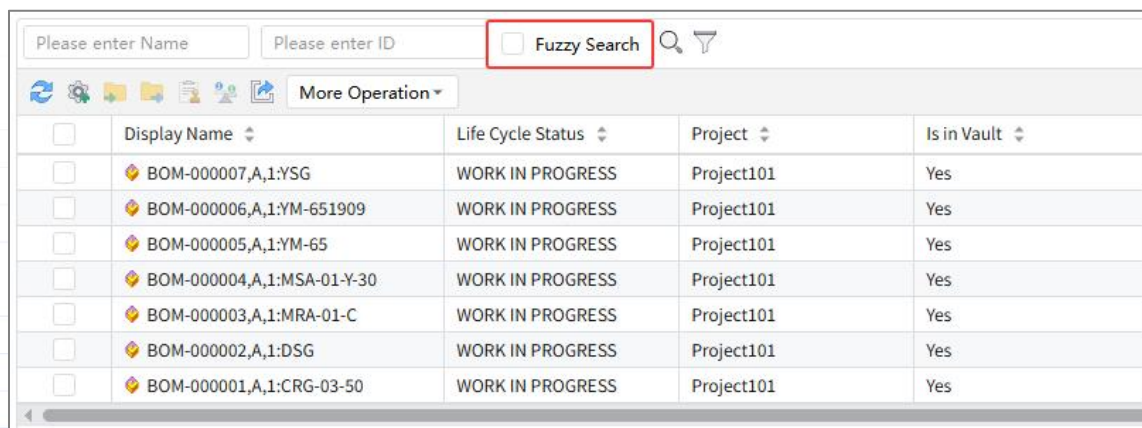
Confirm Cancel

【Where is it】

Web User Client >> NAV >> Folder >> More Actions >> Import File

6.2.11 Optimizing Fuzzy Search Function

To enable quick queries for business objects, remove the * expression for fuzzy queries and add a fuzzy query checkbox. When checked, fuzzy matching can be performed based on object ID or name.



Please enter Name Please enter ID ☐ Fuzzy Search

More Operation

	Display Name	Life Cycle Status	Project	Is in Vault
☐	BOM-000007,A,1:YSG	WORK IN PROGRESS	Project101	Yes
☐	BOM-000006,A,1:YM-651909	WORK IN PROGRESS	Project101	Yes
☐	BOM-000005,A,1:YM-65	WORK IN PROGRESS	Project101	Yes
☐	BOM-000004,A,1:MSA-01-Y-30	WORK IN PROGRESS	Project101	Yes
☐	BOM-000003,A,1:MRA-01-C	WORK IN PROGRESS	Project101	Yes
☐	BOM-000002,A,1:DSG	WORK IN PROGRESS	Project101	Yes
☐	BOM-000001,A,1:CRG-03-50	WORK IN PROGRESS	Project101	Yes

【What users can do】

- ✓ If you do not select "Fuzzy Search", the system will search for the entered numbers and names using exact matching, which is faster.
- ✓ Select "Fuzzy Search" to search for input numbers and names using fuzzy matching.

【Where is it】

Web User Client >> NAV >> Fuzzy Search

Web User Client >> Part >> Fuzzy Search

Web User Client >> Design >> Fuzzy Search

Web User Client >> Document >> Fuzzy Search

Web User Client >> Change Request >> Fuzzy Search

Web User Client >> Change Notice >> Fuzzy Search

6.2.12 Optimize Folder Sorting Mechanism

The default folder sorting order has changed from update time to name.

【Where is it】

Web User Client >> NAV >> Folder