ZW3D WHAT'S NEW

V 2023

ZWSOFT CO., LTD.(Guangzhou)

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ZW3D™ V2023 What's New

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Highlight of ZW3D 2023

Basic：

New Multi-Print

Pick Interaction Improvement

Datum Improvement

Thickness Analysis Improvement

Color Improvement

CAD：

New Smart Constraint Inference

Face Offset Supports G2

Spiral Helix Improvement

New Revolve Flange

Lofted Flange Improvement

New Smart Fastener

New Envelop

2D/3D Sketch External Reference Improvement

Sketch Curve Continuity Inquire Improvement

3D Sketch Improvement

Loft Improvement

New External Geom Copy

Blend Face Improvement

Pattern Feature

UDF Improvement

New Double Bend Flange

New Universal Joint

New Pulley Assembly Feature

New Motor in Assembly Animation

Drawing Sheet：

New 3D Crop

View Projection Efficiency Improvement

New Batch Sheet Format Attributes

Mold Design：

New Engrave

New Oil Groove

New Load Asm-Tree in Mold

Workpiece Improvement

Layout Improvement

Standard Part Mechanism Improvement

CAM：

Tool Speed and Feed Overall Upgrade

Tool Library Overall Upgrade

New Multi Work Station

New Reference Tool in 2 Axis Operation

FMS Support 5-axis Index Milling Simulation

FMS Support Auto On/Off RTCP

Speed Up QM Rough Operation Calculation

Support Taper Hole and Taper Boss Milling

2D Pocket Operation Supports External Lead-in Open Pocket

New Follow Part Path Pattern in 2X Contourcut operation

Profile Feature Curve Attribute Optimization

Update User Customized Tool Functionality

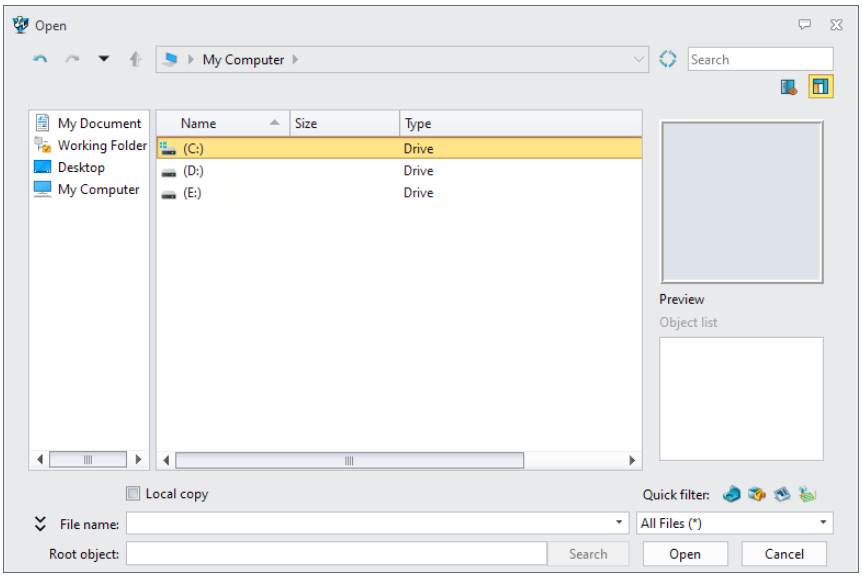
Note: *Important enhancements in this article are marked with★*

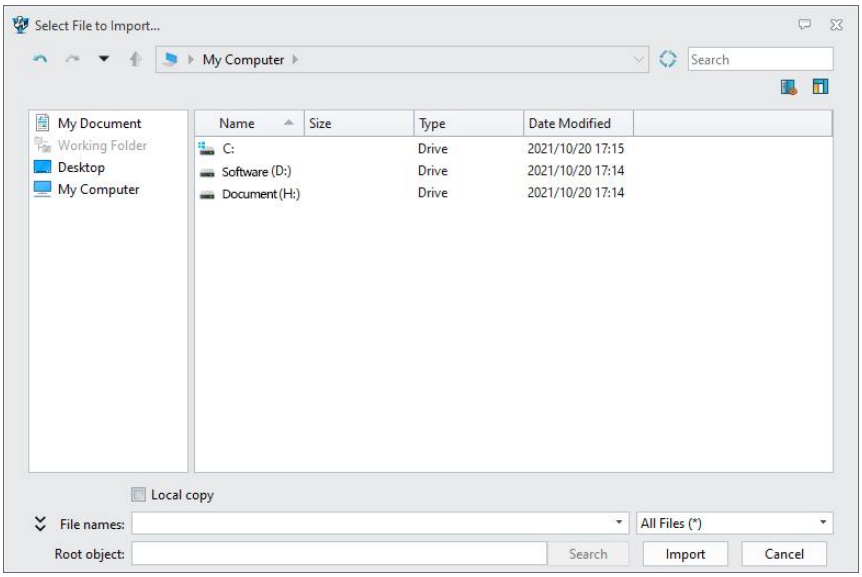
1 Basic

1.1 File Dialogue Improvement

1.1.1 Open/Import Panel Improvement

We improved Open/Import panel which supports more functions such as file search, quick filter, and shortcut folder in ZW3D 2023.



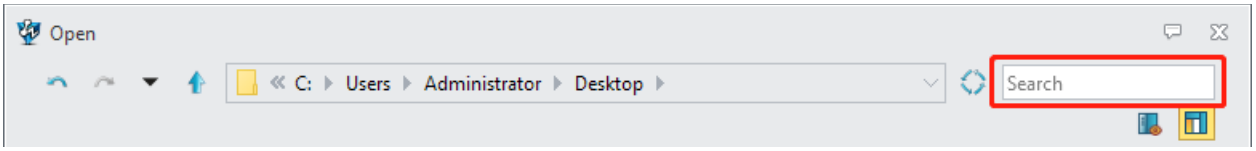


➔ Where it is

Any Environment >> File >> Open/Import

1.1.2 New File Name Search

User can search file name by partially searching a file name that contains key words in the open panel.



1.1.3 New Quick Filter

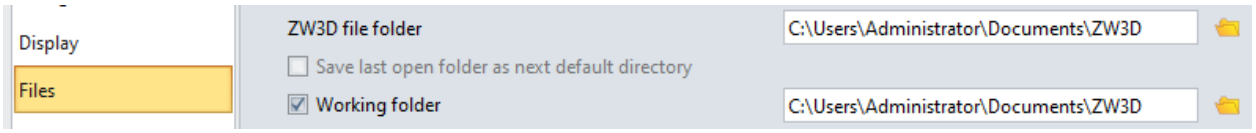
- Default file type as: All files, only display all format files that are supported and do not display the files that cannot be opened.
- Quick filters response to: Part, Assembly, Drawing Sheet and CAM Plane. Filter only self-owned format.
- Bind quick filter with file type. When user switches to quick filter, file type switches to all files. When user switches file type to other formats, quick filters are all deactivate.

1.1.4 New Shortcut Folder Function

1.1.4.1 Working Folder

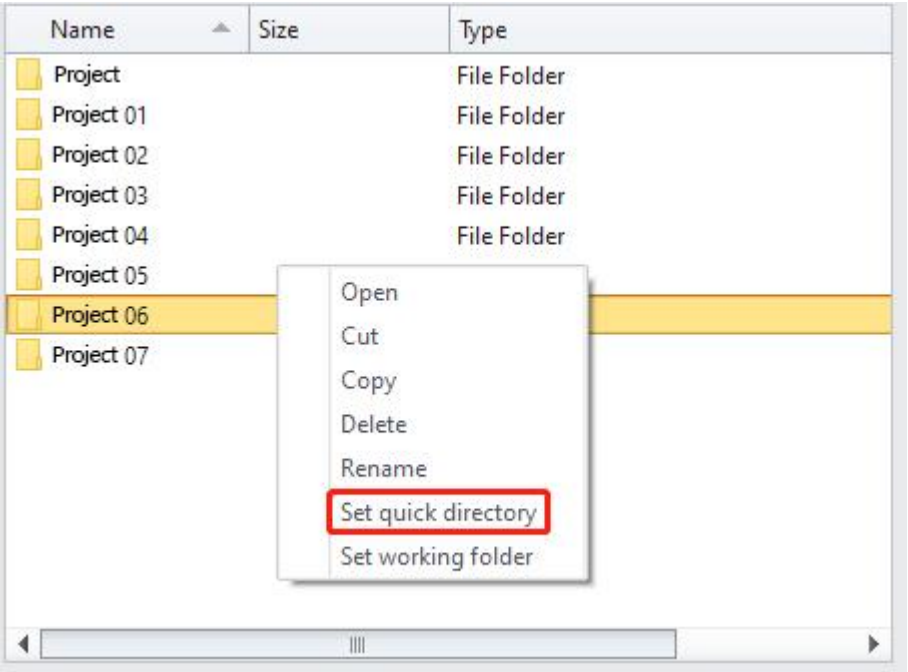
User can set folder as working folder by context menu.

- After working folder is set, open panel default working folder as working folder path.
- If working folder is unset, open panel default directory as the last open file directory path.
- User can uncheck working folder setting in “Files” configuration.



1.1.4.2 Custom Shortcut Folder Context Menu

- User can drag and adjust folder up or down by left clicking mouse.
- User can set folder as shortcut folder or delete it through the context menu.

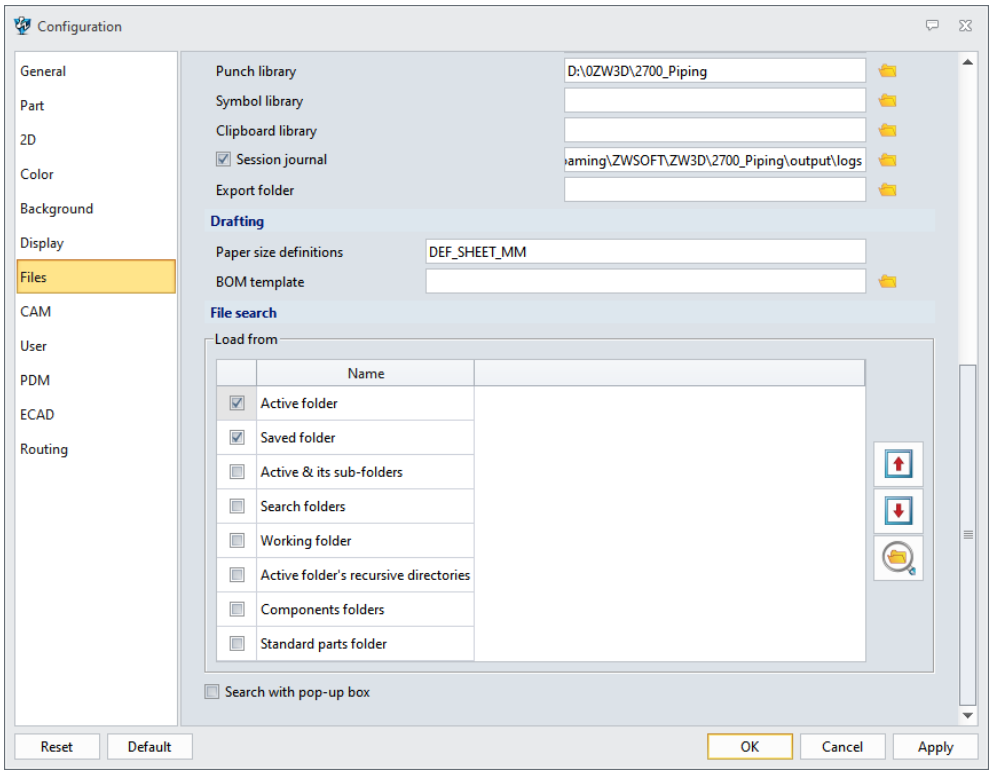


1.2 File Path Management Improvement

1.2.1 File Search in Configuration

"Load setting" was transferred from "Open" panel to "Configuration" panel. User can customize the file search directory and the search priority. If it is configurated, it can be saved after ZW3D closed. The relevant functions of ZW3D in file loading and positioning all follow the path and sequence of loading Setting.

ZW3D will only search files that are checked in the list. The search order applies to top-to-down sequence. The upper path in the panel will be searched in priority. User can adjust the order by "Move up" and "Move down" buttons.



1.2.2 Search Path Expansion

We added three search paths: Active folder's recursive directories, Components folders, and Standard parts folder.

Active folder: Activate the current folder and the option cannot be cancelled.

Saved folder: The saved folder path in the file.

Active & its sub-folders: The current activated folder and its sub-folder.

Search folders: List out the folder path in ZW3D search path panel.

Working folder: The working folder that set by user.

Active folder's recursive directories: Activate the path for recursive folder.

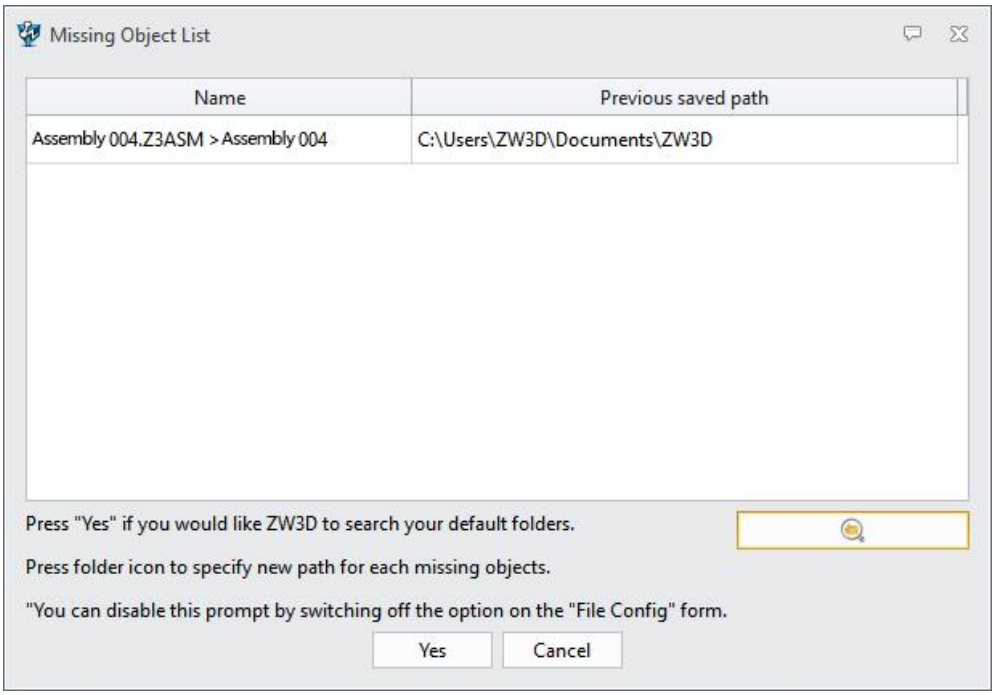
Components folders: The folder that locates the components.

Standard parts folder: The folder of standard parts.

	Name
☑	Active folder
☑	Saved folder
☐	Active & its sub-folders
☐	Search folders
☐	Working folder
☐	Active folder's recursive directories
☐	Components folders
☐	Standard parts folder

1.2.3 Search with Pop-up Box

"Search with pop-up box" option was transferred from "Search path" panel to "File search" panel. Check this option to control whether pop-up windows display and collect all missing objects when discovering missing objects.



The popup windows improvements mainly include the following:

- The popup window at the end of the file opening process can list all the missing components at one time, which is convenient for users to understand all the missing contents and integration processing
- The popup windows interface can list all the missing object file names and their original paths.

- The popup windows interface provides search path file button and supports recalling search path as well as setting search path.

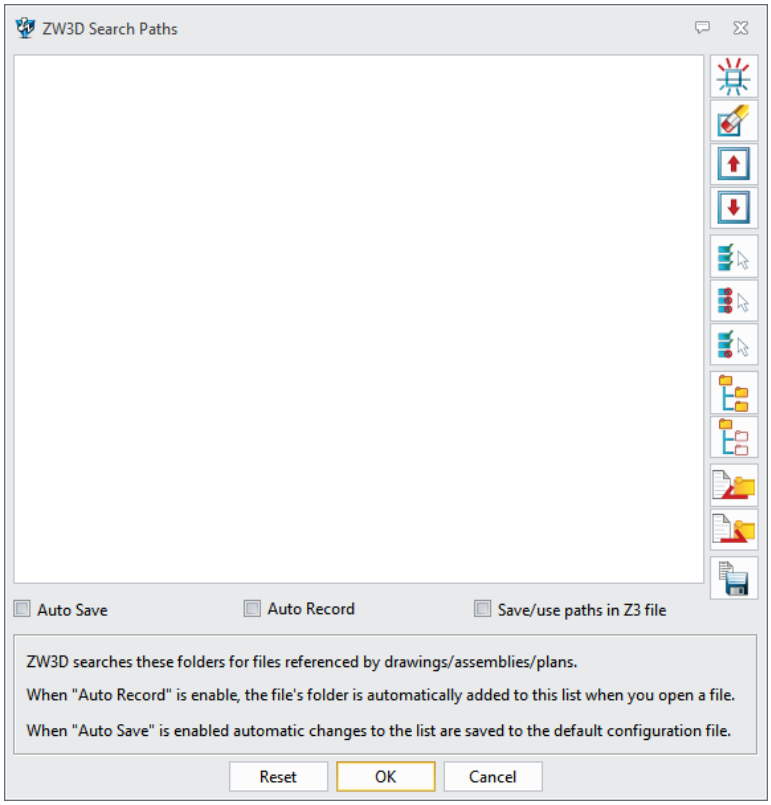
➔ **Where it is**

Configuration >> File >> File search

1.2.4 Search Path Adjustment

Search path can automatically record all open files' paths as the evidence to locate sequence files. To increase its accuracy, we adjusted the search path as follow:

- Close "Auto Save" "Auto Record" by default to avoid excessive use of past paths when using search paths.
- Remove "Auto Search" option.
- "Search Paths" list is modified to common one.
- "Search with pop-up options" is transformed to "File search" panel in configuration.

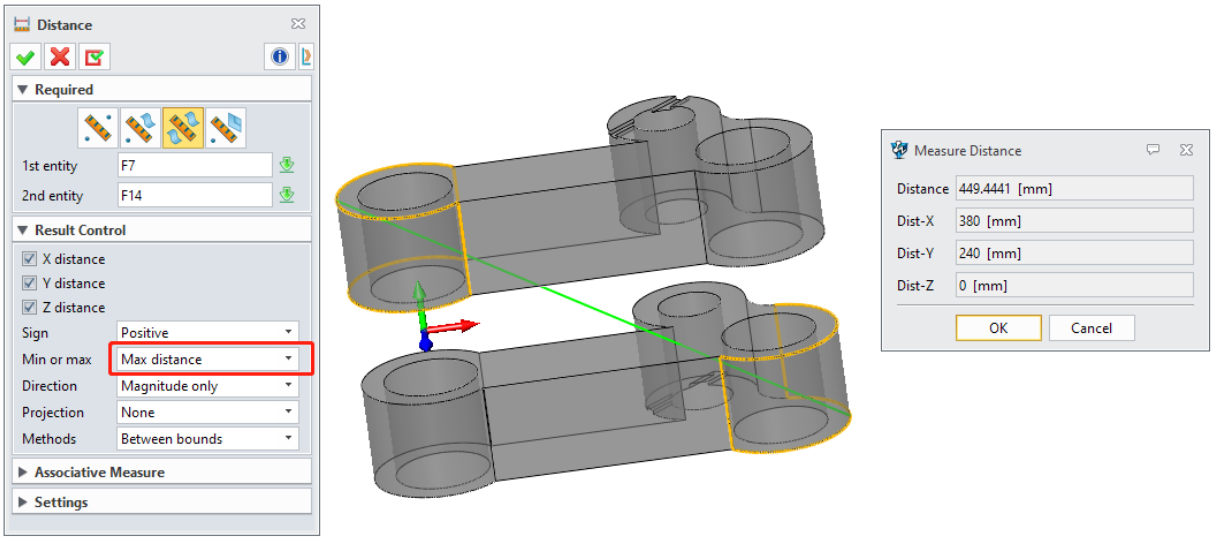


➔ **Where it is**

Menu Bar >> File >> File search >> ZW3D Search paths

1.3 Support Maximum Distance Measure

We added “Max distance” option to the distance measure, which was applied to the part environment, assembly environment, sketch environment, 3D sketch environment, drawing sheet environment and CAM plan environment. The two measure methods: Geometry to point, Geometry to geometry support the option.



➔ Where it is

Part/Assembly Environment >> Inquire>> Measure >> Distance

1.4 Echo Effect Improvement

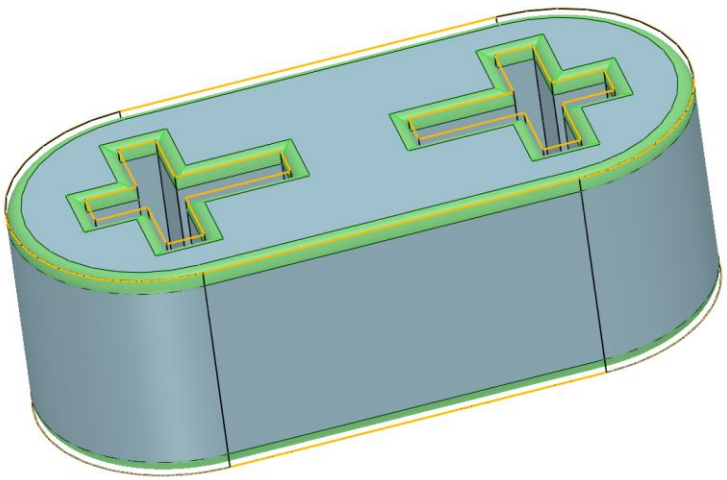
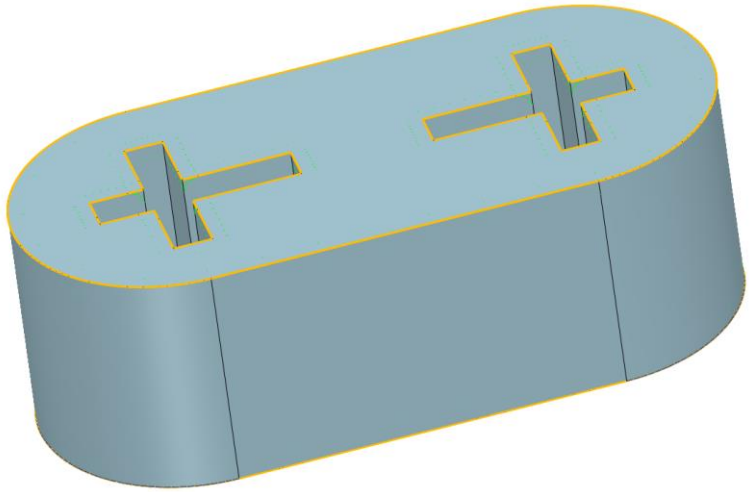
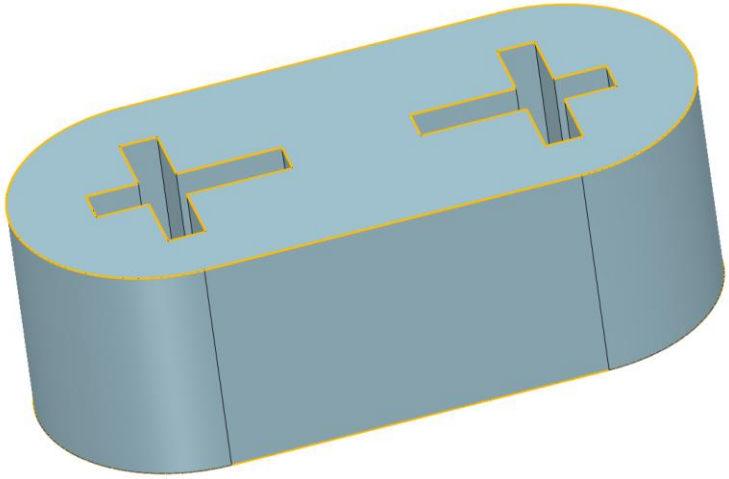
1.4.1 Modeling Command Echo Improvement

We added “Partial Echo” and “No Echo” effects.

Modeling command	Fillet	Chamfer	Draft	Face Offset
Partial Echo	Already have	New	New	New
No Echo	Already have	New	Already have	New

1.4.1.1 Echo Effect Display

Full Echo, Partial Echo, and No Echo effects are as follows:

Icon	Name	Effect
F	Full Echo	
P	Partial Echo	
O	No Echo	

➔ **Where it is**

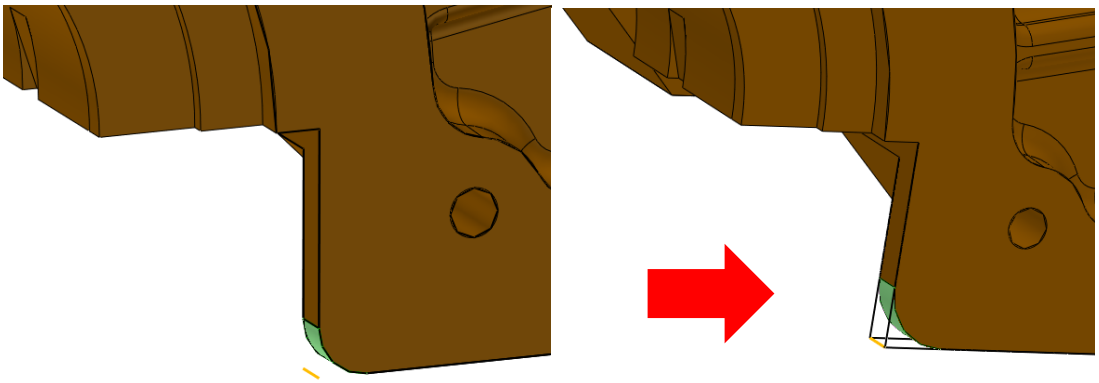
Part Environment >> Shape>> Engineering Feature >> Fillet/Chamfer/Draft

Part Environment >> Shape>> Edit Shape >> Face Offset

1.4.1.2 Full Echo Effect Improvement

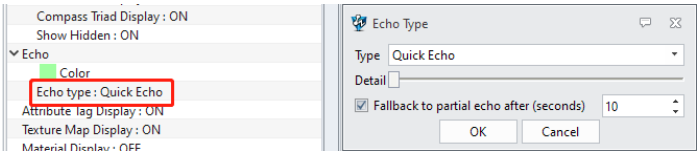
The selected part is displayed in highlight color while the modified part is displayed in echo color and the original part which picked is displayed in wireframe. Those can distinguish original part display and echo display easily.

The commands that improved real echo effect include Fillet, Chamfer, Draft, Face Offset and Shell.



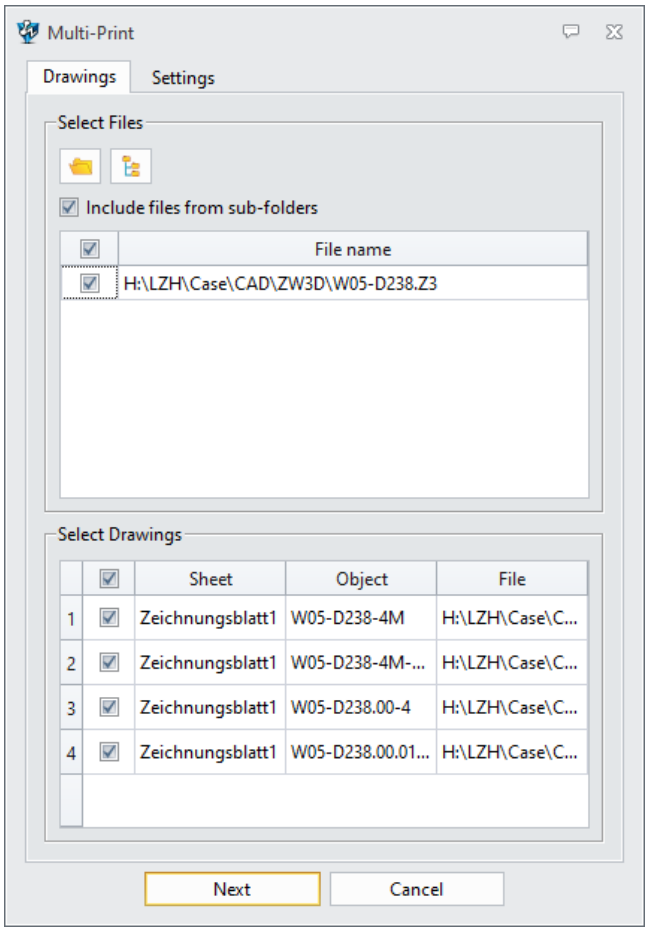
1.4.1.3 New Auto Switch Partial Echo Checkbox

Under the “Reach Echo” status, when partial echo consumed time exceeds certain time, the system will automatically switch to “Partial Echo” to reduce consumed time of modeling calculation to improve work efficiency.



1.5 ★New Multi-Print

We added “Multi-Print” to increase the efficiency of sheet printing in batch. Multi-Print supports printing multiple drawing sheets of multiple files at once. After the files are selected, the drawing sheets of files will be listed in the sheet list. Check the checkbox in front of the sheets for batch printing.



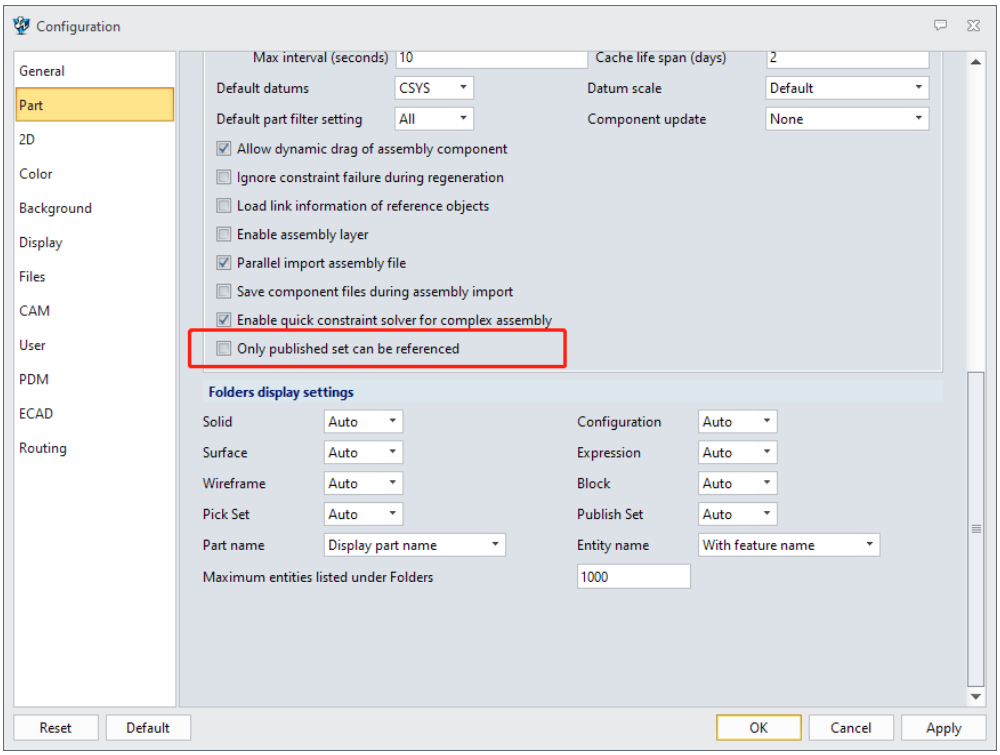
➔ Where it is

Menu Bar >> Multi-Print

1.6 Equation and Variable Browser

1.6.1 New Global Control Function

To reduce occurrence of cycle reference, we added the global control function to restrict reference in published objects. As for the unpublished objects such as geometry objects, other sheets cannot directly refer to other sheet objects through “Indirect reference” command. For example, in-place-edition, sketch environment cannot directly participate in geometry objects of other components.

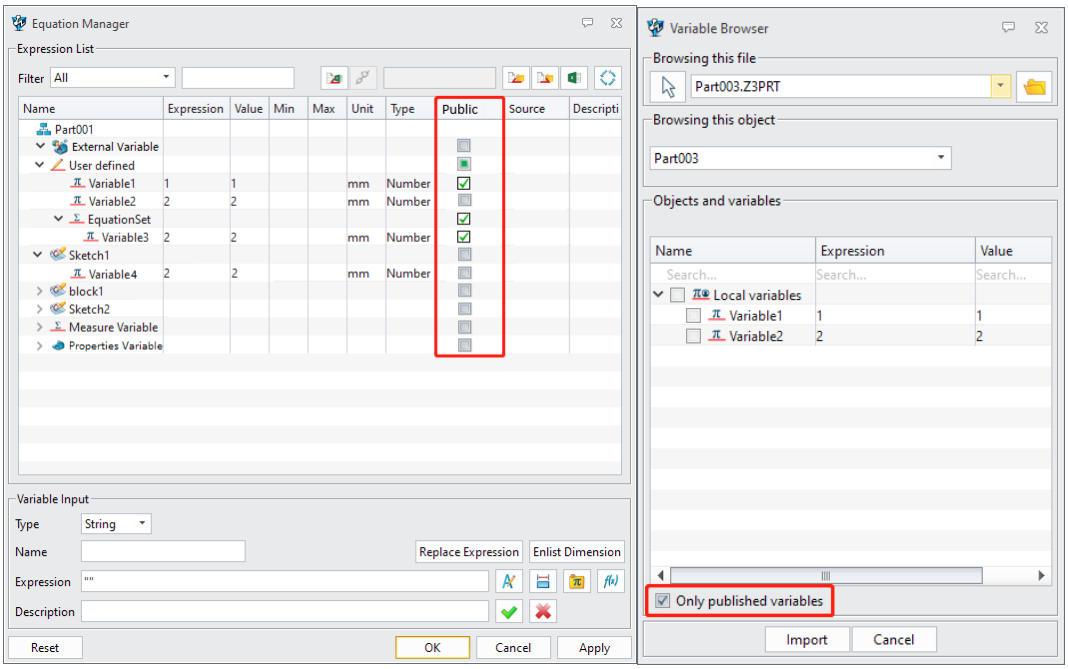


➔ **Where it is**

Part Environment >> Configuration >> Part

1.6.2 New Publish Variable Function

User can specify variables to publish in the variable list by Publish checkbox. Other drawing sheets can select and refer to published variable through the published variable option in the variable browser.

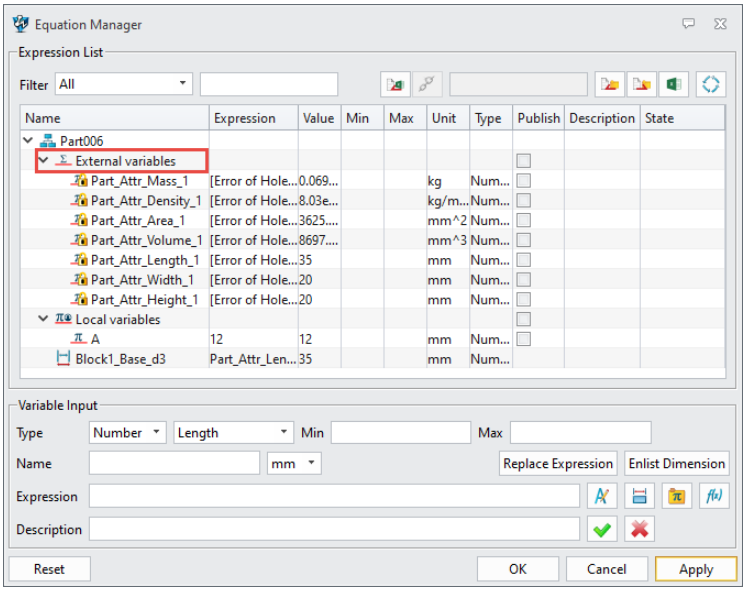


➔ **Where it is**

Part Environment >> Tools >> Equation Manager/Variables Browser

1.6.3 Equation Manager Function Adjustment

We added “External Variable” to list out all referenced external variables, which was placed in “Top”. We also optimized the layout of equation manager and added external variable node tree structure that could quickly locate and search when using it.



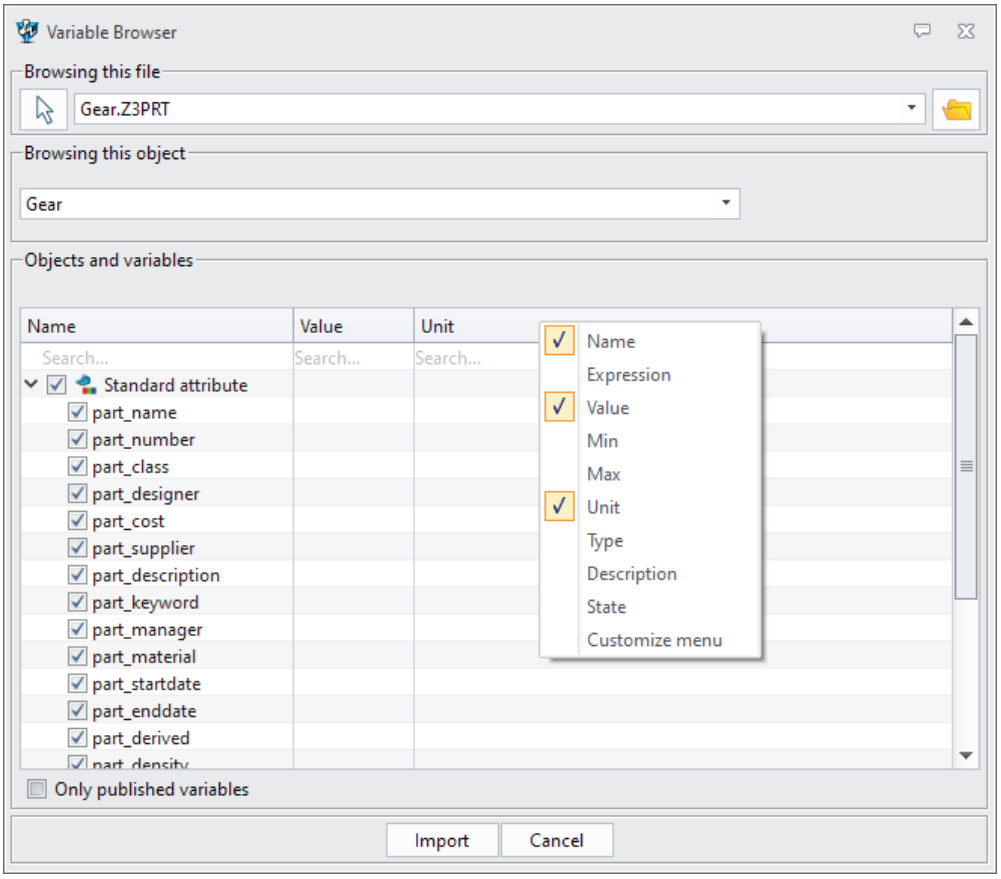
➔ **Where it is**

Part Environment >> Tools >> Equation Manager

1.6.4 Variable Browser Improvement

In ZW3D 2023, the variable browser has the following improvements:

1) Add the function of variable reading-in through selecting part in the assembly tree.
2) Expand the variable display contents and default display variable name, value, and unit.
3) Only display the parameter value under the current object instead of the parameter value under its sub-assembly node.
4) Add the search filter function to display column.
5) Display list structure is adjusted to tree structure.

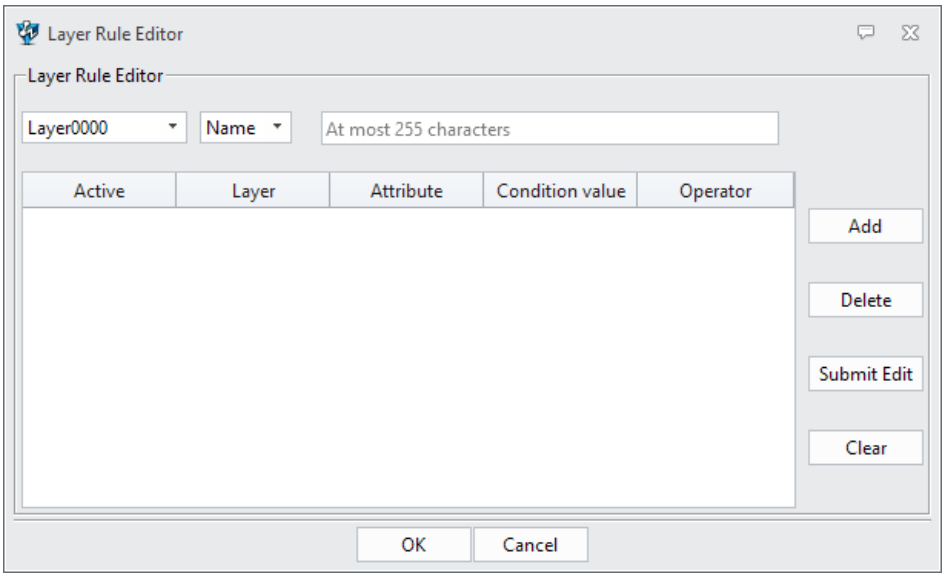


➔ **Where it is**

Part Environment >> Tools >> Variable Browser

1.7 New Layer Rule

Define layer rule according to attribute (such as type, name, and color). When the created objects match with the corresponding layer rules, the objects will be automatically moved to the corresponding layers.



➔ Where it is

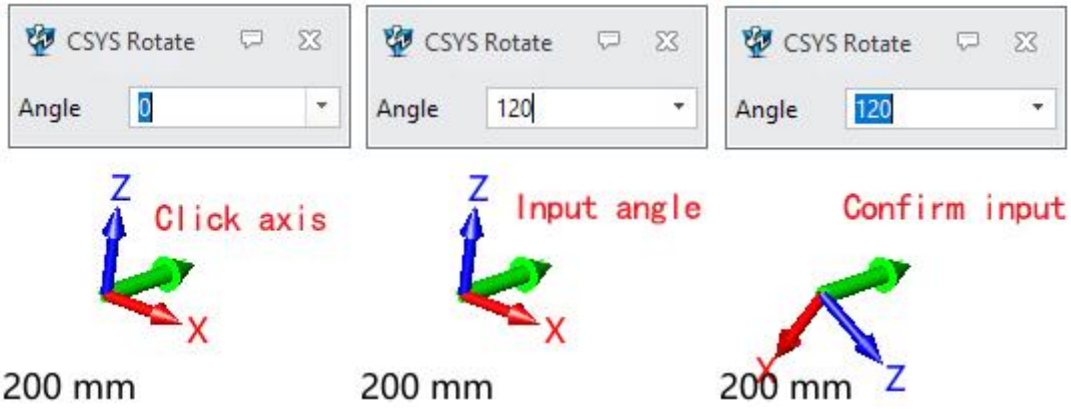
Part/Assembly/Drawing Sheet Environment >> Layer Manager >> Layer Rule

1.8 Ease of Use Improvement

1.8.1 3D Navigation Tag Supports Specifying Angle

To adjust view more quickly, user can input specified angle to rotate view through the 3D navigation tag at the lower left corner of the drawing region.

Left click in the axis of 3D navigation tag, and popup angle input box. The view will rotate according to the selected axis and the input angle.

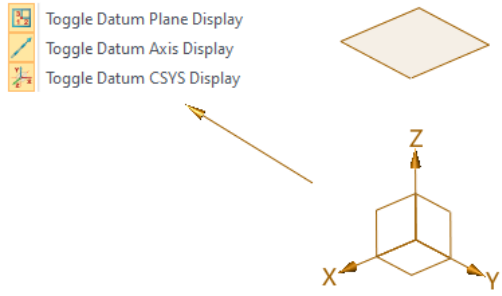


➔ **Where it is**

Part/Assembly Environment >> Drawing Region >> 3D Navigation Tag

1.8.2 New On/Off Datum Plane/Axis/CSYS

We added the switch to the display status of datum plane, datum axis, and datum CSYS on the DA toolbar to control whether display the datum element in whole.

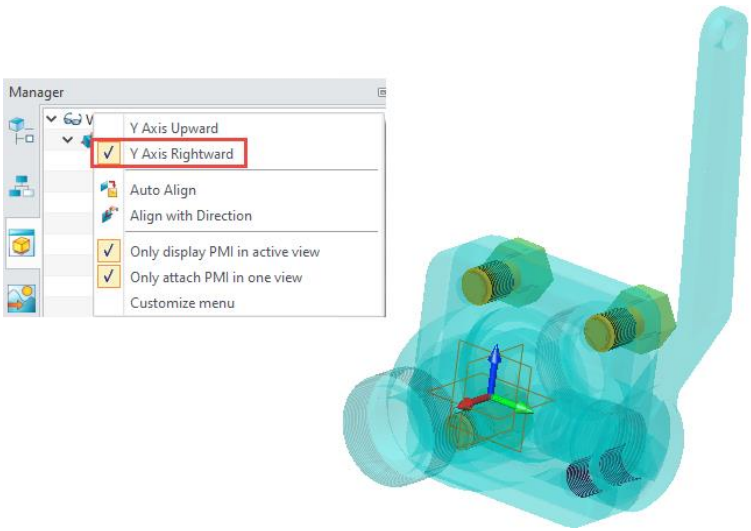
Turn on datum display, icons are highlighted, and datums are displayed	Turn off datum display, icons are not highlighted, and datums are hidden
	

➔ **Where it is**

Part/Assembly Environment >> DA Toolbar >> Toggle Datum Plane/Axis/CSYS Display

1.8.3 New Y Axis Rightward

In the view manager, we added “Y Axis Rightward” to control Z axis upward and Y axis rightward to achieve the initial coordinate that meets with right-hand rule.

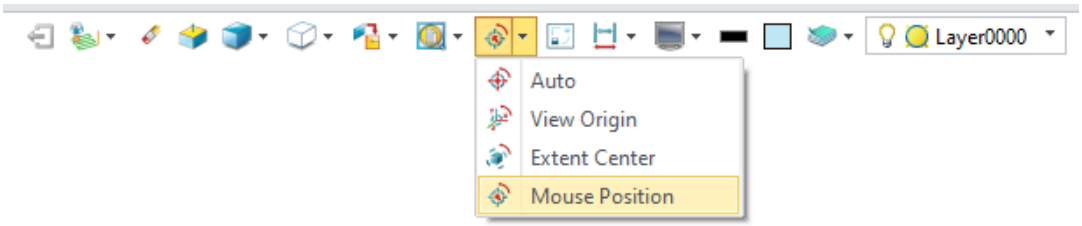


➔ **Where it is**

Part/Assembly Environment >> View Manager >> View Context Menu >> Y Axis Rightward

1.8.4 DA Toolbar Support Rotation Center

We added switch buttons of selection center on the DA toolbar in ZW3D 2023, including Auto, View Origin, Extent Center, and Mouse Position to help user switch the view of rotation center.



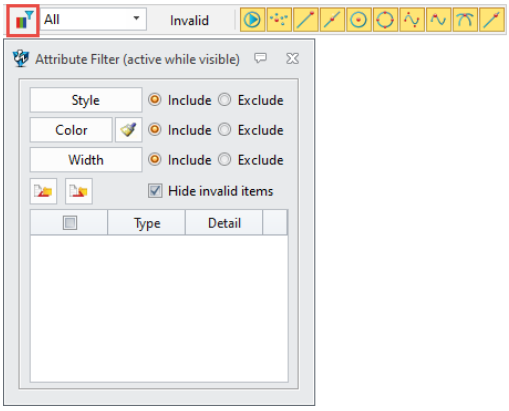
➔ **Where it is**

Part/Assembly Environment >> DA Toolbar >> Rotation Center

1.9 ★Pick Interaction Improvement

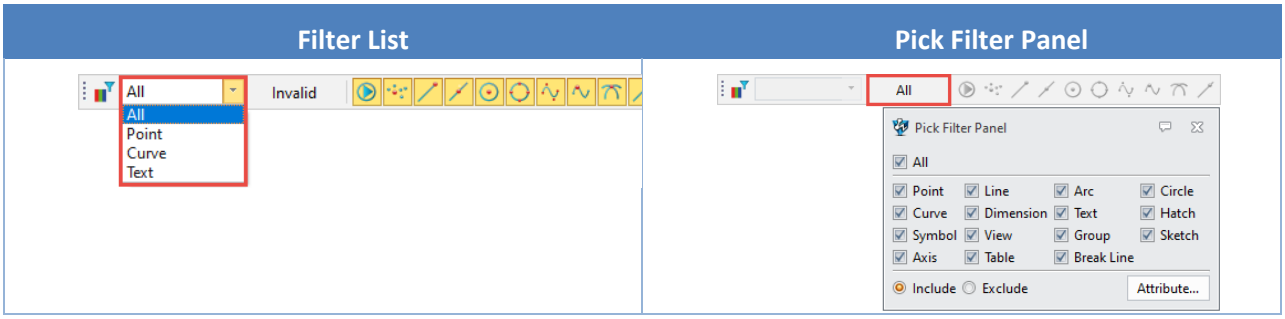
1.9.1 Sketch/Drawing Sheet Environment Filter Improvement

We added Filter to the sketch/drawing sheet environments. The upgraded attribute filter in the sketch/drawing sheet environments is the same as the part environment.



We added filter list based on the original filter panel to the sketch/drawing sheet environments and opened it to specific commands. The filter list is drop-drown menu, which only supports single selection, and it is the same as the part environment.

The filter list and the pick filter panel are incompatible, meaning that if one works and the other does not. Without disabling any commands, the pick filter panel works while the filter list turns grey. After enabling, the specific command starts the filter list and at this point the pick filter panel displays “Invalid” and becomes grey.



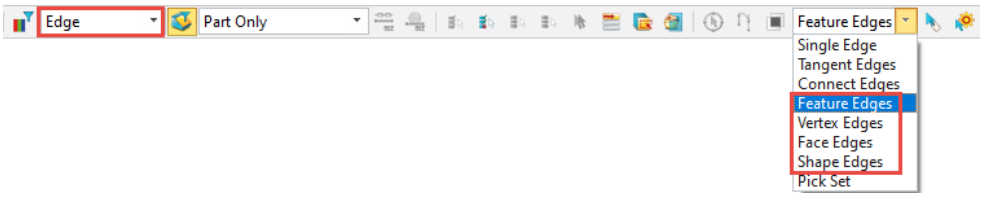
➔ **Where it is**

Sketch/Drawing Sheet Environment >> Pick Toolbar>> Attribute Filter/Filter List/Pick Filter Panel

1.9.2 Support More Intentional Selection Behaviors

1.9.2.1 Edges Intentional Selection Behavior

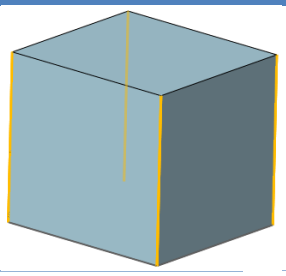
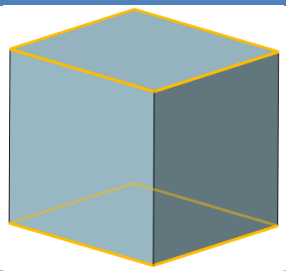
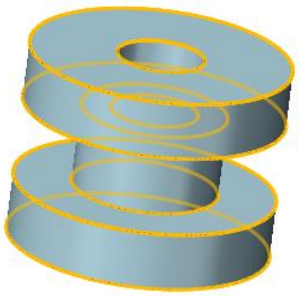
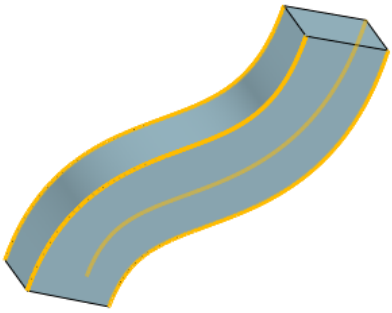
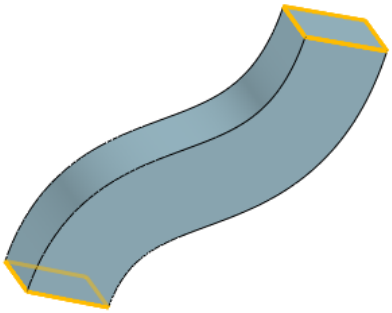
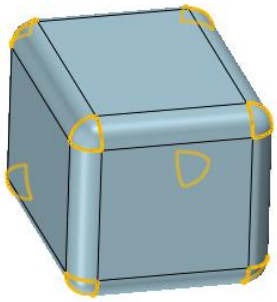
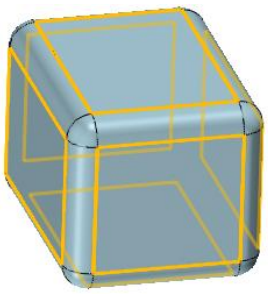
To improve edge selection efficiency, we added four intentional selection behaviors in ZW3D 2023 including **Feature Edges**, **Vertex Edges**, **Face Edges** and **Shape Edges**. When Edge is selected in the filter list, the four pick rules are displayed in the drop-drown list.

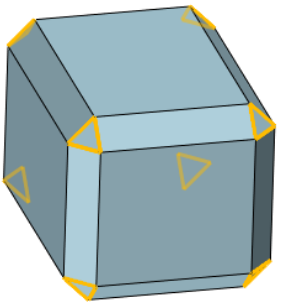
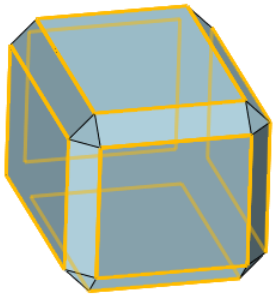
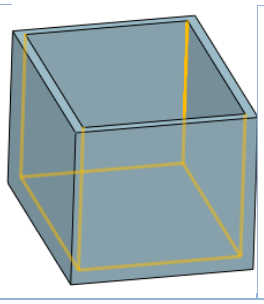
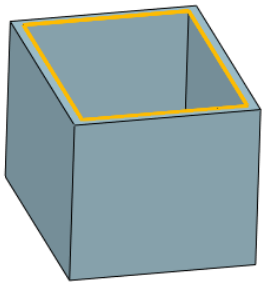
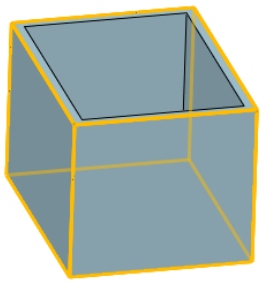


1.9.2.1.1 Feature Edges

The "**Feature Edges**" pick rule is based on feature automatically selecting edges.

"Feature Edges" Picking Effect Display

Feature	Feature Edges Picking Effect	
Extrude		
Revolve		
Sweep		
Fillet		

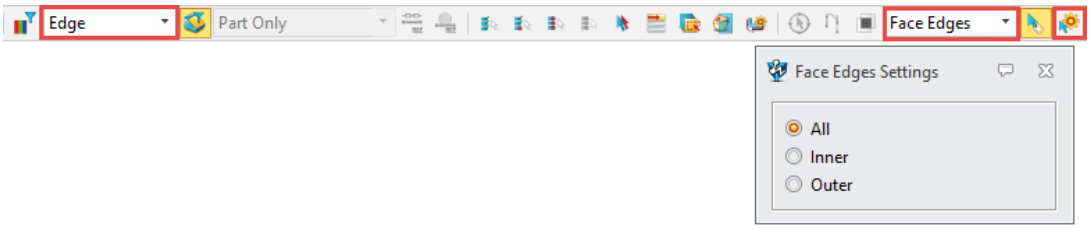
Chamfer		
Shell		 

1.9.2.1.2 Vertex Edges

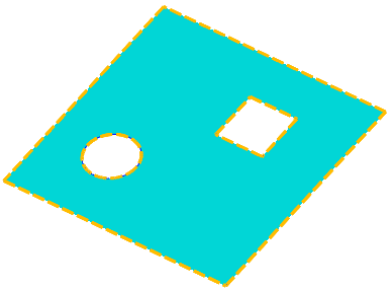
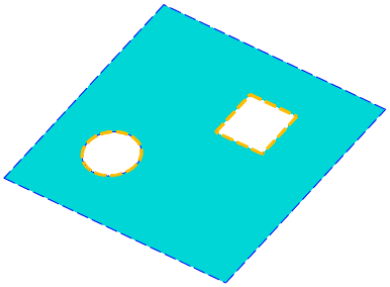
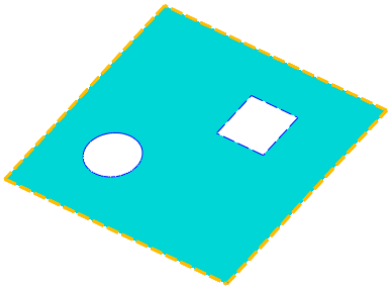
The "**Vertex Edges**" pick rule is to automatically select edges whose common vertex is the point picking up. In following figure, the system will automatically pick up all edges of the vertex by picking the center vertex of the five-star.

1.9.2.1.3 Face Edges

"**Face Edges**" is to pick up the edges on the selected face. The system will automatically pick edges on face based on the rule. The "Face Edges" rule settings is to set "Edge" in filter list and pick "Face Edges", and then click "Rule Pick Setting" to see "Face Edges Settings" panel where user can select three options: **All, Inner,** and **Outer**.

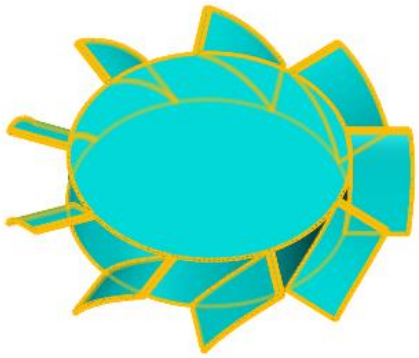


"Face Edges" Pick Effect Display

Rule	Pick Effect
All	
Inner	
Outer	

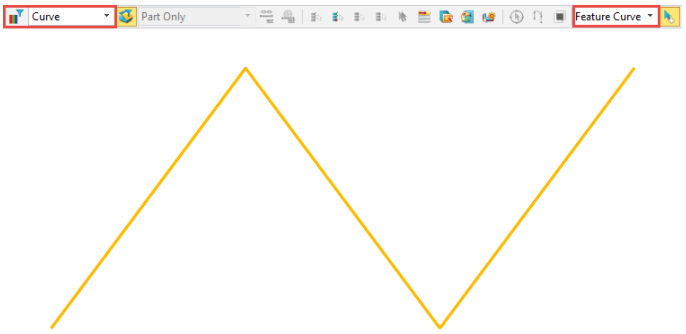
1.9.2.1.4 Shape Edges

Picking an edge by "**Shape Edges**", the system will automatically pick all edges on the shape in which the edge is located. The shape edge effect shows as following figure:



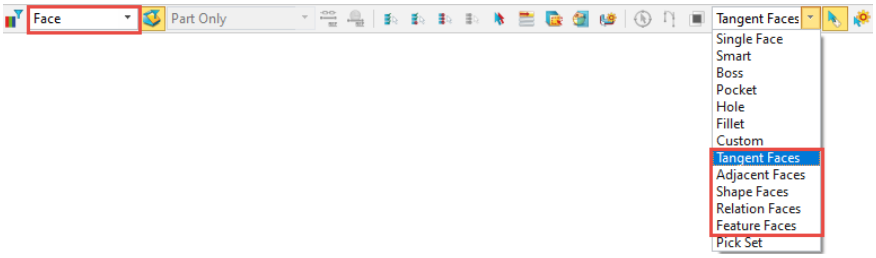
1.9.2.2 Intentional Behavior on Curve

The "**Feature Curve**" is added to the curve pick rule list. All curves with feed curve feature will be picked up based on the feed curve. "Curve" is selected in "Filter List" and "Feature Curve" is selected in "Pick Rule List". The system will automatically pick other curves with the same feature of the curve picked up from the polyline feature. The effect displays as below:



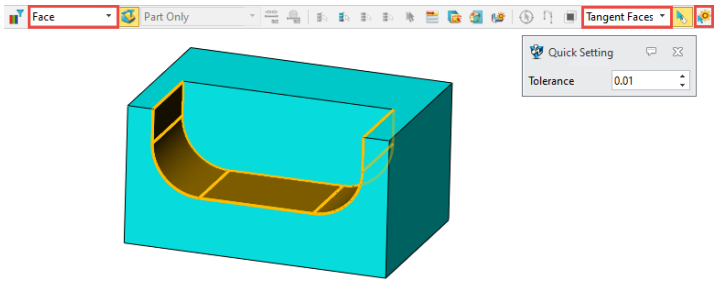
1.9.2.3 Face Intentional Selection Behaviors

To increase the face selection efficiency, we added five face's intentional behaviors in ZW3D 2023 including **"Tangent Faces" "Adjacent Faces" "Shape Faces" "Relation Faces"** and **"Feature Faces"**. User can switch Face to the filter list and the pick rule list will display the five options.



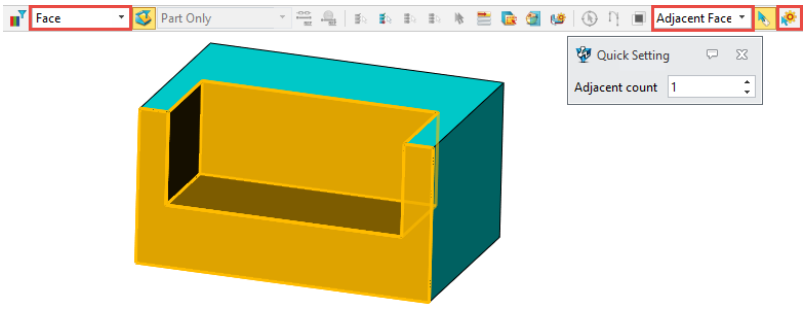
1.9.2.3.1 Tangent Faces

When "**Tangent Faces**" are selected as pick rule, user can execute the selection behavior by left-clicking the mouse on a face. The system will automatically find all tangent faces of the face. User can set the tangent tolerance (0.01 by default) on Quick Setting panel under the pick rule.



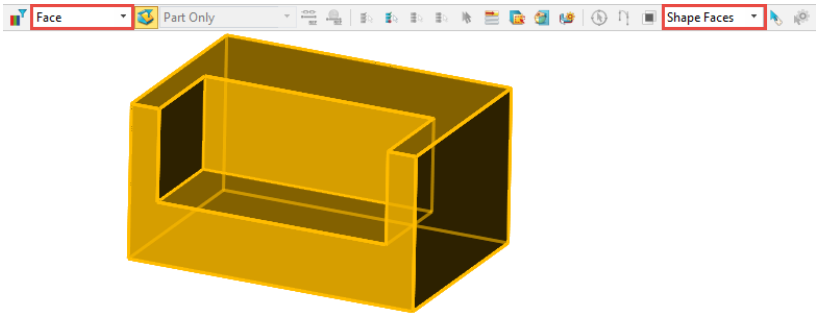
1.9.2.3.2 Adjacent Faces

When "**Adjacent Faces**" are selected as pick rule, user can execute the selection behavior by left-clicking the mouse on a face. The system will automatically find all adjacent faces to the face. User can set the adjacent count (1 by default) on Quick Setting panel under the pick rule.



1.9.2.3.3 Shape Faces

When "**Shape Faces**" are selected as pick rule, user can execute the selection behavior by left-clicking the mouse on a face. The system will automatically find all faces on the shapes where the face is located.



1.9.2.3.4 Relation Faces

When "**Relation Faces**" are selected as pick rule, you can execute the selection behavior by left-clicking the mouse on a face. The system will automatically find the corresponding face based on certain relation. ZW3D 2023 supports seven relations, they are **Coaxial, Tangent**, **Coplanar**, **Coplanar axes**, **Equal radius**, **Symmetric**, and **Offset**, respectively.

- **Coaxial**

  Find faces that are coaxial with the feed face which could be cylindrical, conical, spherical and torus.

- **Tangent**

  Find faces that are tangent with the feed face.

- **Coplanar**

  Find faces that are coplanar with the feed face.

- **Coplanar axes**

  Find faces that are coplanar axes with the feed face which could be cylindrical and conical.

- **Equal radius**

  Find faces that are equal radius to the feed face which could be cylindrical and spherical.

- **Symmetric**

  Find faces that are symmetric with the feed face based on the default coordinate plane by the system. The feed face could be cylindrical and conical.

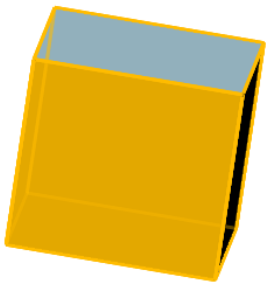
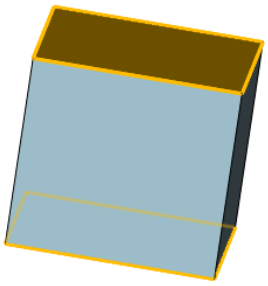
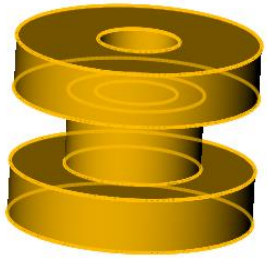
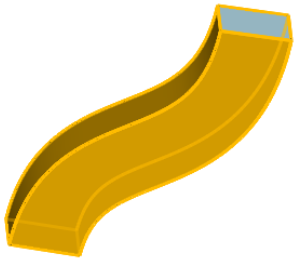
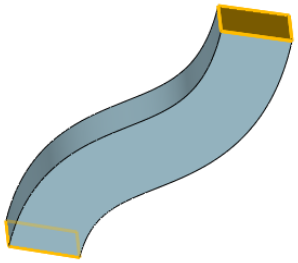
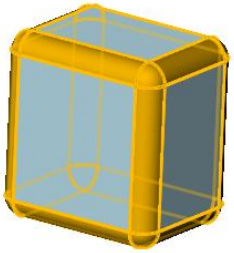
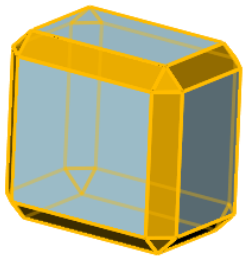
- **Offset**

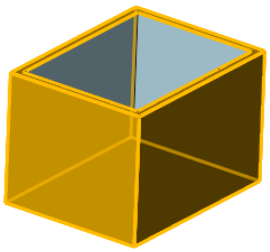
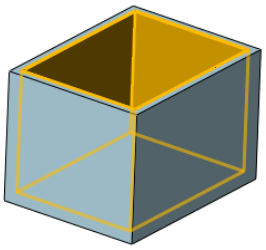
  Find faces that have partial area overlapping and opposite vector with the feed face. There is only one offset face on the feed face.

1.9.2.3.5 Feature Faces

When "**Feature Faces**" are selected as pick rule, user can execute the selection behavior by left-clicking the mouse on a face. The system will find the corresponding faces based on the feature.

"Feature Faces" Pick Effect Display

Feature	Feature Faces Pick Effect	
Extrude		
Revolve		
Sweep		
Fillet		
Chamfer		

Shell		

1.9.2.4 Shape Intentional Behaviors

1.9.2.4.1 Feature Shapes

When "**Shape**" is selected in filter list and "**Feature Shapes**" is switched as the pick rule, left-clicking the feed shape, the system will pick the feature based on the shape and other shapes generated by the feature.

1.9.2.4.2 Distinguish between Solid and Surface in Shape Selection

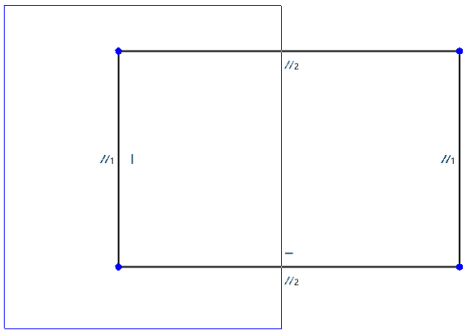
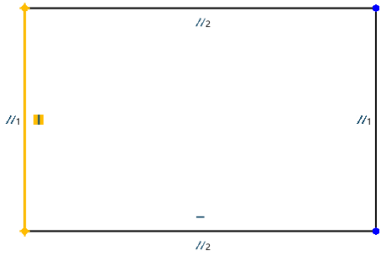
When "**Shape**" is selected in the filter list. "Solid" and "Surface" options are added to the pick rule to better distinguish solid and surface selection.

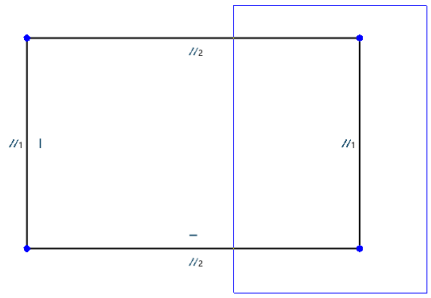
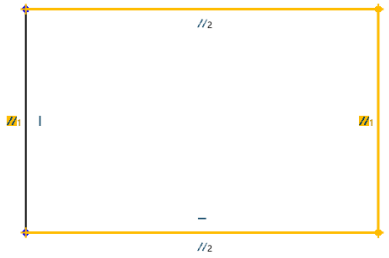
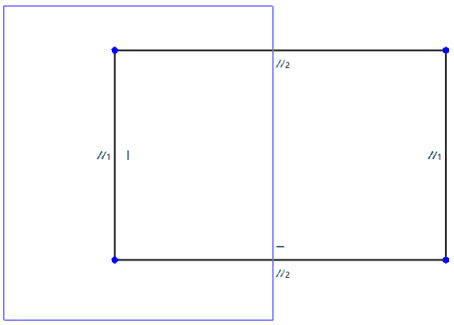
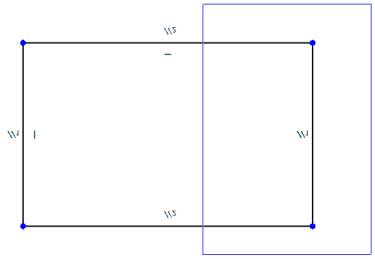
1.9.3 Pick Interaction Behavior Improvement

1.9.3.1 Windows Pick Behavior Improvement

The windows pick supports the differences from left to right and from right to left as follows:

- When the current pick mode is "**In/Cross**", then from left to right is "**In**" and from right to left is "**Cross**".
- The current pick mode "**Out/Cross**", then from left to right is "**Out**" and from right to left is "**In**".

Pick Mode	Windows Pick Direction	Pick Result
In/Cross	**From left to right** 	

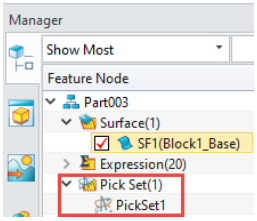
In/Cross	From right to left 	
Out/Cross	From left to right 	
Out/Cross	From right to left 	

1.9.3.2 Pick List Supports Quick Switch

Press Alt (cannot be released) + right clicking (continuously): traverse and pre-highlight (pre-selection) the legal objects under the current mouse location and left click the mouse when pre-highlight some entity to confirm the current selection.

1.9.4 New Pick Set

We added “Pick Set” to ZW3D 2023, which supports curves, edges, parting edges, faces, shapes, points, datum planes, datum axes, datum frame, and the existing pick sets. After a pick set is created, it will display in the manager. At this point, you can select the existing pick sets when “Pick Set” is switched to “Pick Rule List”.



➔ **Where it is**

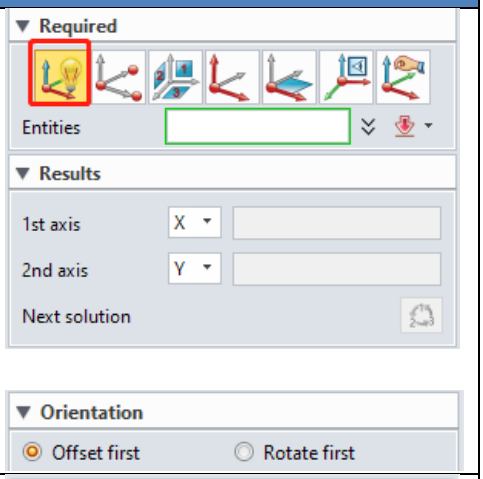
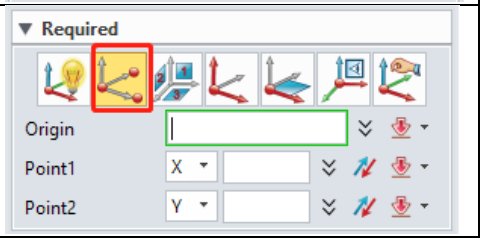
Part Environment >> Tools >> Insert >> Pick Set

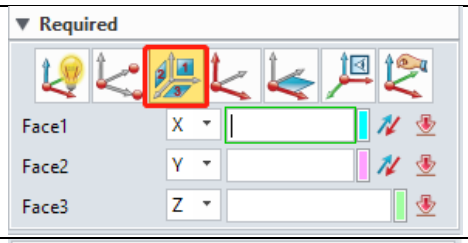
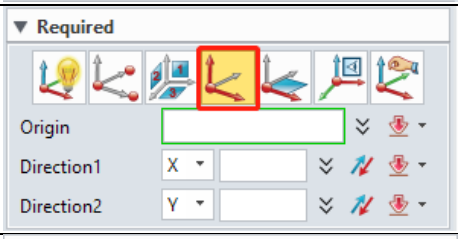
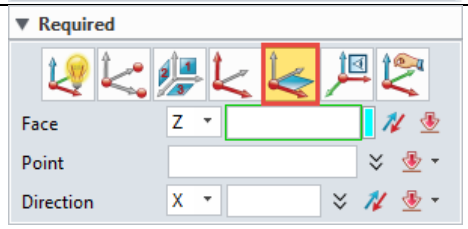
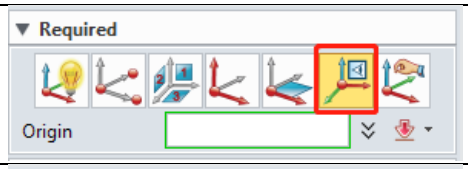
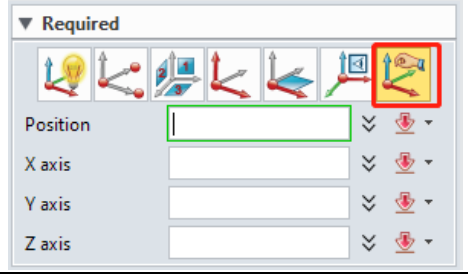
1.10 ★Datum Improvement

1.10.1 "Datum CSYS"

We optimized "**Datum CSYS**" command in ZW3D 2023. The new datum CSYS supports "**Geometry**" "**3 points**" "**3 faces**" "**Point and 2 directions**" "**Plane, point and direction**" "**View plane**" and "**Dynamic**" methods to create a CSYS.

The improvements show as below:

Method	Main Improvement	Icon
Geometry	With the improvement, user can select up to three reference geometry objects at once. It will automatically create a Datum CSYS and support switching axes. The reference geometry includes points, lines, edges, faces and CSYS (User can set offset or rotation first when CSYS is referenced).	
3 points	(1) Icon updated. (2) Support switching axes and reverse their directions (Specify X and Y as default axes)	

3 faces	The new method allows users to select three reference planes to create a Datum CSYS, switch axes and reverse the axis direction.	
Point and 2 directions	The new method allows users to select an origin and 2 directions to create a Datum CSYS, switch axes and reverse the axis direction.	
Plane, point and direction	The new method supports creating a Datum CSYS where the selected plane is defined as Z-axis plane (XY plane), the projections of the point and the direction on the plane are defined as the origin and X-axis and supports switching axes and reversing the axis direction.	
View plane	Only icon is updated.	
Dynamic	Only icon is updated	

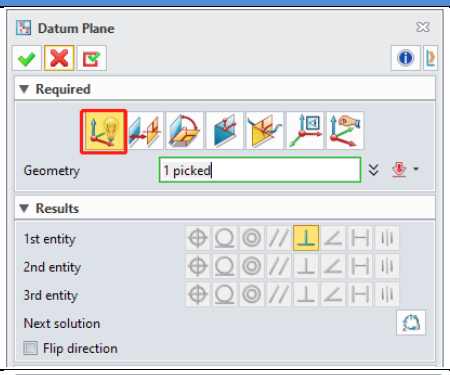
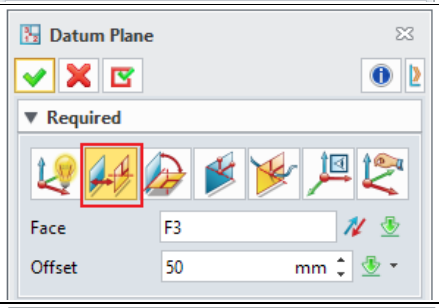
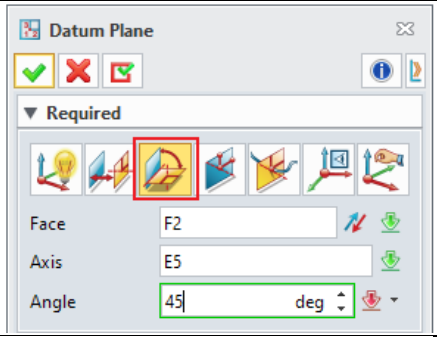
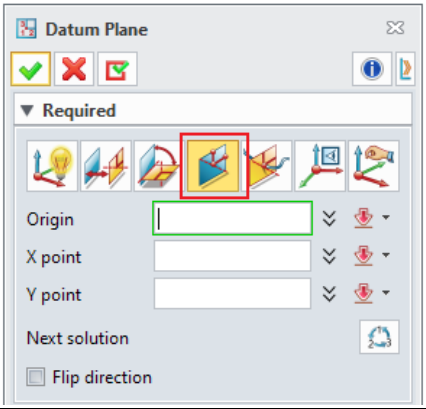
➔ **Where it is**

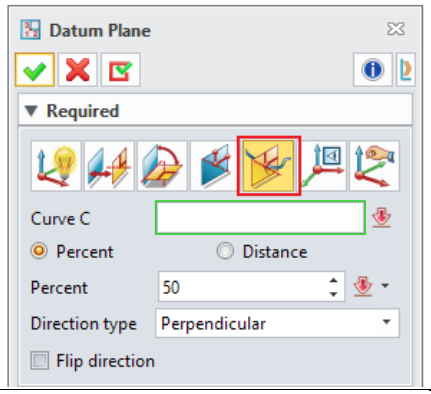
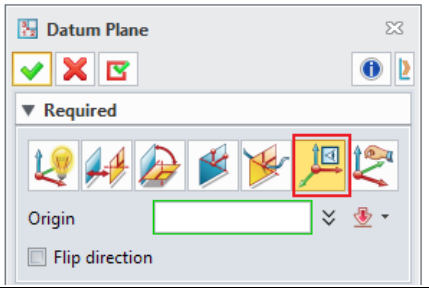
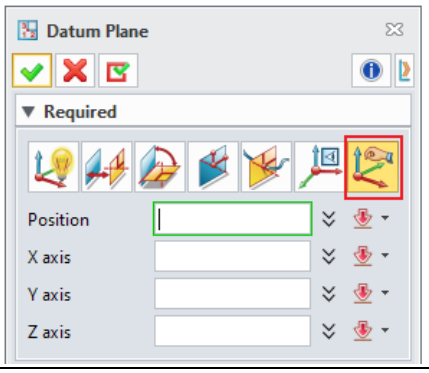
Part Environment >> Shape/Free Form/Wireframe/Weldments >> Datum >> Datum CSYS

1.10.2 Datum Plane Improvement

We reconstructed the “Datum Plane” command in ZW3D 2023, which supports “**Geometry**” “**Offset plane**” “**Angle to face**” “**3 point plane**” “**On curve**” “**View plane**” and “**Dynamic**” methods to create a datum plane.

The improvements show as below:

Method	Main Improvement	Icon
Geometry	The method supports user selecting at most three reference geometric objects at one time. The system will automatically analyze the "Constraint" relationship between geometry and the creating datum face and highlight the constraint icon. The supporting geometries include points, lines, edges, axes and faces.	
Offset face	The new method supports user creating a datum plane by specifying a plane or datum plane to offset and supports to reverse the normal of datum plane.	
Angle to face	The new method supports user specifying reference plane, rotation axis and angle to create a datum plane which is at an angle to the reference plane and supports to reverse the normal of datum plane.	
3 point plane	The upgraded method supports user to specify at most three points to create a datum plane. Next solution is available for single point and the default normal of the created datum plane can be along with the three axes. Support flipping the normal of the datum plane.	

On curve	The new method supports user specifying reference curve/edge to create datum plane and supports to control the curve position via Percent and Distance. The direction type refers to the relation between datum plane and the reference curve, which includes Perpendicular, Tangent, Perpendicular to curve and Tangent to curve. Support flipping the normal of the datum plane.	
View plane	The icon is updated and flipping direction is supported.	
Dynamic	Only the icon is updated.	

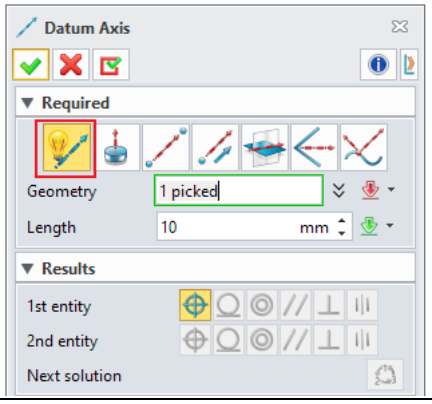
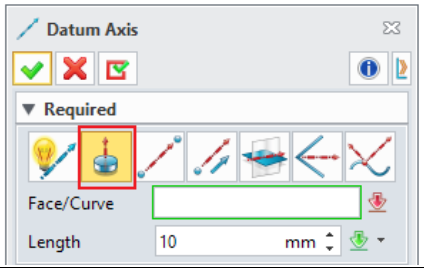
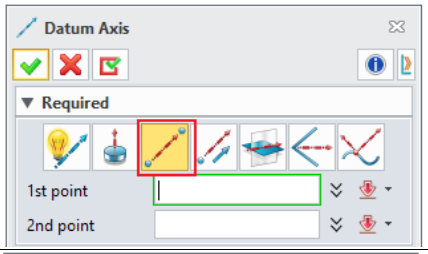
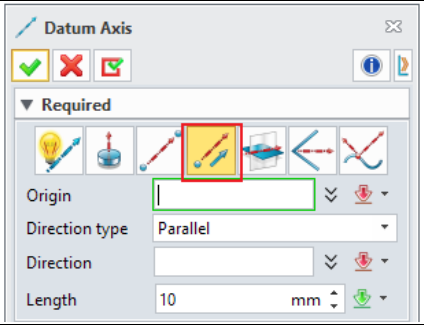
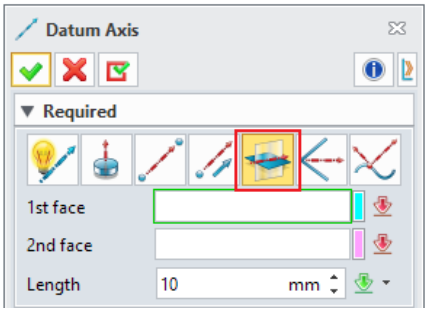
➔ **Where it is**

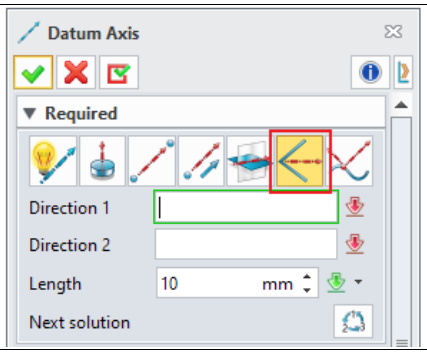
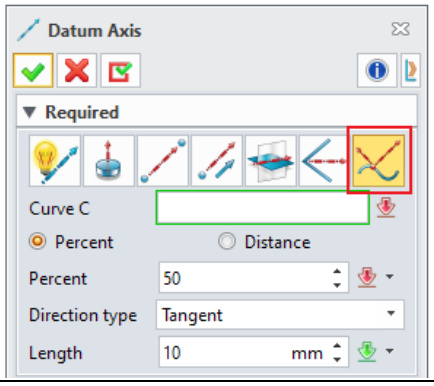
Part Environment >> Shape/Free Form/Wireframe/Weldments >> Datum >> Datum Plane

1.10.3 Datum Axis Improvement

We reconstructed "**Datum Axis**" command in ZW3D 2023, which supports "**Geometry" "Center axis" "2 points" "Point and direction" "Intersected face" "Angular bisector"** and **"On curve"** creating a datum axis.

The improvements show below:

Method	Main Improvement	Icon
Geometry	The upgraded method supports user picking at most 2 reference geometry objects and the system will automatically analyze the "Constraint" relationship between the picked geometry and the creating datum axis and highlight the corresponding constraint icon. The reference geometry includes points, lines, edges, axes, and surfaces.	
Center axis	The upgraded method can create a datum axis through plane curve axis.	
2 points	Only the icon is updated.	
Point and direction	The upgraded method adds the direction types including Parallel and Perpendicular. User can specify point and direction and create a datum axis according to the specified direction type.	
Intersected faces	Keep unchanged.	

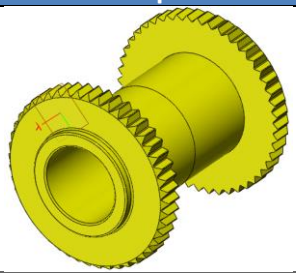
Angular bisector	The new method supports user specifying two intersected lines. The intersected point is as the start of datum axis while the angular bisector or supplementary bisector is as the direction of the datum axis.	
On curve	The new method supports user creating a datum axis by specifying reference curve/edge and controlling the curve position by Percent and Distance. The direction type refers to the relationship between datum axis and reference curve including Tangent, Perpendicular, Parallel to curve and Perpendicular to curve.	

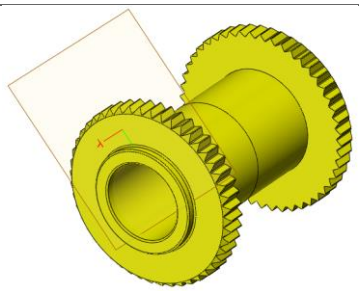
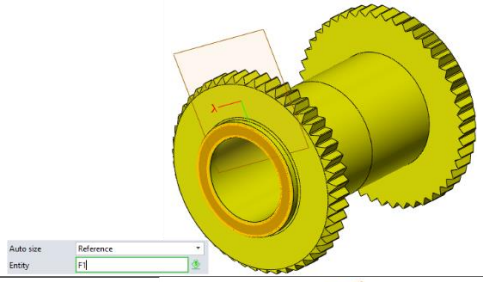
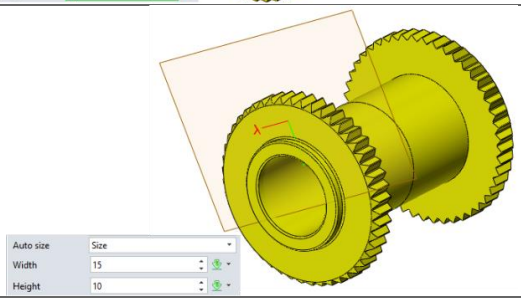
➔ **Where it is**

Part Environment >> Shape/Free Form/Wireframe/Weldments >> Datum >> Datum Axis

1.10.4 New Auto Size in Datum Plane

We added the “Auto size” option to better assist users in adjusting the datum plane size when they create a new datum plane. The drop-drown list includes four options Default, Auto size, Reference, and Size.

Option	Definition	Sample
Default	Use default size to display datum plane	

Auto size	Automatically use the plane that datum referred as the size of datum plane	
Reference	User can define random plane as the reference size for datum plane	
Size	User can customize the width and height for the datum plane	

➔ **Where it is**

Shape >> Datum >> Datum Plane >> Datum Attributes

1.11 UI Efficiency Optimization

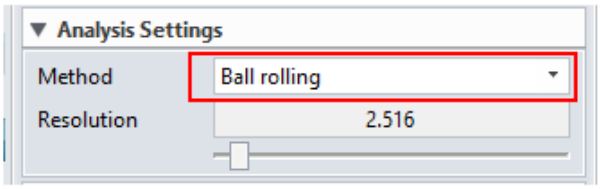
The UI enhancement of current version can provide better user experience as below:

1. Increase the file opening speed.
2. Increase the speed of selecting and invert-selecting of the assembly tree node.
3. Regenerate history much faster.
4. Improve the blank/unblank speed of selected entities.
5. Reduce the stuck while initially switching to assembly manger UI, particularly for more sophisticated assembly tree files.

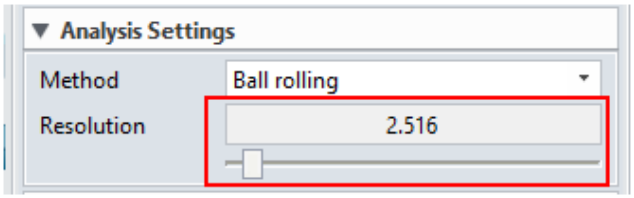
1.12 ★Thickness Analysis Improvement

We added a new method Ball rolling (Inner Sphere Searching (ISS) Method) to increase the efficiency and resolution of model thickness analysis and improve the display effect after analyzing.

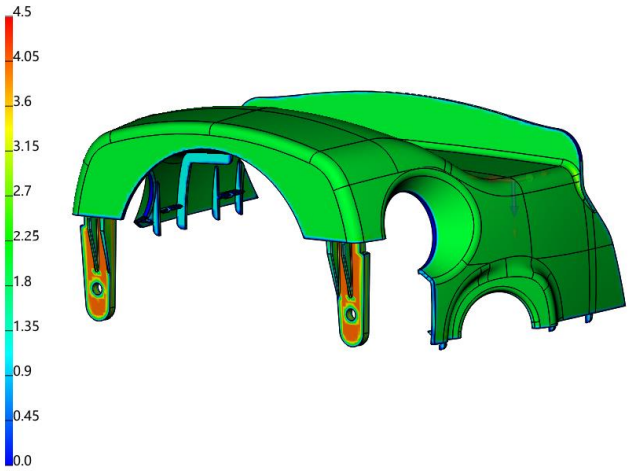
1） New thickness analysis method: Ball rolling



2） The analysis resolution from the selections of “Coarse” “Standard” “Fine” and “Very Fine” are changed to the display of value + rolling bar. The system will automatically match a suitable sampling value according to the model size and user can drag the rolling bar to change the value according to the request. The smaller the value, the finer the analysis, but it takes longer time.



3） We optimized the display color after analysis, from dark blue to dark red corresponding to the minimum and maximum thickness and enhanced the recognition for user to better observe the thickness change in each area.

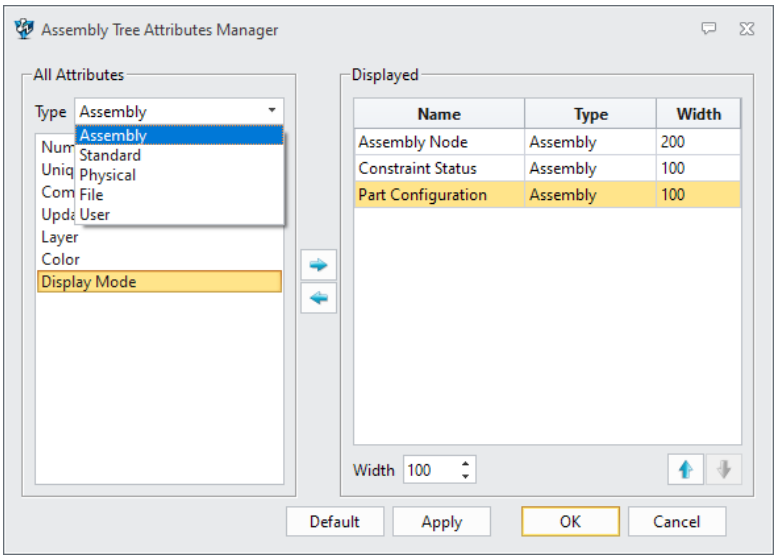


➔ Where it is

Part Environment >> Inquire >> Inspect Entities >> Thickness Analysis

1.13 Assembly Tree Manager Improvement

The improved assembly tree supports managing more attributes, and the attributes list supports displaying more standard and user attributes.



- Increase attribute types and facilitate attribute managing.
- Introduce part attributes to the assembly tree attributes manager and support more attribute types as well as user attributes.
- Support setting the displayed width.
- Support adjusting the display order of attribute column.

➔ Where it is

Assembly Environment >> Assembly Context Menu>> Assembly Tree Attributes Manager

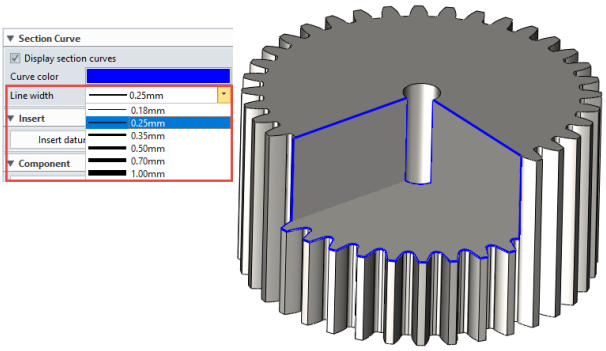
Part Environment >> Feature Node Context Menu>> History Tree Attributes Manager

1.14 Display Improvement

1.14.1 Section Improvement

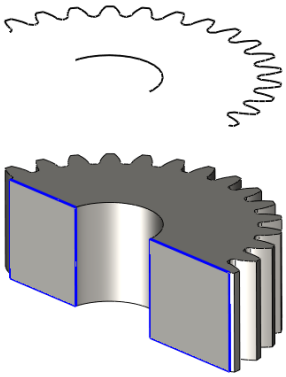
1.14.1.1 New Line Width to Control Section Curve

We added “Line width” to the Section command in the part environment to get a clearer view of the section profile.



1.14.1.2 New Clip Wire

We added “Clip wireframe” to Section to better observe section target in the complex wireframe design. If checked, user can section wireframe and sketch.

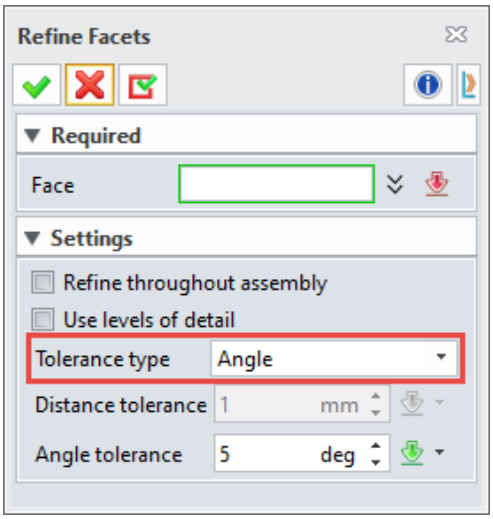


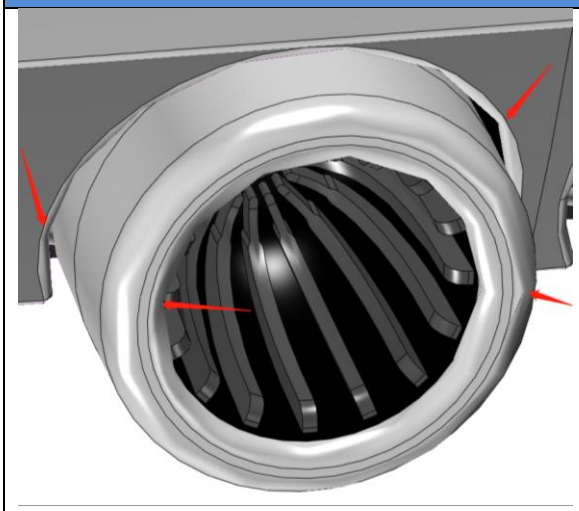
➔ Where it is

Part Environment >> Inquire >> Inspect Model >> Section

1.14.2 New Angle and Distance Tolerance Type

When the span of modeling model size and the minimum feature size reaches to 1,000 times, the display accuracy of the minimum feature is quite poor, which needs user to refine the small feature by manual picking, but it is quite low. We added “**Distance**” and “**Angle**” as tolerance type to adjust the whole refined facets. The system will automatically refine the display facet size of new generated face to raise the display accuracy.



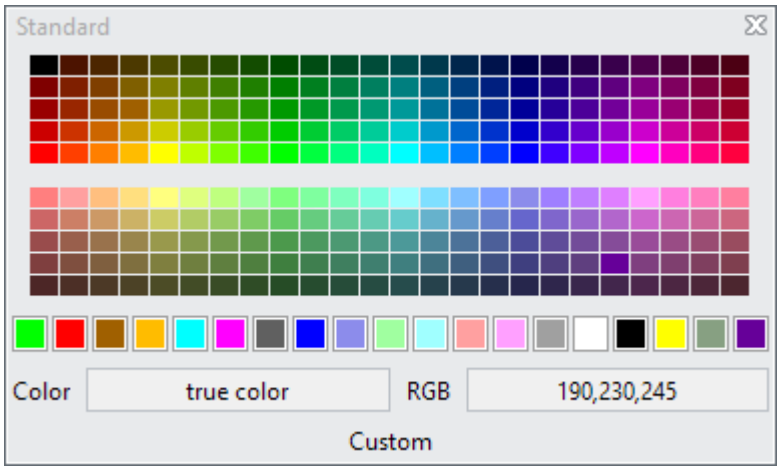
Pixel	Angle/Distance
	

➔ **Where it is**

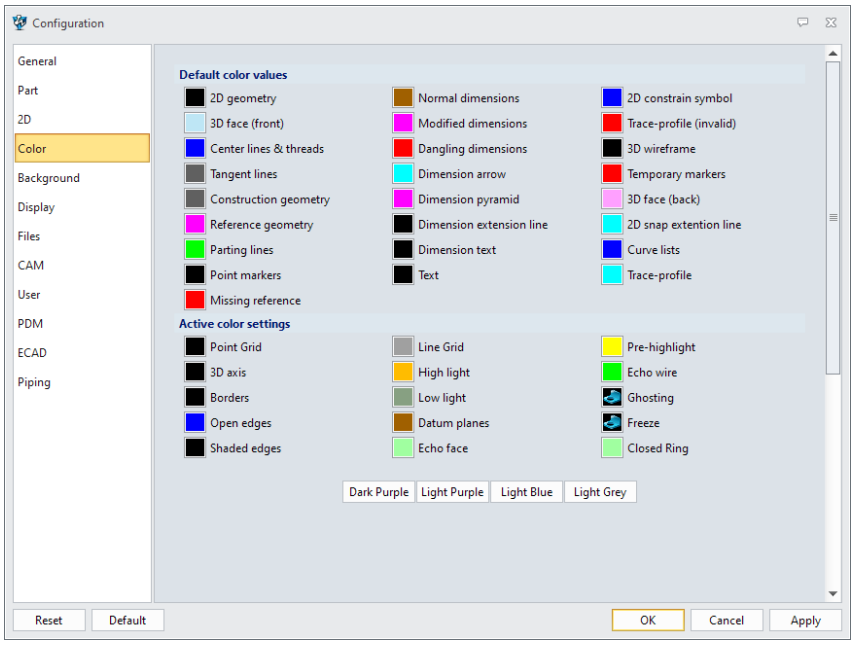
Menu Bar >> Visualize>> View >> Refine Facets

1.14.3 ★Color Improvement

We added more colors to the color selection panel, from 18 standard colors to 240 index colors.



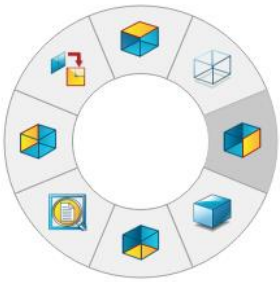
Wherever in “Attribute Filter”, “Configuration -> Color” or the attribute setting of point/line/surface, user can get more options in any color selection panel.

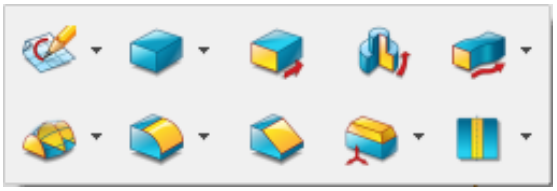


1.15 Usability Improvement

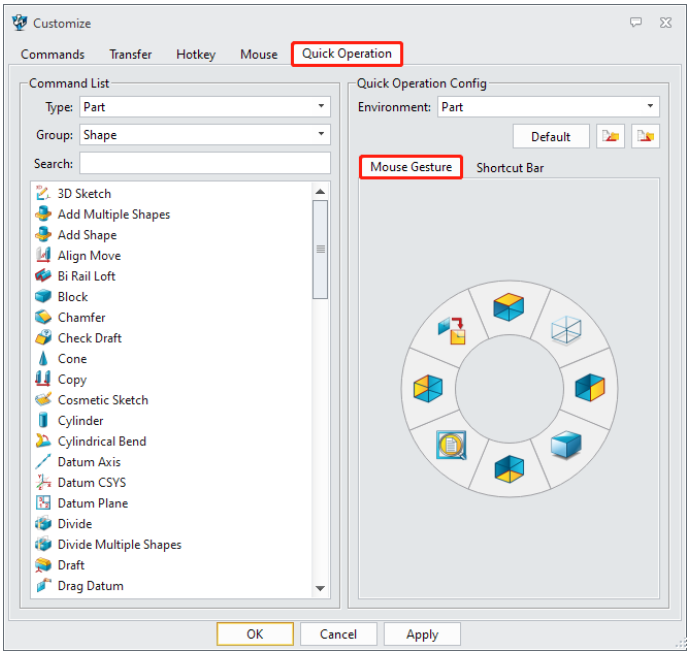
1.15.1 New Mouse Gesture and Shortcut Bar

We added the mouse gesture and quick operation toolbar for user to quickly utilize common commands. User can operate by the shortcut keys (customized) and quickly open the corresponding function by the shortcut toolbar button.

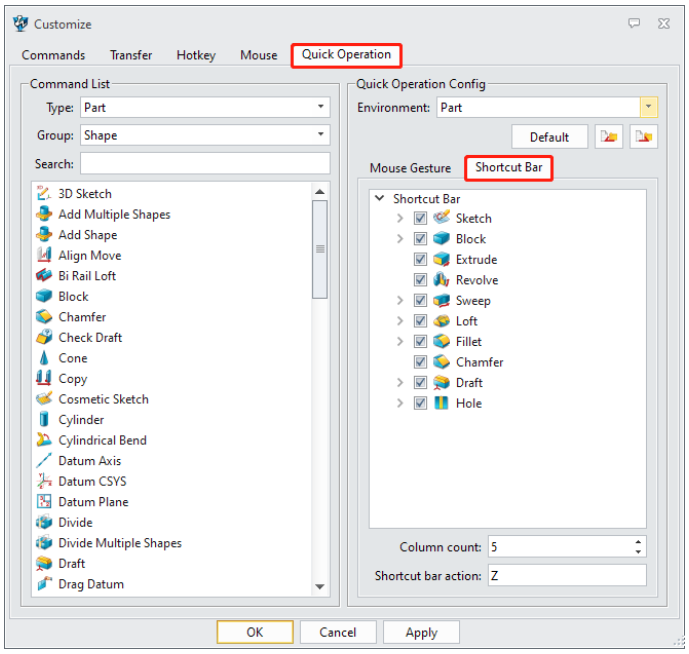




In Menu >> Tools>> Customize, open the Customize panel as below figure. User can drag the command icons by left-clicking the mouse to add or move commands from “Mouse Gesture”.



In Menu >> Tools >> Customize, open the Customize panel as the below figure. User can drag the command icons by left-clicking the mouse to “Shortcut Bar” and can adjust the shortcut bar.



➔ **Where it is**

Mouse Gesture: Shift + Right Mouse Click

Shortcut Bar: Press shortcut key Z

1.15.2 New Activated Method Zoom All

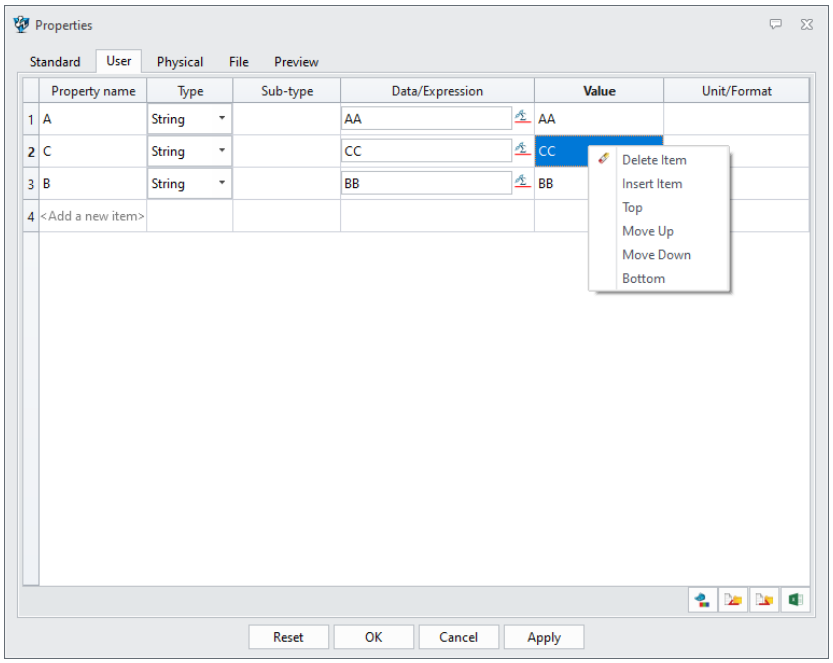
In the assembly, part, sketch and drawing sheet environments, we added a method of double-clicking left of the mouse button in the blank drawing area to activate Zoom All.

1.16 Attributes Improvement

1.16.1 Support Adjusting User Properties Order

User variable can be managed according to certain order. The order of new item can be set according to options including Move Up, Move Down, Top and Bottom. User can add a new item above the specified item by “Insert Item”.

- Select the variable row and right click to pick “Move Up” “Move Down” “Top” and “Bottom” to adjust the order.
- Select the variable row and right click to pick “Insert Item” to add a new item above the selected row.

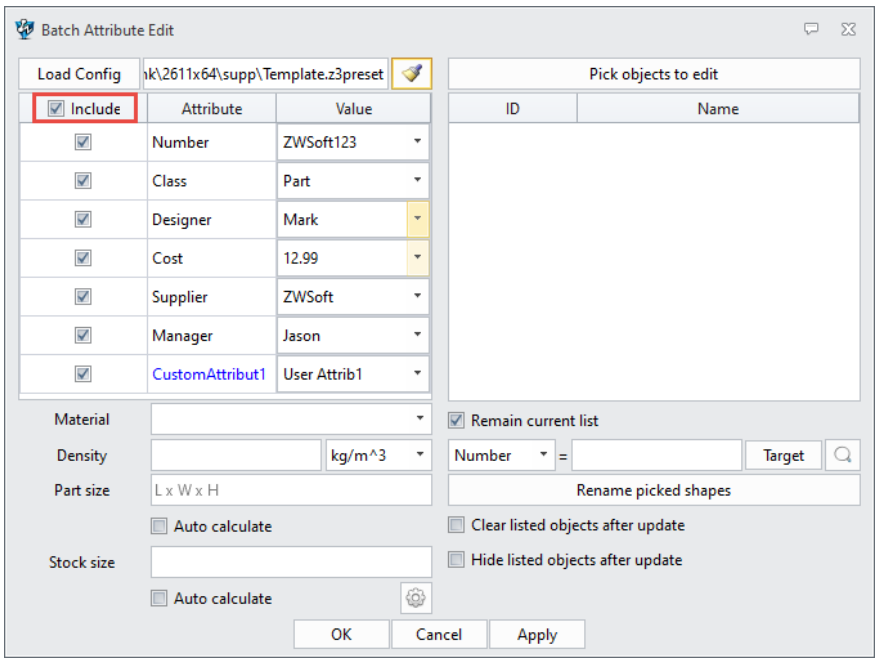


➔ **Where it is**

Part Environment >> Attributes >> Properties >> User

1.16.2 Batch Attribute Edit Improvement

We added Check All/Uncheck All checkbox in Column “Include” in “Batch Attribute Edit” to aid user quickly pick/unpick all attributes. User only needs to check the box for fast check/uncheck all attribute items.

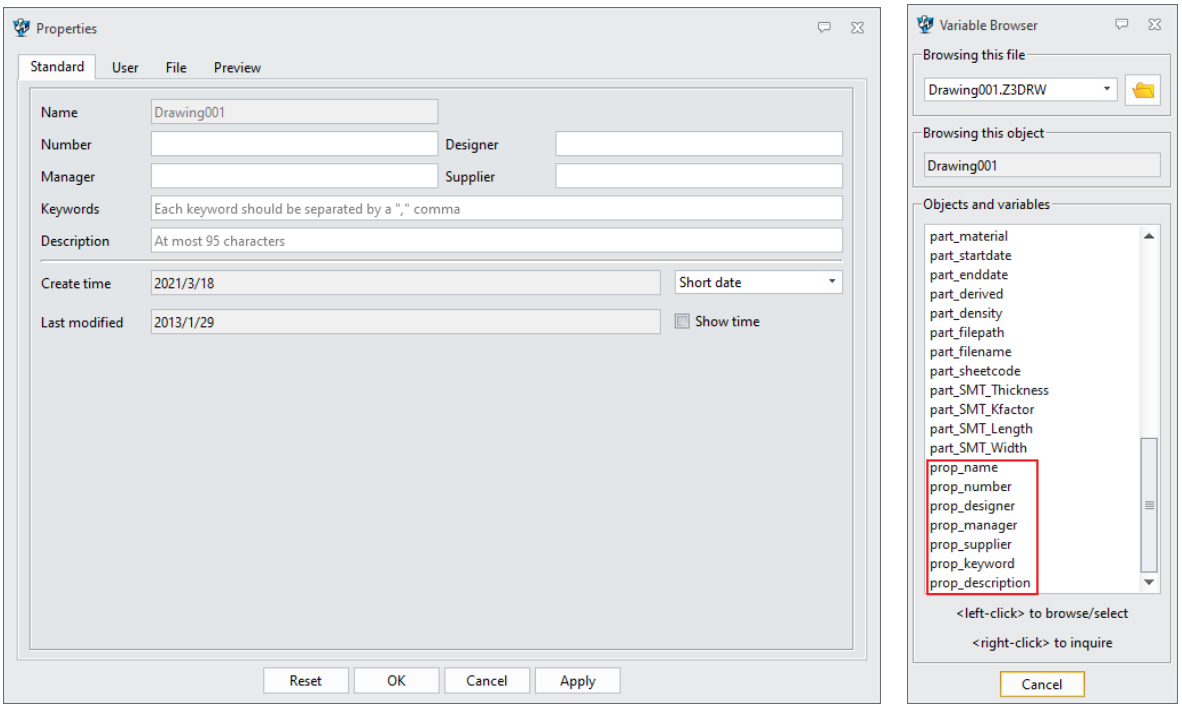


➔ **Where it is**

Part Environment >> Tools >> Attributes >> Batch Attribute Edit

1.16.3 Attributes Setting Improvement

We added attributes setting to CAM, drawing sheet and 2D sketch environment, which supports standard and user properties. Support the attributes referring in Variable Browser.



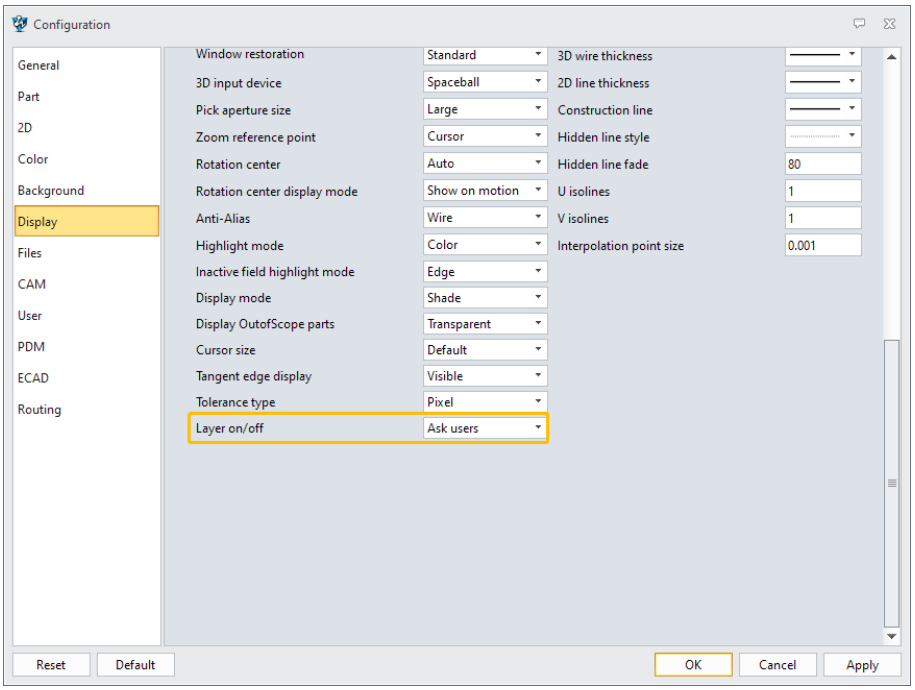
➔ **Where it is**

Menu Bar >> Attributes >> Properties

1.17Configuration Improvement

1.17.1New Layer On/Off Option

The original item “Turn on layer when toggle visibility” in configuration was removed in ZW3D 2023 and changed to “Layer on/off” drop-down list, achieving various settings for layer visibility.

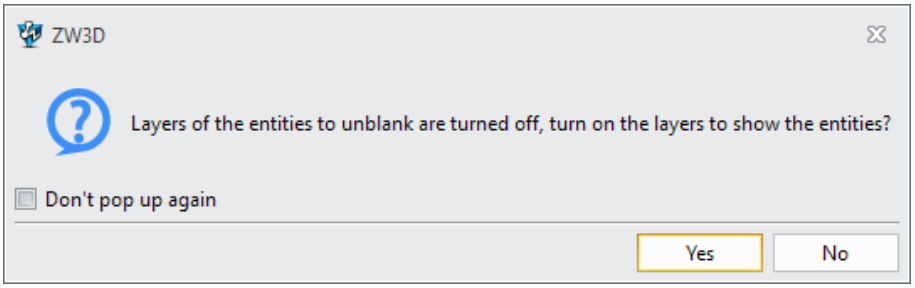


The option is to display the object that the layer is closed and the object itself is hidden. Three options in the drop-down list available, including:

Off for visibility: When displaying objects, the layer will not be opened automatically. Thus, the objects will not be displayed.

On for visibility: When displaying objects, the layer will automatically be opened. Thus, the objects will be displayed.

Ask users: When displaying objects, popup the following dialogue and aske user whether to open the layer. If Yes, open the layer and display objects; if NO, withdraw the current operation and not display objects.

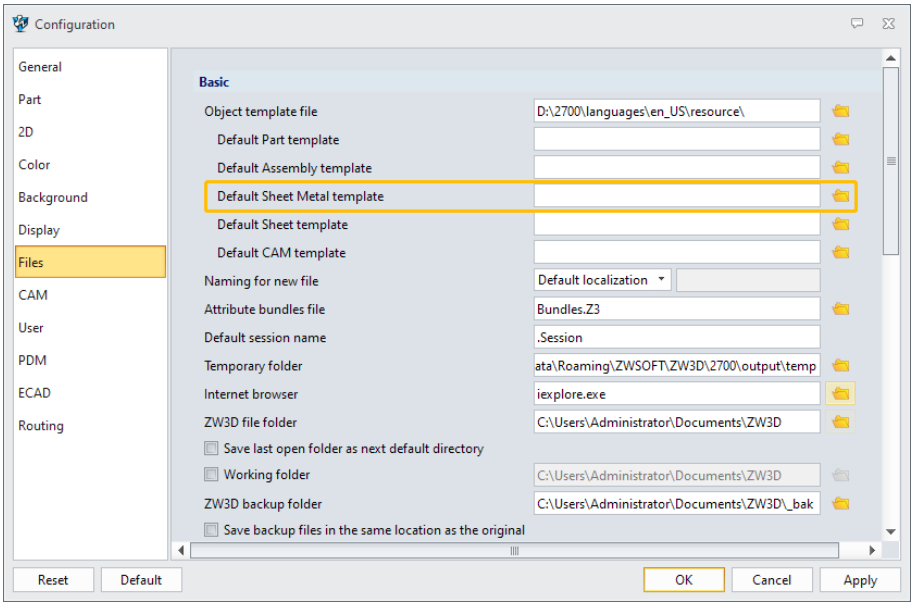


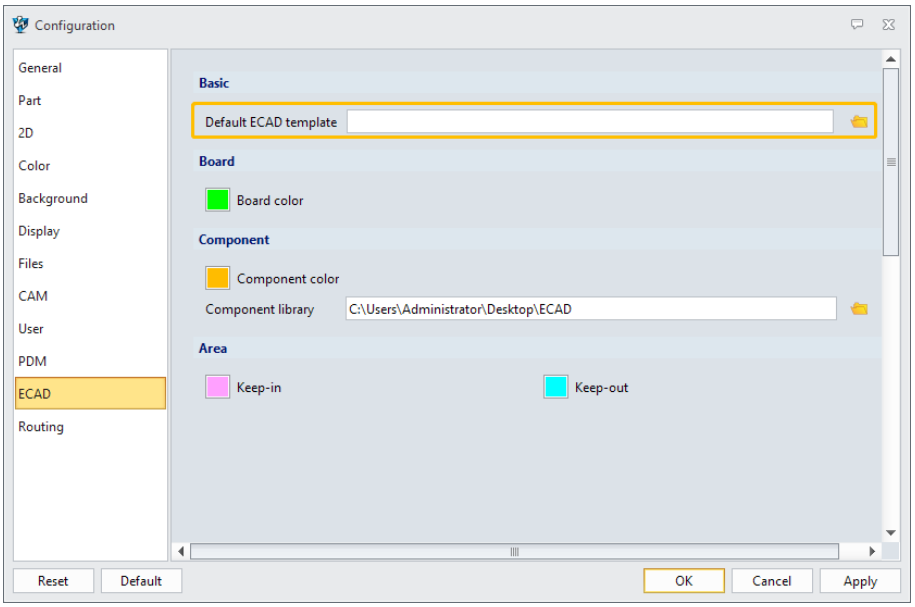
➔ Where it is

Configuration >> Display >> Options & defaults >> Layer on/off

1.17.2 New Default Template

We added sheet metal template and ECAD default template in configuration to help user better define sheet metal templates.





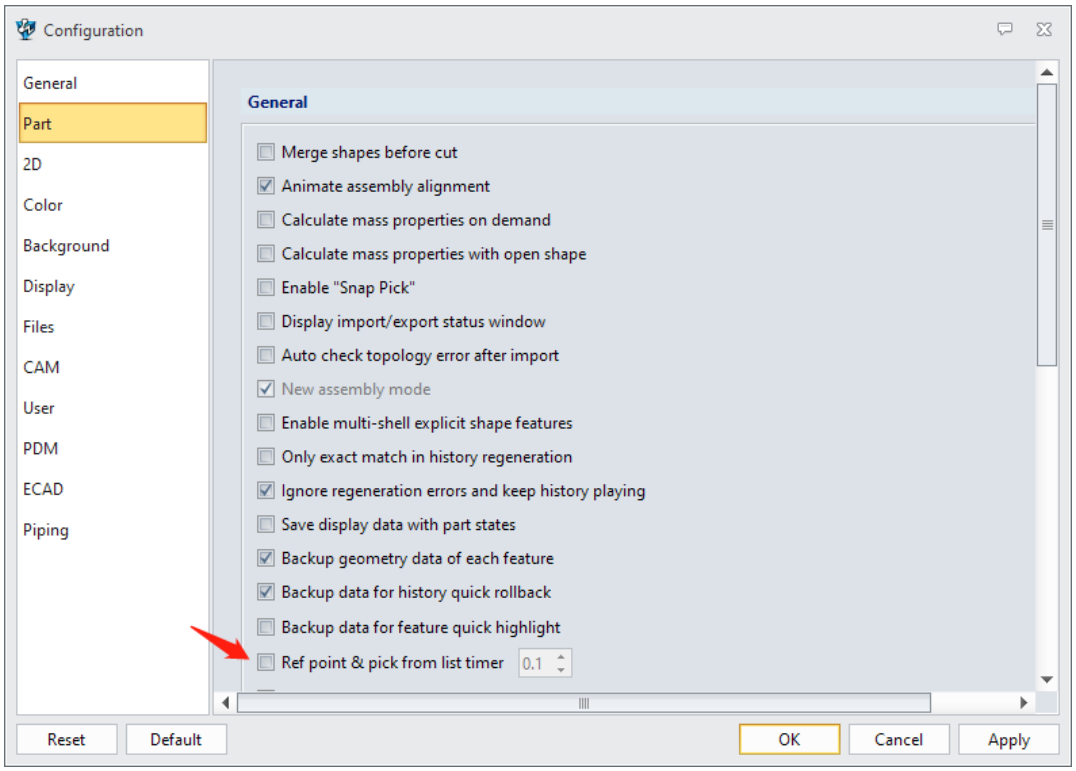
➔ Where it is

Configuration >> Files >> Basics >> Default Sheet Metal template

Configuration >> ECAD >> Basics >> Default ECAD template

1.17.3 Ref Point & Pick from List Timer Improvement

We added a checkbox to “Ref point & pick from list timer” to enable the function. If unchecked, disable the function. If checked, user can set timer time from .0.1 to 4s with an accurate to one decimal place. The function is enabled by default.

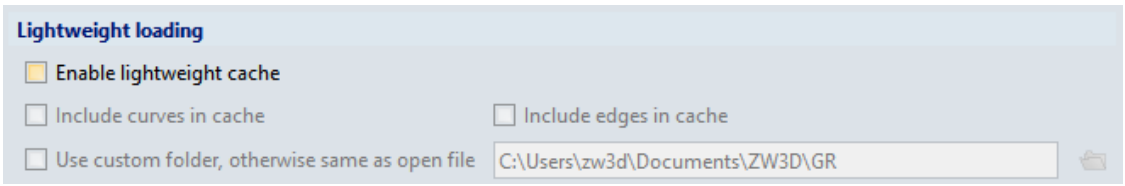


➔ Where it is

Configuration >> Part >> General >> Ref point & pick from list timer

1.17.4 Opening Efficiency of Big File Optimization

The lightweight loading allows user to cache file through Z3DS, which increase the opening efficiency of big files.



If “Enable lightweight cache” is checked, it will:

1） Generate cache file if there is no cache file when open file.

2） Load cache file if there is cache file when open file.

3） Automatically generate cache file when save file.

Besides, user can decide whether to include curves or edges in cache.

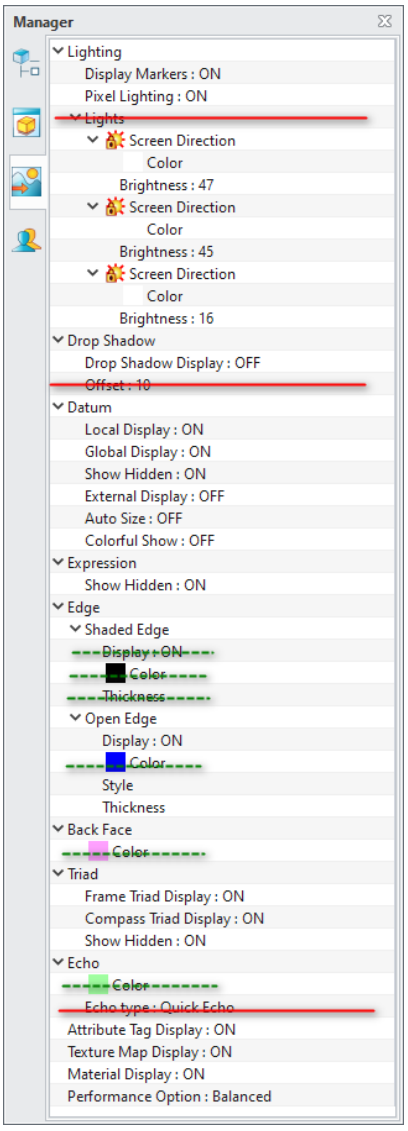
➔ **Where it is**

Configuration >> Files >> General >> Lightweight loading

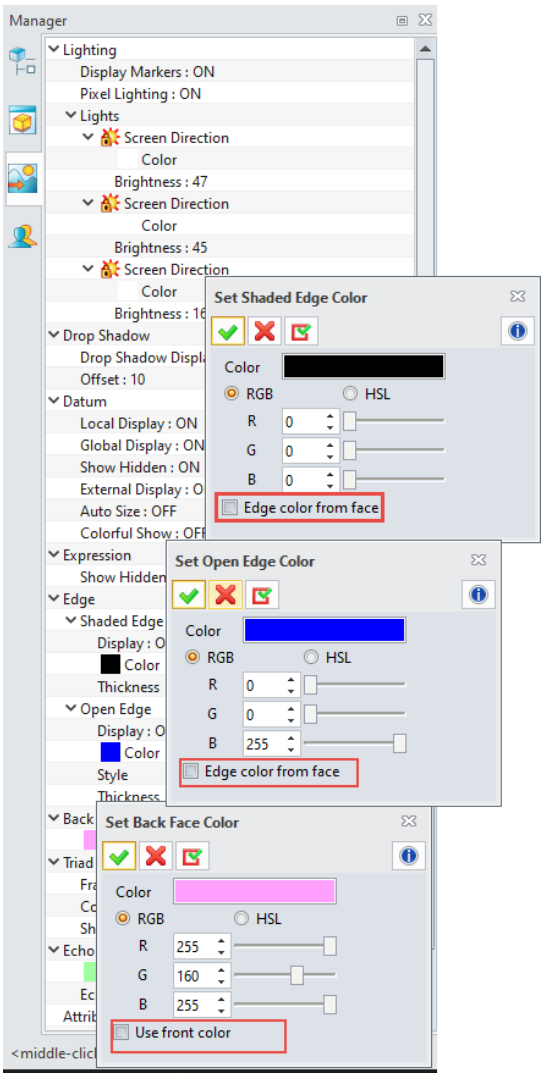
1.17.5 New “Visual Manager” Configuration

We added a function to save configuration files in the visual manager. When user changes the settings in visual manager and closed ZW3D, those changes will be saved in the configuration files of user directory by ZW3D. The changed status will be kept in the visual manager while staring ZW3D next time. The configuration files are saved in the path: User Folder\custom_config_en_US.xml

Notes: There exists two special settings. One is the relevant settings that follow the files. The setting change will not be saved to the configuration files, as the red line marked in the following figure; while the other is the existing settings in global configuration panel, which has been recorded in the configuration file, as the green dotted line marked in the following figure.



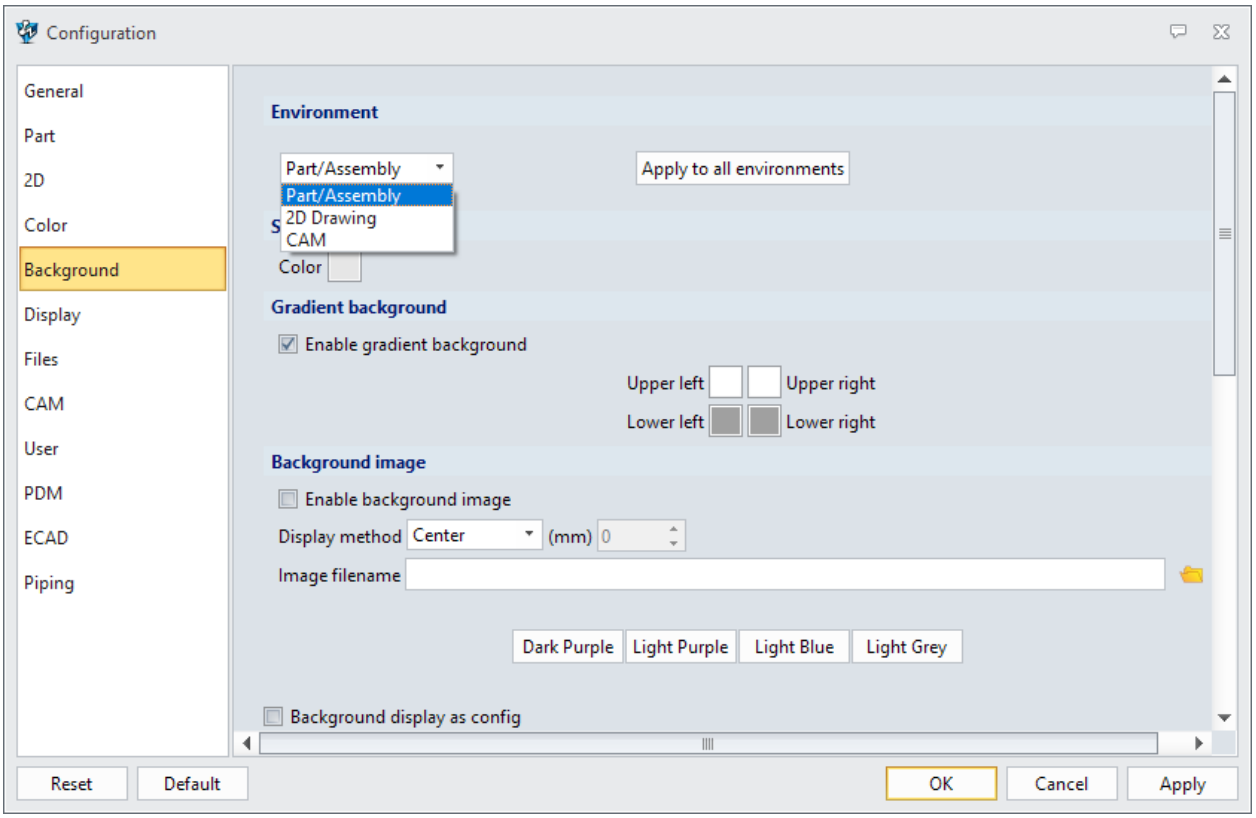
Check the option “Use front color” in Back Face>>Color, all faces of the currently open or newly drawn models keep the same color on both the front and back sides. When restart the system, it will keep the previous status.



1.17.6 Background Improvement

We added "**Environment**" to background for setting the background color in Part/Assembly Environment, Sketch Environment, Drawing Sheet Environment and CAM Environment.

Click "**Apply to all environments**" to apply the current background setting to all environments so that to unify all environment backgrounds.



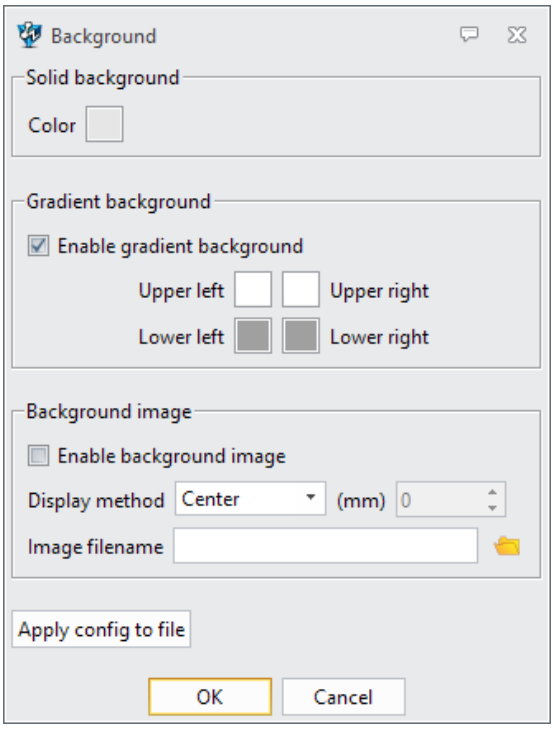
We added the option "**Background display as config**". If it is checked and activated, the corresponding background will display as configuration style; otherwise, it will display as the display style of the current file.

➔ Where it is

Configuration >> Background

1.17.7 New Background Modification Function

We added "**Background**" command to each environment in "Tools" Ribbon. Use this command to set the display style of current file. The setting contents and interface are the same as that in "Configuration". When "**Background display as config**" is unchecked, the display style of current environment is the same as the setting style of this command.



"**Apply config to file**" is to copy the background setting in configuration to the current file background.

➔ Where it is

All Environments >> Tools >> Settings >> Background

1.18 Other

1.18.1 New Function Expression in "Function List"

The celling2(X,Y) returns a function that rounds parameter X up (in the direction of increasing absolute value) to a multiple of the nearest specified cardinality, taking the following arguments:

- X is required and it is a rounded value.
- Y is requited and it is a rounded multiple.

The function has the following characteristics:

- If any parameter is non-numeric type, an error message pops up.
- Regardless of the symbol of the parameter X, the value is rounded up in the direction of increasing absolute value. If X is exactly a multiple of Y, no rounding is performed.

- If both X and Y are negative, the absolute value rounded down away from 0.
- If X is negative and Y is positive, the absolute value rounded up away from 0.

Sample:

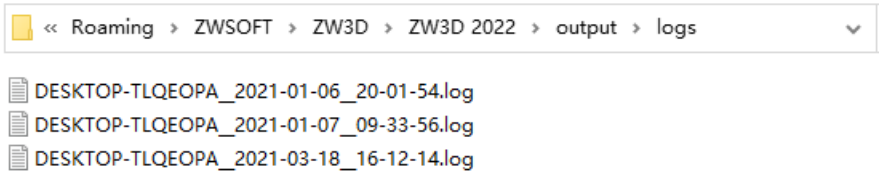
Equation	Description	Result
celling2 (2.5, 1)	Rounds 2.5 up to the nearest multiple of 1	3
celling2 (-2.5, -2)	Rounds -2.5 up to the nearest multiple of -2	-4
celling2 (-2.5, 2)	Rounds -2.5 up to the nearest multiple of 2	-2
celling2 (1.5, 0.1)	Rounds 1.5 up to the nearest multiple of 0.1	1.5
celling2 (0.123, 0.01)	Rounds 0.123 up to the nearest multiple of 0.01	0.13

➔ **Where it is**

Tools >> Insert >> Equation Manager >> Function List

1.18.2 New Logs Saved Number

Logs are an important assisting approach to locate issues. In the previous version, the logs number was only limited to 8, which was quite a small number. But in ZW3D 2023, the logs number has been upgraded to 30.



➔ **Where it is**

User Directory >> output >> logs

1.18.3 ZW3D 2023 Will Give Up Supporting 32-bit Version

We will give up supporting the 32-bit Windows operating system started from V2023 and ZW3D network services suite will simultaneously give up supporting it.

2 Translator

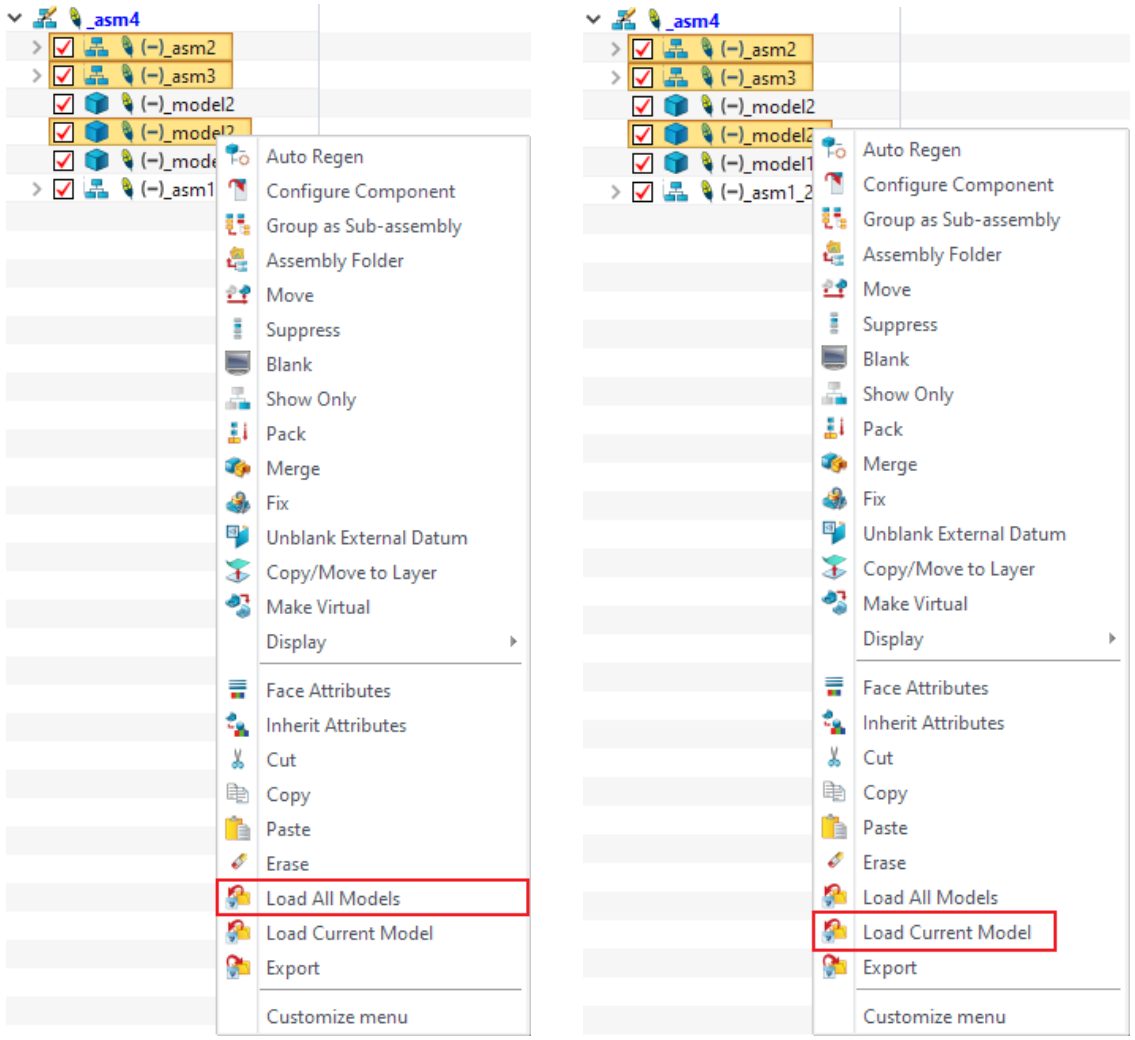
2.1 Input Format Update

The following table lists out the input formats supported in ZW3D 2023, and the updated formats are marked in red.

Format	Postfix	Version Supported
Catia V4	.model, .exp, .session	4.1.9 – 4.2.4
Catia V5/V6	.CATPart, .CATProduct, .CGR	V5R8---V5/V6R2021
Catia_2D	.CATDrawing	V5R8---V5/V6R2021
NX(UG)	.prt	11– NX 1980
Creo(Pro/E)	.prt, .prt.*, .asm, .asm.*	16 – Creo 7.0
SolidWorks	.sldprt, .sldasm	98– 2021(only 64-bit supported)
SolidWorks_2D	.slddrw	2013-2021(only 64-bit supported)
SolidEdge	.par, .asm, .psm	V18 – SE2021
Inventor	.ipt, .iam	.ipt (V6 – V2022) .iam (V11 – V2022)
ACIS	.sat, .sab, .asat, .asab	R1 – 2021 1.0
DWG	.dwg	R11 -
DXF	.dxf	R11 -
IGES	.ige, .iges	
STEP	.stp, .step, .stpz	203, 214, 242
Parasolid	.x_t, .x_b, .xmt_txt, .xmt_bin	Up to 30.0
VDA	.vda	
Image File	.bmp, .gif, .jpg, .jpeg, .tif, .tiff	
Neutral File	.z3n, .v3n	
PartSolutions	.ps2, .ps3	
STL	.stl	
3DXML	.3dxml	4.0 - 4.3
JT	.jt	6.4-10.4
OBJ	.obj	

2.2 Lightweight Model Support Batch Loading

After quickly importing third party assembly, the models inside assembly stay in the state of lightweight. User can select the model to load. There are two loading model commands in ZW3D “Load All Models” and “Load Current Model”. In this version, we improved the commands “Load All Models” and “Load Current Model” which supported selecting multiple part execution on the assembly tree.

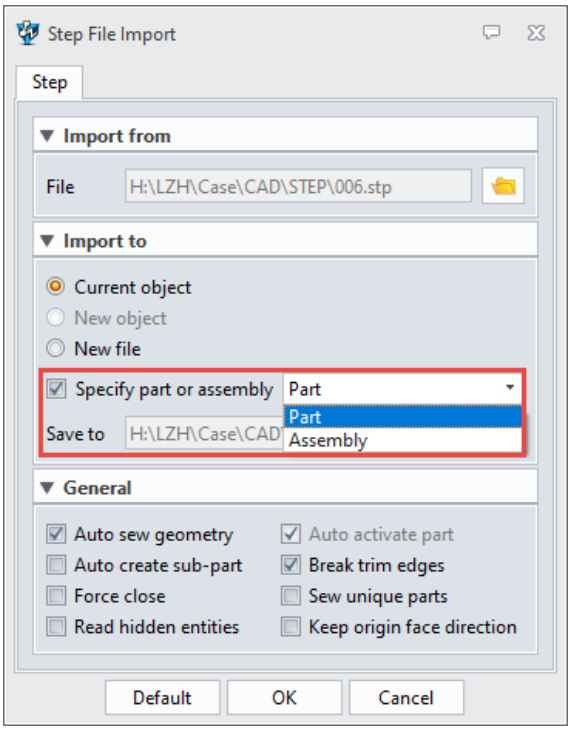


➔ Where it is

Assembly Environment >> Assembly Tree >> Multiple Selection Light Weighted Components >> Load Current Model

2.3 New “Specify part or assembly” in Step File Import

While importing STEP files, if the STEP files are assembly, user can specify to import as **Part** or **Assembly** files. It only takes effect on STEP files when themselves are assembles. The option takes effect when the checkbox is checked.

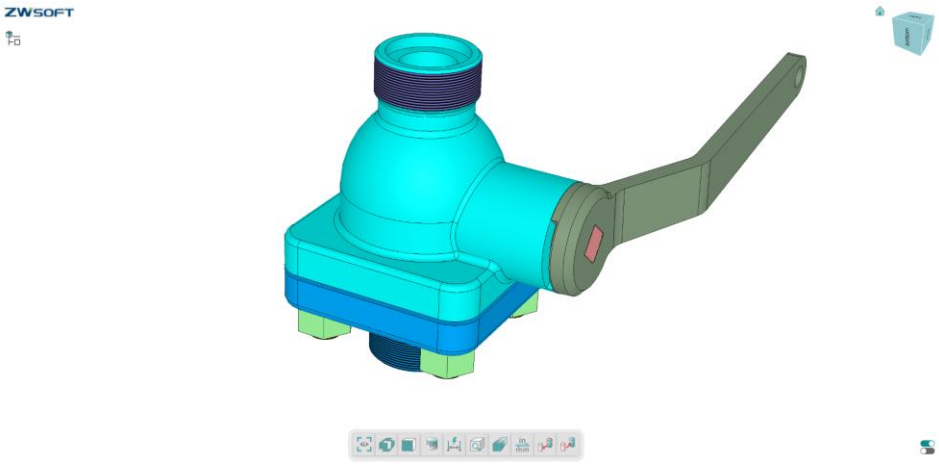


➔ **Where it is**

File >> Import >> Import >> STEP

2.4 Export Html Format Improvement

We improved Html format export files, which can be directly opened through browser.



➔ **Where it is**

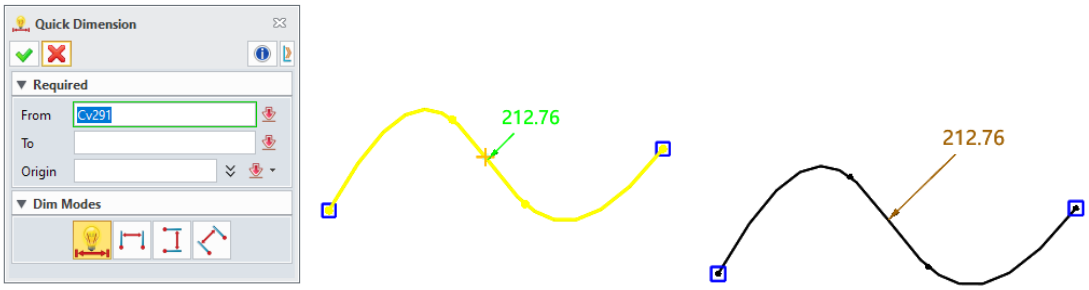
File >> Import >> Html

3 CAD

3.1 Sketch Design

3.1.1 Quick Dimension Supports Perimeter Dimension of Curves

We added perimeter dimension of curves to Quick Dimension.

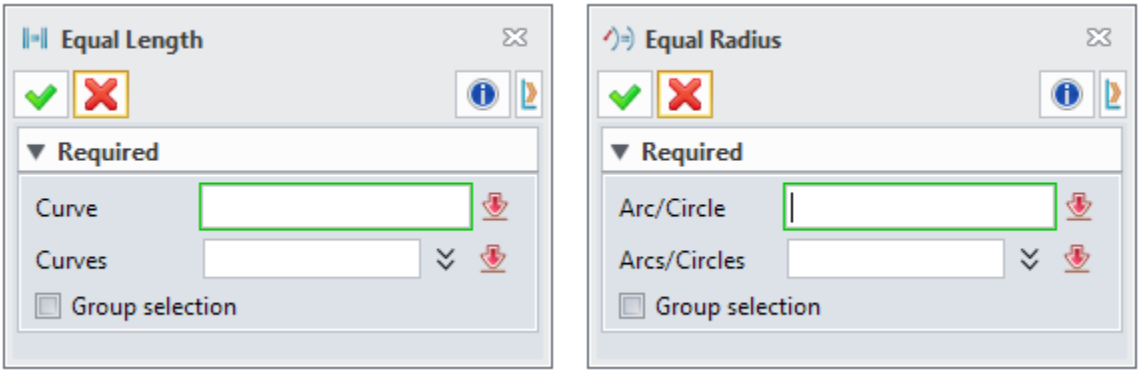


➔ **Where it is**

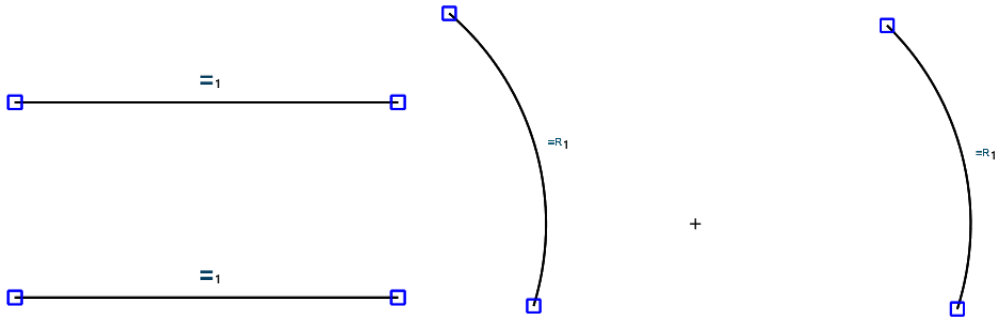
Shape >> Sketch >> Dimension >> Quick Dimension

3.1.2 Equal Length/Radius Improvement

We optimized the constraints of Equal Length and Equal Radius. The former is used to make two curves equal to length, which supports any combination of lines, circles, arcs, and curves as objects; the latter is used to make two circles/arcs equal to radius, which supports any combination of circles and arcs.



Meanwhile, the icons of Equal Length and Equal Radius are distinguished in ZW3D 2023.



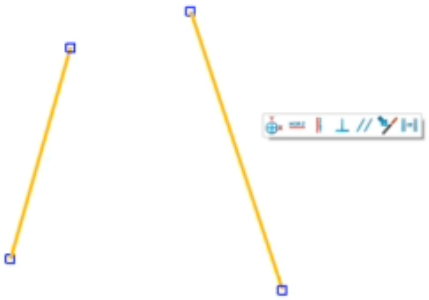
➔ **Where it is**

Sketch Environment >> Constraint >> Equal Length

Sketch Environment >> Constraint >> Equal Radius

3.1.3 ★New Smart Constraint Inference

Automatically infer constraints based on the picked objects: after picking the geometries, it will automatically prompt up the constraint toolbar where user can add constraints. If user does not pick anything, the toolbar will disappear automatically.

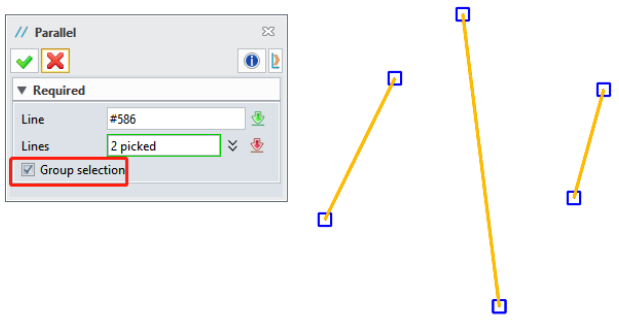


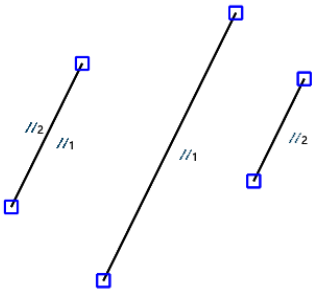
➔ **Where it is**

Sketch Environment >> Picked Objects >> Constraint Toolbar

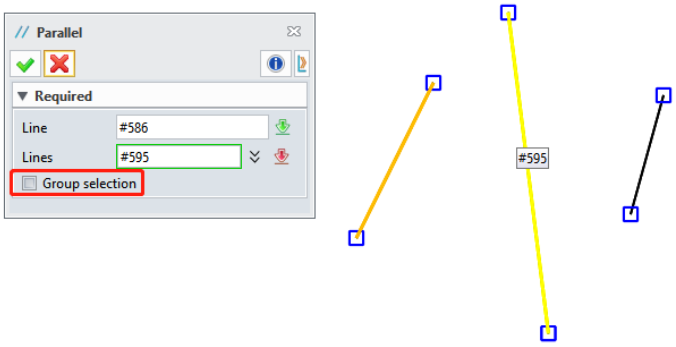
3.1.4 Constraints Support Group Selection

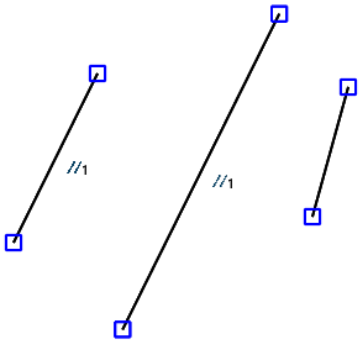
Constraints support continuously inputting. In the constraints Point Horizontal/Points Vertical/Point to Line/Curve/Point Coincident/Perpendicular/Parallel/Collinear/Tangent/Equal Length/Equal Radius/Point to Center, we added an option of Group Selection. When this option is checked, it supports continuously inputting geometries so until user click OK before it executes the constraint condition. When this option is unchecked, any input geometries will immediately execute the constraint if they meet the condition.





"Group selection" is checked, the constraint supports contiously inputting objects, and it will not performed untill you click OK





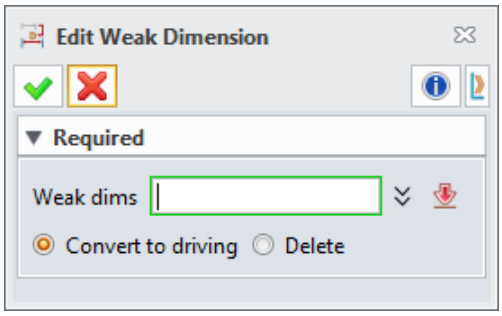
"Group selection" is unchecked, after picking two objects, and the contraint will be directly perform when they meet the contraint conditions

➔ **Where it is**

Sketch Environment >> Sketch >> Constraint >> Point Horizontal/Points Vertical/Point to Line/Curve/Point Coincident/Perpendicular/Parallel/Collinear/Tangent/Equal Length/Equal Radius/Point to Center

3.1.5 New Edit Week Dimensions

We added New Week Dimensions to convert weak dimension to driving dimensions or delete weak dimensions.

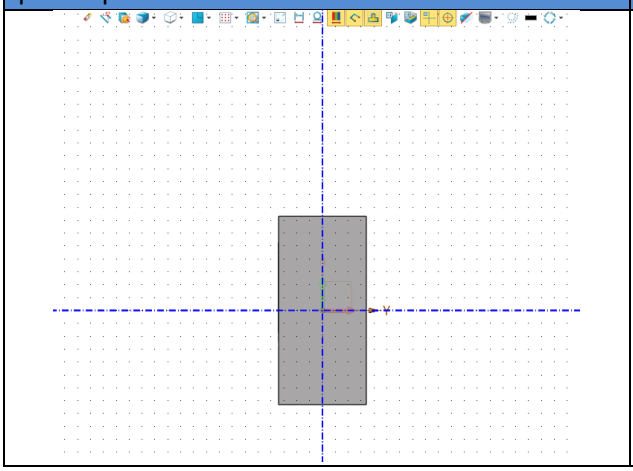
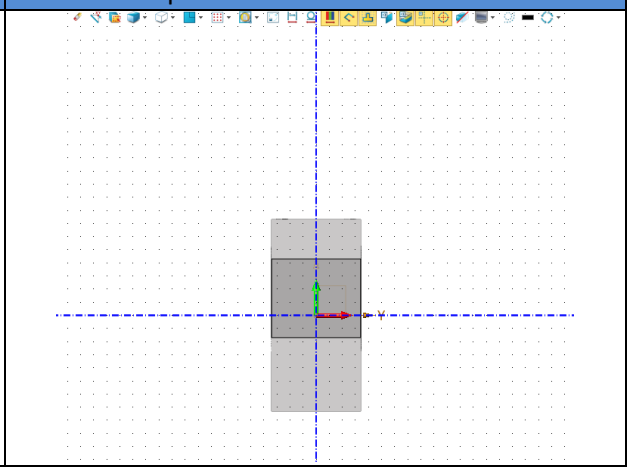


➔ **Where it is**

Sketch Environment >> Sketch >> Settings >> Edit Weak Dimensions

3.1.6 New Section On/Off in Sketch

When sketching, user can section solids/components through sketch plane by the new "Section on/off" and observe or capture the inner objects of solids/components without exiting the sketch. After entering the sketch environment, a new "Section on/off" is added to DA toolbar which can be switched to turn on/off the section view.

When setion is off, the sketch plane is sheltered by partial part	When section is on, user can observe the internal structure of part
	

➔ **Where it is**

Sketch Environment >> DA Toolbar >> Section on/off

3.1.7 Sketch Box Text Supports Dimensions

User sometimes may need to control the overall text's height, width, and position instead to adjust text's parameters separately. In ZW3D 2023, Box Text support dimensioning and positioning as the below

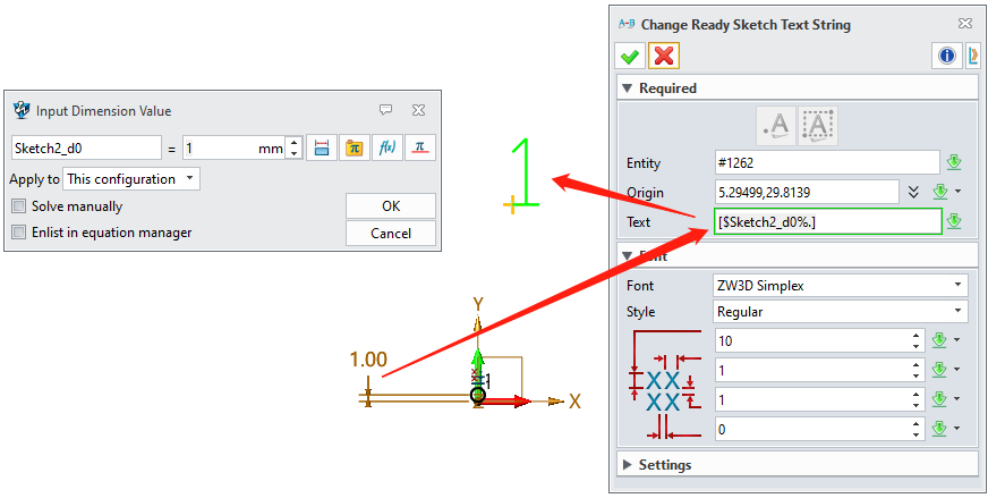
figure shown. The function expands the application range of sketch's text so that user can use it in different scenarios.

➔ **Where it is**

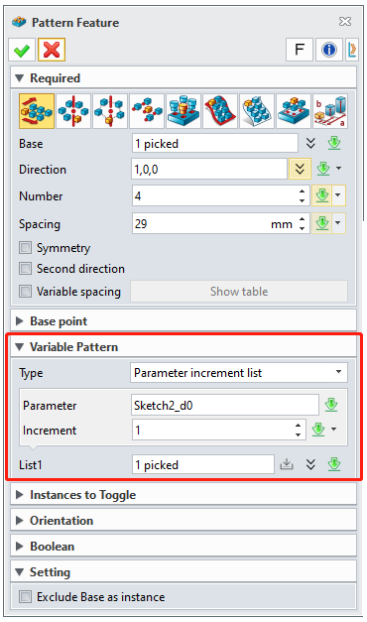
Sketch Environment >> Drawing >> ReadySketch Text >> Box Text

3.1.8 ReadySketch Improvement

ReadySketch supports driving by variables. As the created variable in the following dimension shows:



Sketch text can achieve the increment function when it combines with the feature pattern.



➔ **Where it is**

Sketch Environment >> Drawing >> ReadySketch

3.1.9★2D/3D Sketch External Reference Improvement

3. 1. 9. 1 2D Sketch “External Reference” Improvement

In the 2D sketch environment, we upgraded the “External Reference” command by separating it into two modes: **Projection** and **Intersection.**

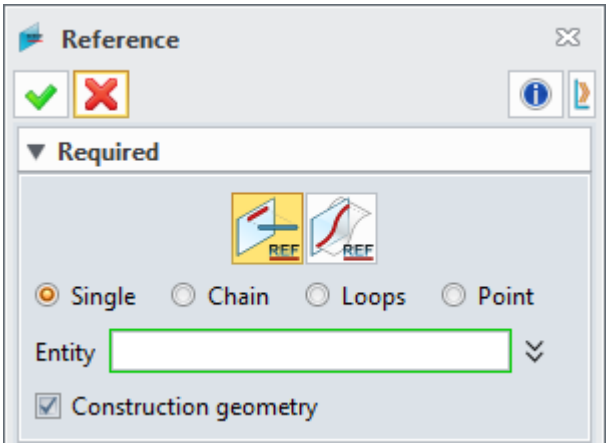
3.1.9.1.1 Projection Mode

Projection means to project the selected external object to the plane where the sketch locates at and takes the projected contour as a reference. If the input object cannot be projected onto the sketch plane, it will ignore the illegal input and prompt up a warning.

The input objects of projection mainly include points, lines, faces, datum planes and datum axes. Therefore, we provided four object input strategies, including **Single, Chain, Loops,** and **Point**. User can only check one mode by selecting a checkbox.

Among the four strategies, **Single** and **Chain** affect the input of lines (including edges, curves, and sketch curves etc.), **Loops** affect the input of faces and **Point** impacts the input of points. While checking **Single** as the strategy, the input edges are all single input. Except for the edges, other objects such as points, faces, datum faces, and datum axes can still be selected. The input of faces follows the type of default in Loops, which is the type of All.

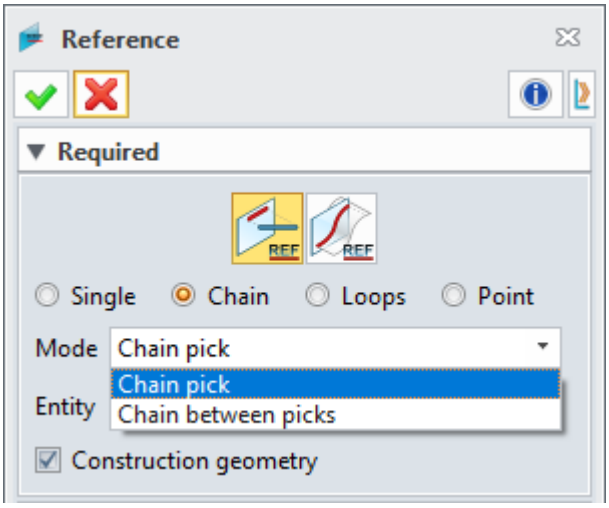
In **Single** input, while selecting an object in the drawing area, do not need to click OK to generate reference immediately. The reference chain information will display immediately in the below panel of Ext Reference Manager. **Single** input supports box selecting input or pressing Ctrl for multiple input.



If checked Chain, there are two modes in the drop-down list: **Chain pick** and **Chain between picks**.

Chain pick refers to automatically select the tangent lines (edges, curves, sketch curves etc.) of the selected line. After picking the line, user does not need to click OK to generate results immediately. The mode supports box checking or pressing Ctrl for multiple input and only impacts on the line selection. As for other objects such as points, faces, datum planes, and datum axes can still be picked by Single mode.

In Chain between picks, while picking two lines, the system will select the edges of the chain that connects the two lines. Chain input will not take effect immediately while inputting the first line. Only when input the second line will it take effect. If the two lines cannot form a chain, they will be taken as single inputs separately. The command does not support box checking or pressing Ctrl for multiple input.



Loops focus on the picking face loops. The rule only takes effect on faces. Other objects such as points, lines, datum faces, and datum axes can be still picked by Single. The mode supports box selecting or Ctrl for multiple input.

The drop-down menu lists out loop picking rules including:

All: Input all loops onto the faces.

Boundary: while picking a face, the input is the face boundary including outer loop and inner loop; while picking adjacent faces, the input is the most outer boundary of selected face (which means unpick the middle adjacent edges); while picking multiple non-adjacent faces, the input of each face's boundary including outer loop and inner loop separately. (Boundary includes open boundary and closed boundary)

Shared: While picking multiple faces, the input is the shared boundary between the multiple faces.

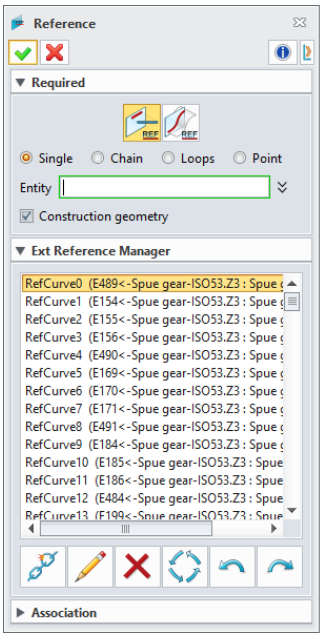
Inner: Input the inner loops of the picked face.

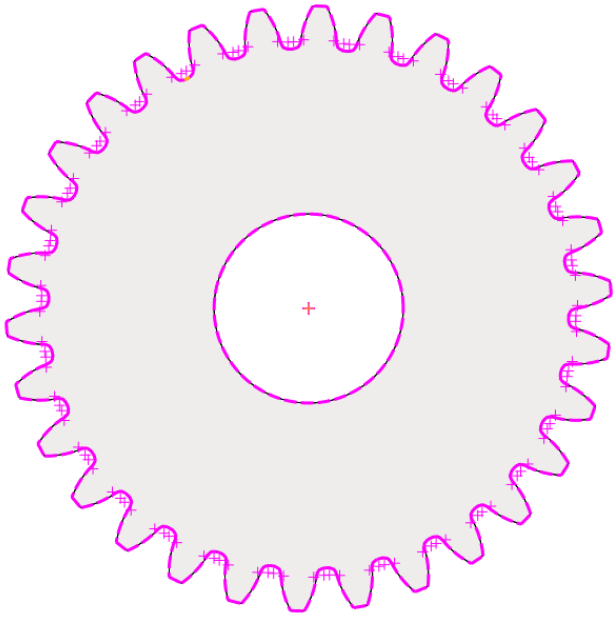
Outer: Input the outer loops of the picked face.

Open: Input the open boundary of the picked face.

Non-shared boundary: Input the non-shared boundary of the picked face. (Only include closed boundary)

Silhouette: Input the outer contour of the picked face.

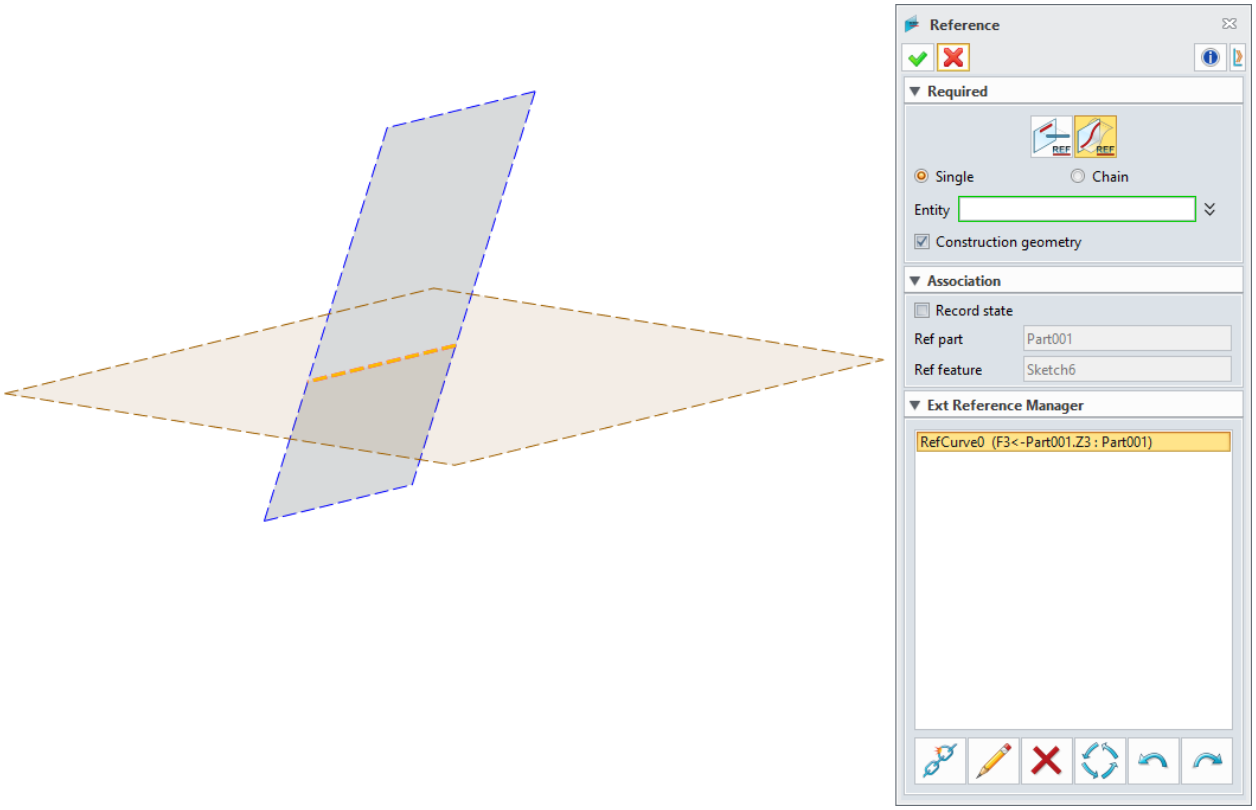




3.1.9.1.2 Intersection

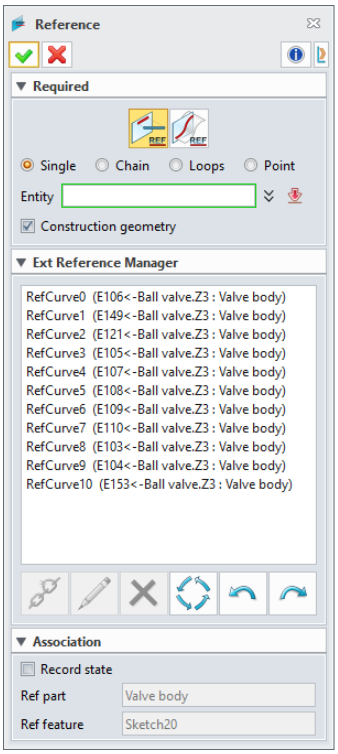
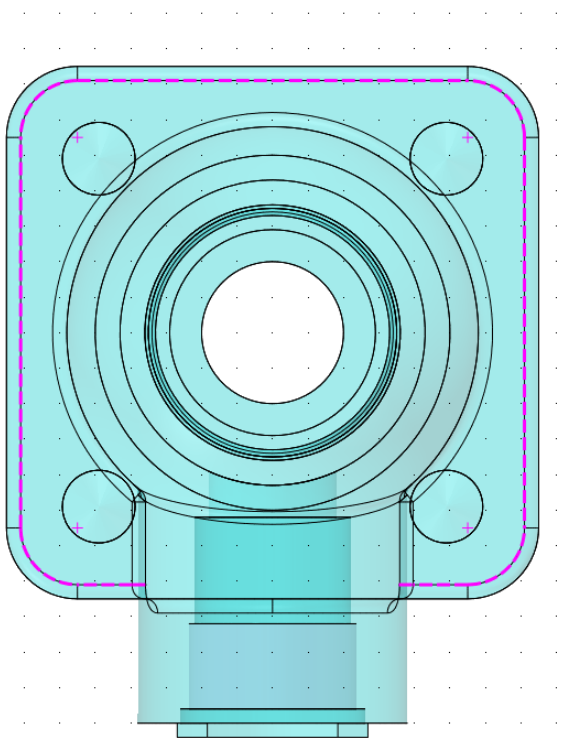
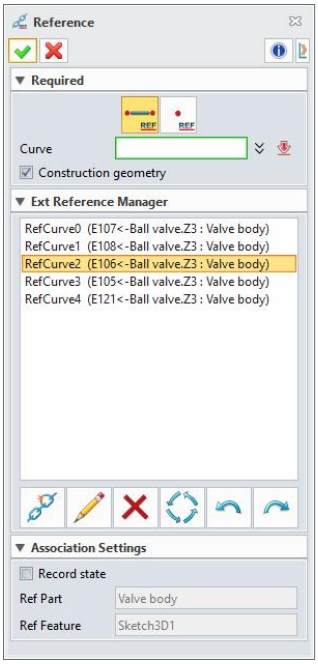
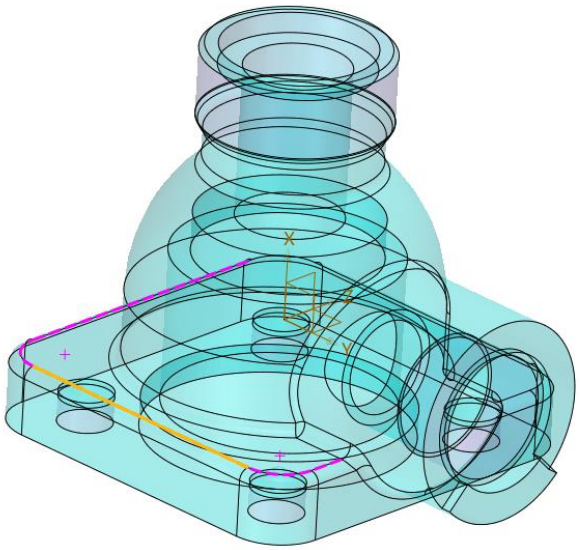
Intersection means to take the intersecting line as a reference when the picked external object is interacted with the sketch plane. If there is no intersecting line between the input object and the sketch plane, it will ignore the illegal input and prompt a warning.

The input objects of intersection mode mainly include lines, faces, datum planes, and datum axes. There are two rules of intersection mode: **Single** and **Chain**.



3.1.9.2 Ext Reference Manager

When 2D/3D sketch refers to external references, it often needs to add, delete, or modify references in manager. To facilitate user's operation, we merged the external reference manager into the reference panel. Support Shift or Ctrl for multiple selections. The manger includes Unlink, Edit, Delete, Regen, Undo, and Redo buttons. In the cases of multiple choices, without selecting objects or the command unable to use, the buttons become grey.

2D Sketch External Reference Manager	 
3D Sketch External Reference Manager	 

➔ **Where it is**

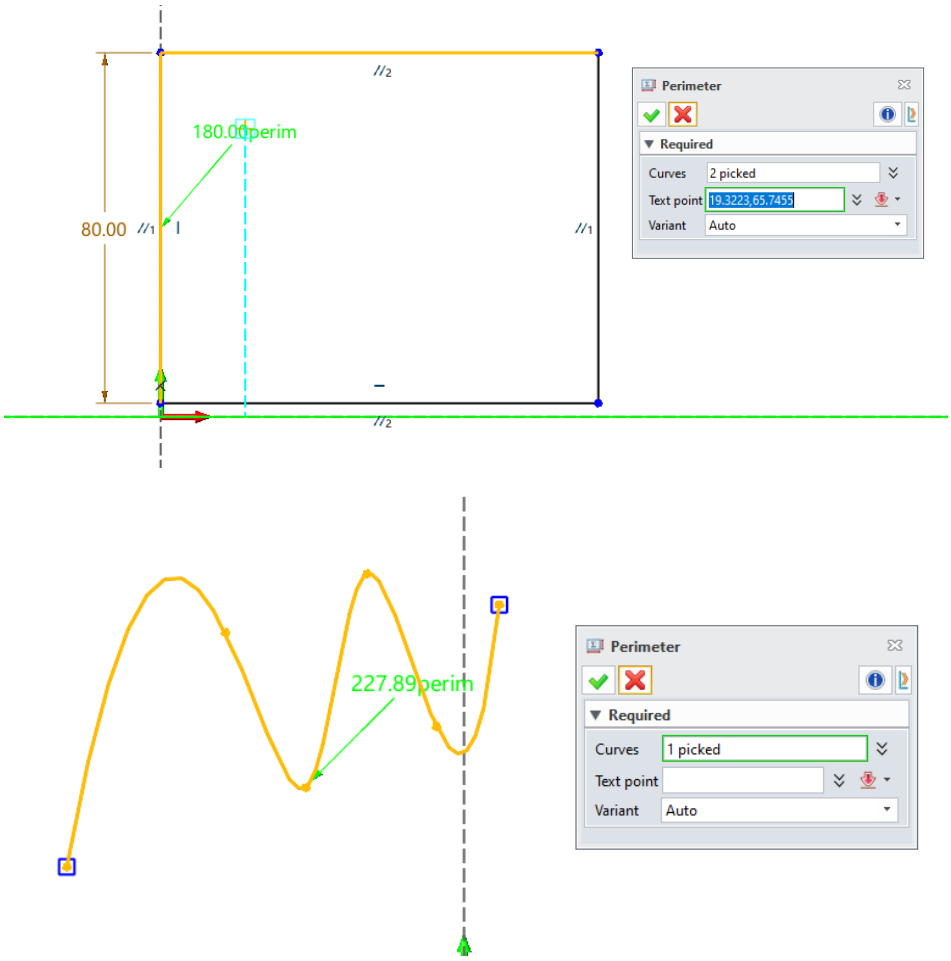
Sketch Environment >> 2D/3D Sketch >> Reference >> Reference

3.1.10 Dimension Improvement

3.1.10.1 New Perimeter Dimension

In the sketch environment, we added Perimeter which is used to constrain the length of the curve chains that are composed by the lines, arcs, and curves. The curve chain can be continuous as well as noncontinuous and achieved by a single dimension as variant.

Perimeter dimension is marked by label and the identification is on the first curve of the picked curve chain. Besides, the "**perim**" is marked along with the label value to indicate that is a perimeter dimension, as the below figure.

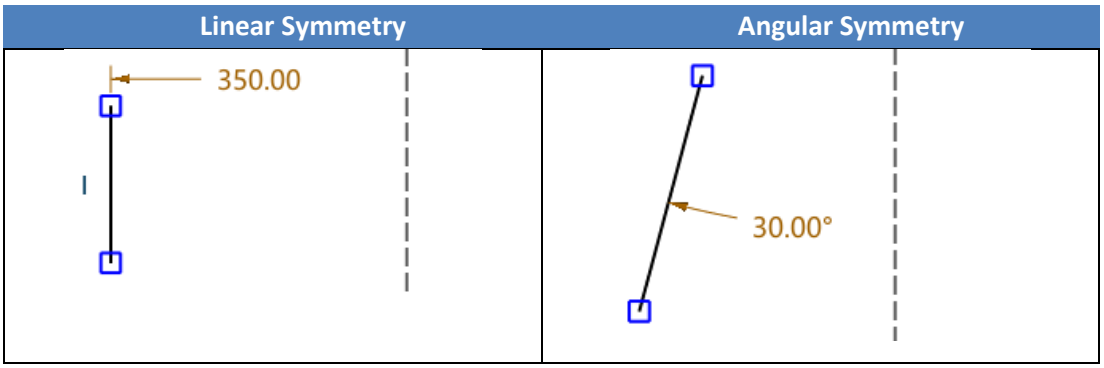


➔ **Where it is**

Sketch Environment >> Sketch >> Dimension >> Perimeter

3.1.10.2 New Symmetry Dimension

Like the symmetry dimension in the drawing sheet, we added Symmetry in the sketch that was separated into linear symmetry dimension and angular symmetry dimension.

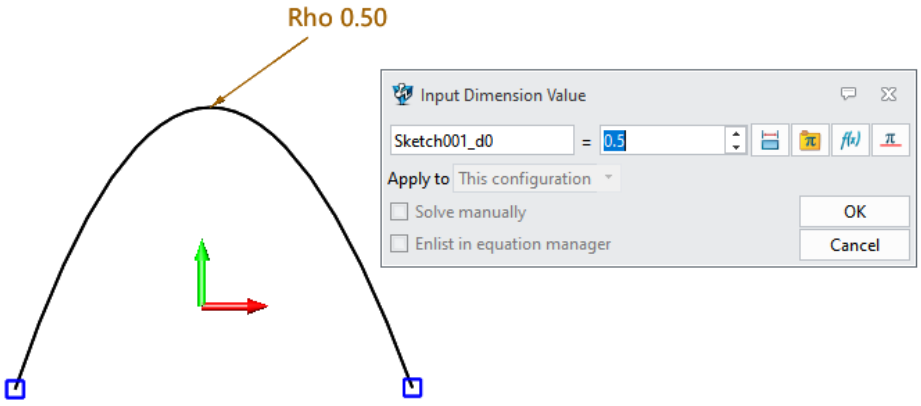


➔ **Where it is**

Sketch Environment >> Sketch >> Dimension >> Symmetry

3.1.10.3 Support Adjusting Rho Value in 3 Point Conic

After successfully created a 3-point conic, the system automatically assigns Rho value and dimensions it, which can be modified by user.



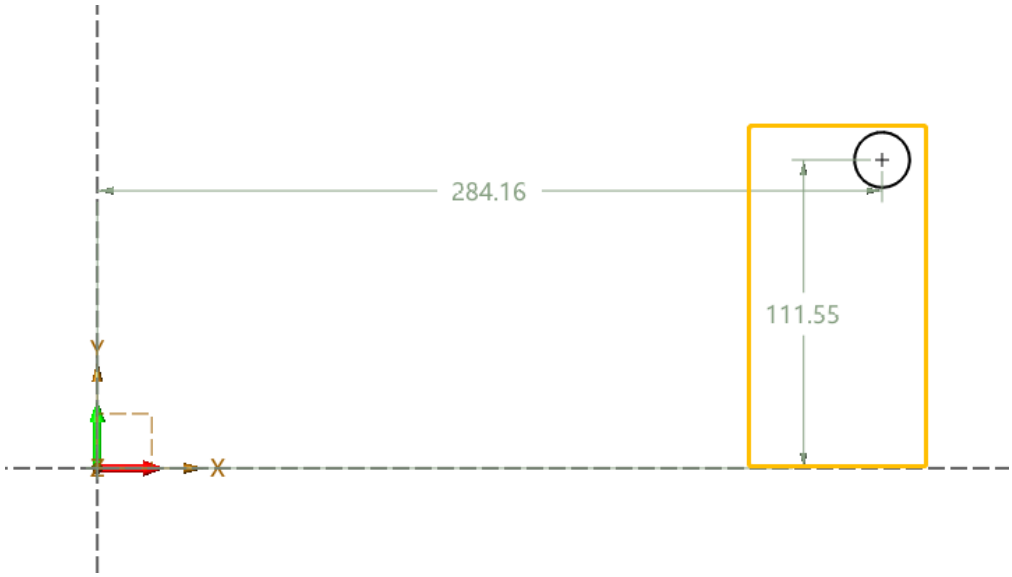
➔ **Where it is**

Sketch Environment >> Sketch >> Curve >> 3 Point Conic

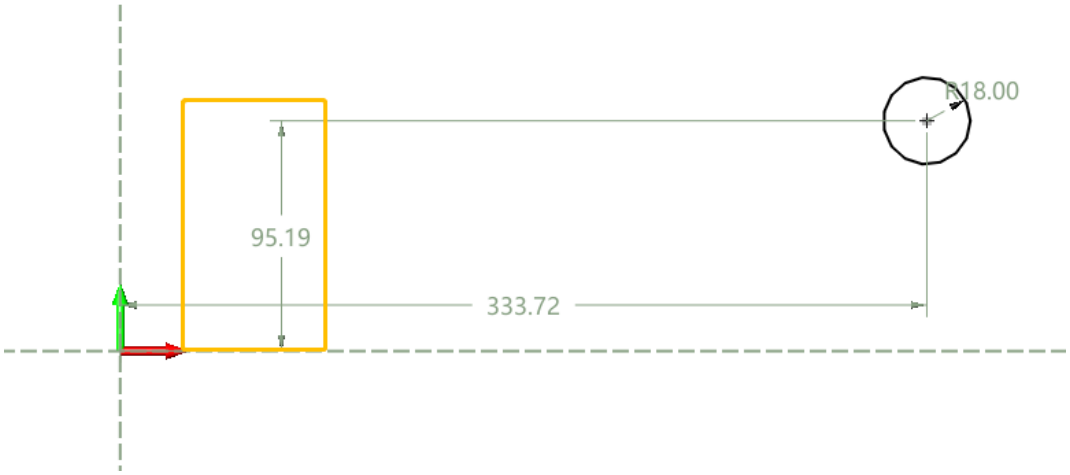
3.1.10.4 Weak Dimension Improvement

When “Add Weak Dimensions Automatically” is enabled, the dimension will be automatically placed near the drawn graphics, rather than close to the reference object according to the area where the figure locates. Therefore, user can quickly modify the dimension and reduce the scaling and moving operations.

The improved weak dimension closes to the drawing figure.



Before that, the weak dimension is closed to the coordinate. In zooming in the view, user needs to scale or move the view to find the corresponding dimension, which was unfit for operation.



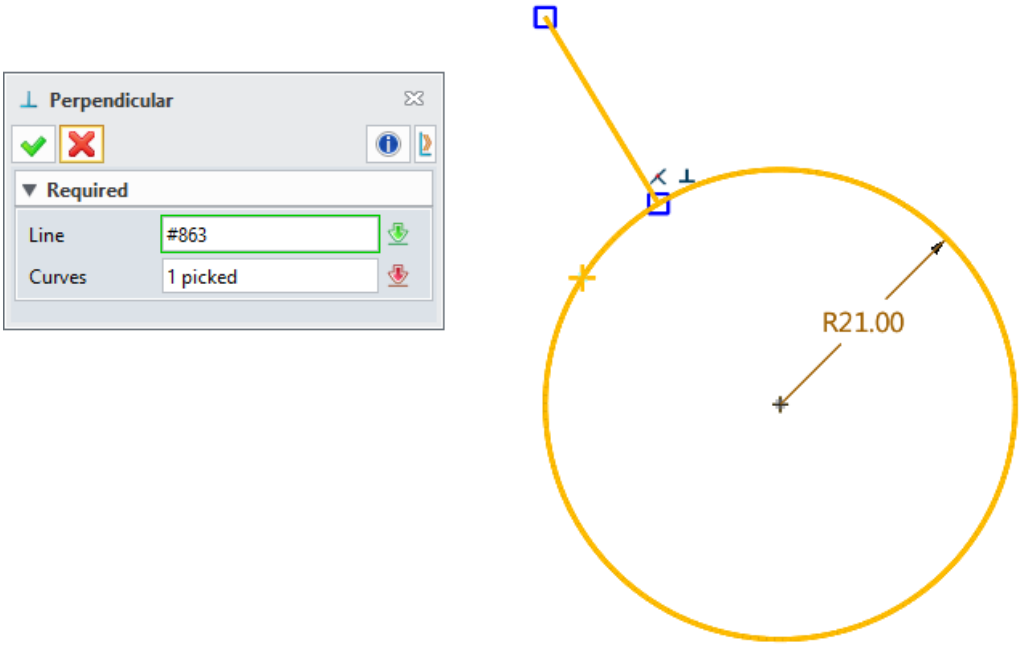
➔ Where it is

Sketch Environment >> Menu >> Curve >> Add Weak Dimensions Automatically

3.1.11 Constraint Improvement

3.1.11.1 New Vertical Constraint Between Line and Circle (Arc)

In the sketch environment, we added a vertical constraint between lines and circles (arcs).

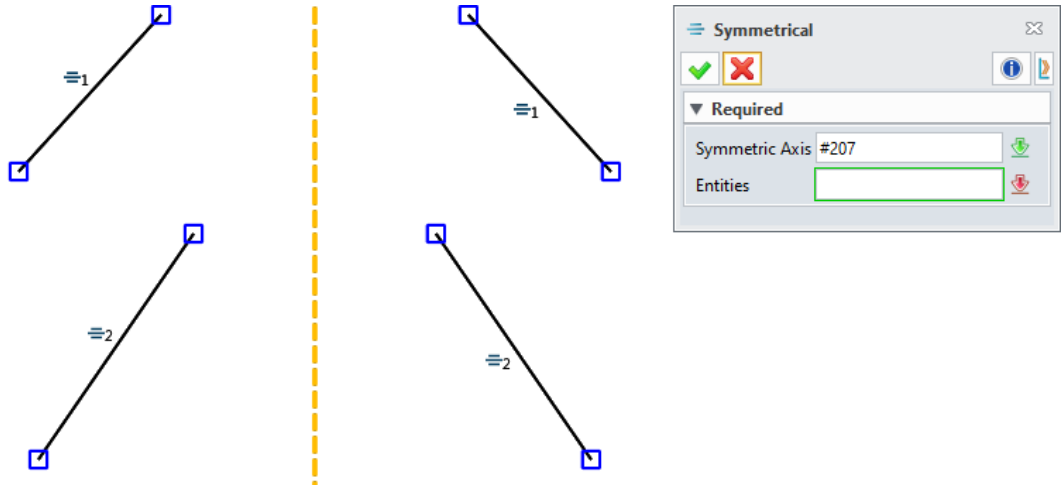


➔ **Where it is**

Sketch Environment >> Sketch >> Constraint >> Add Constraint

3.1.11.2 Symmetrical Constraint Improvement

As the following figure shows, after selecting the symmetrical axes, the mouse automatically jumps to the entities selection box while defining the symmetrical constraints. Meanwhile, user can continuously select the entity objects that need symmetrically constraining. It will immediately execute a pair of entities after selecting. User does not need to repeat picking the symmetrical axes so to increase the efficiency of selecting multiple pairs until the command closed.

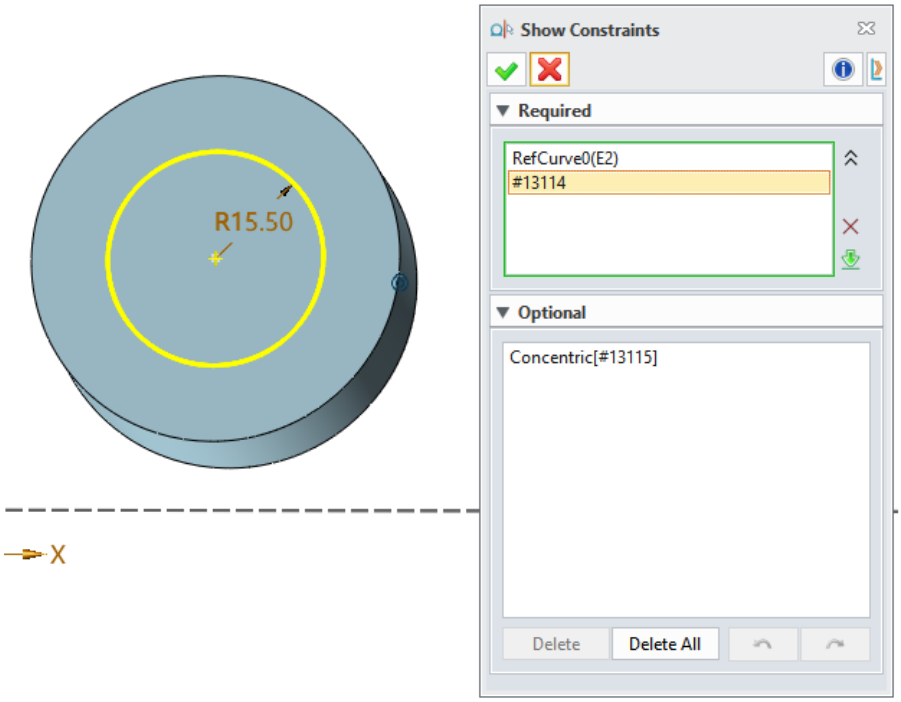


➔ **Where it is**

Sketch Environment >> Sketch >> Constraint >> Symmetrical

3.1.11.3 Quickly Search Constrained Object

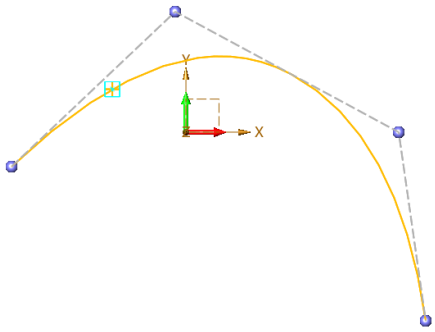
Double click the constraint icon to enable the "**Show Constraints**" command. As the following figure, to find the constrained object, user can double click the concentric constraint icon to quickly show the object of the constraint.



3.1.12 Sketch Spline Improvement

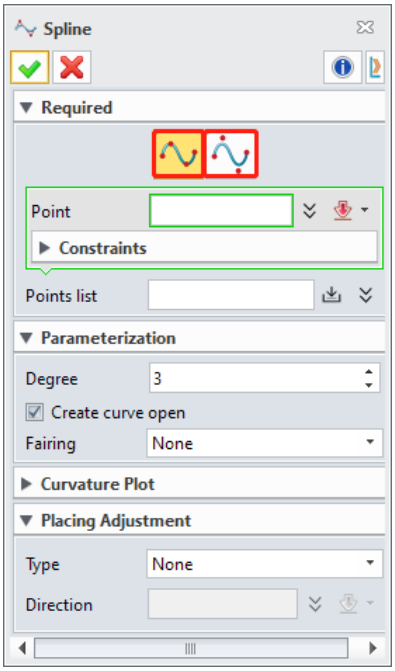
3.1.12.1 New Through Point During Spline Drawing

During the drawing, we found that the inserting point of local shape or the number of control points were not sufficient to adjust the shape and needed adding new points onto the spline that had been drawn. So, we added a new point function of inserting point during the spline drawing to improve efficiency.



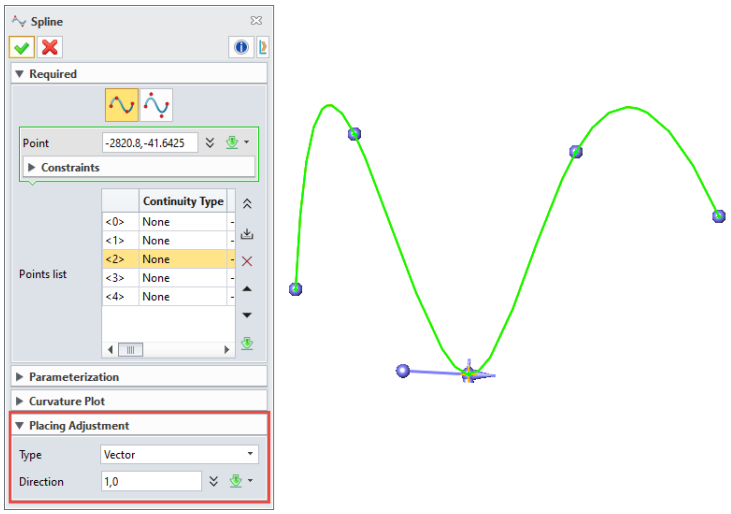
3.1.12.2 Combination of Through Point and Control Point into Spline

Through Point and Control Point splines are frequently used in the sketch environment. User often needs to switch the spline type during the drawing. Thus, we combined Through Point and Control Point into a new command “Spline” to better switch the types and reduce the repeated action.



3.1.12.3 New Placing Adjustment

We added “Placing Adjustment” in the spline command to better assist user adjusting the node position after drawing spline. There offer multiple methods to adjust node placement.



1) None (2D sketch) /View (3D sketch)

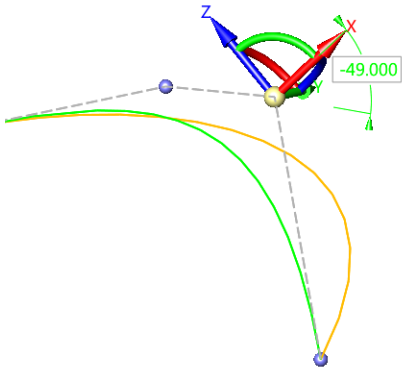
When through points and control points are adjusted, “Placing Adjustment” does not work but utilizes the rules of previous version by default.

2) Vector

When through points and control points are adjusted, it will move along the vector specified by the user.

3) WCS (3D sketch)

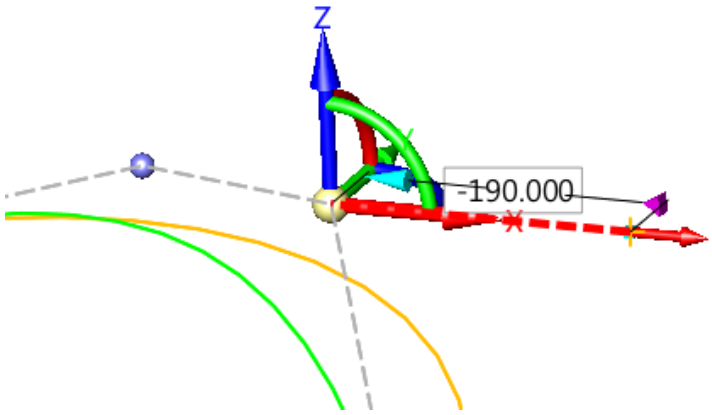
When through point and control point are adjusted, an operator will be displayed in the graphic interface where users can drag its axis line and the node will move along with the corresponding axis line. Drag the operator arc to adjust the attitude of the operator.



4) Plane (3D sketch)

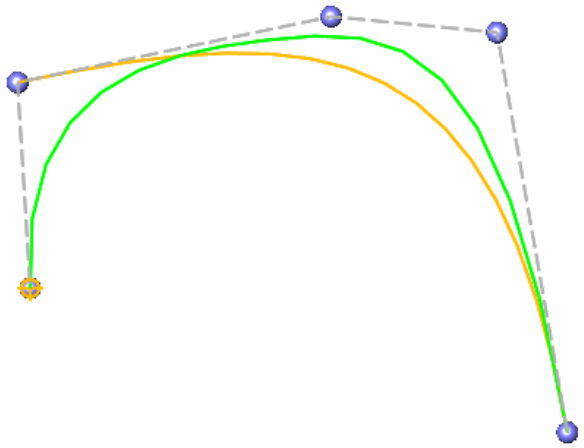
When through point and control points are adjusted, an XY operator (parallel to user-defined plane) will display in the graphic interface. The relevant nodes will move axis line while dragging the operator.

User can adjust the operator posture with dragging it in arc line. By checking “Project to plane”, the relevant nodes can be moved to the projected point of the plane and the adjusted nodes are on the specified plane.



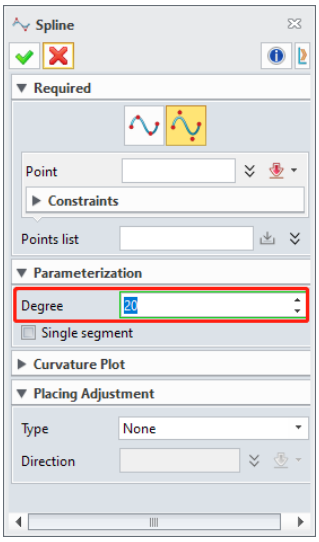
3.1.12.4 New Confirmed Points Spline Preview

The spline shape will change as the mouse moves during the drawing. We added preview function of “Placing Confirmed Spline” (see the yellow line in following figure) in the graphic interface for user to better figure out the changed spline.



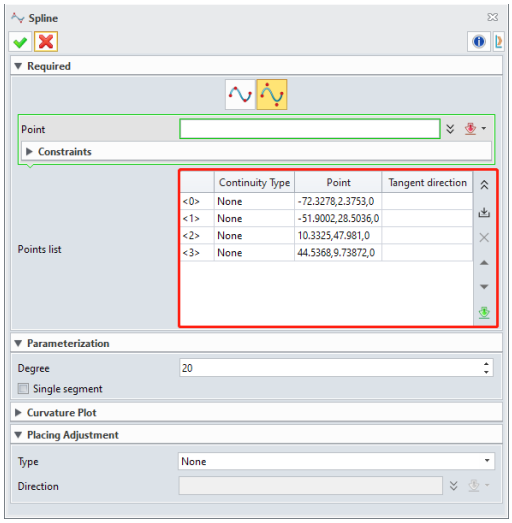
3.1.12.5 Control Point Spline Degree Up to 20

The control point spline is the basic element of surface construction. The smoothness quality of curve determines the quality of surface. To increase the spline smoothing of multiple control points, we raised the degree of control point spline to 20 degrees.



3.1.12.6 Spline Points List Optimization

Points list serves as one of the methods for user to intuitively obtain the spline information. We added the columns of Continuity Type, Point and Tangent direction in Points list for user to better get necessary information.

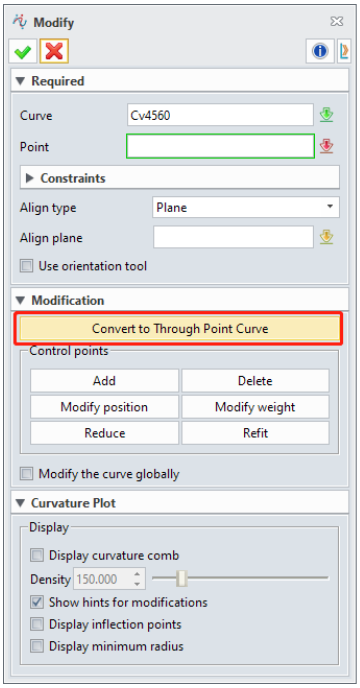


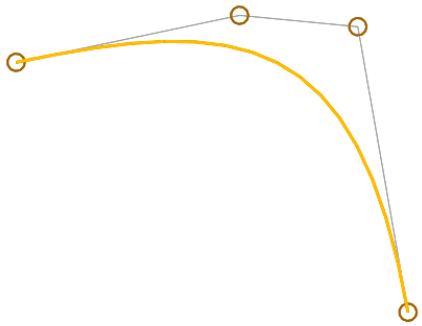
➔ Where it is

Sketch Environment >> Sketch >> Curve >> Spline

3.1.12.7 New Convert to Through Point Curve

It is a common application scene during the drawing process that Through point and Control point convert to each other and keep shapes unchanged. We added “Convert to Through Point Curve” in the new version.



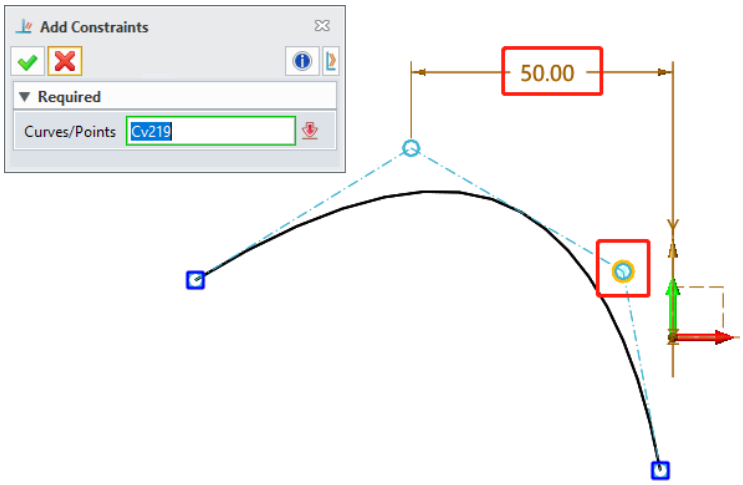


➔ Where it is

Sketch Environment >> Sketch >> Edit Curve >> Modify

3.1.12.8 Dimension and Constraint Support Picking Control Point

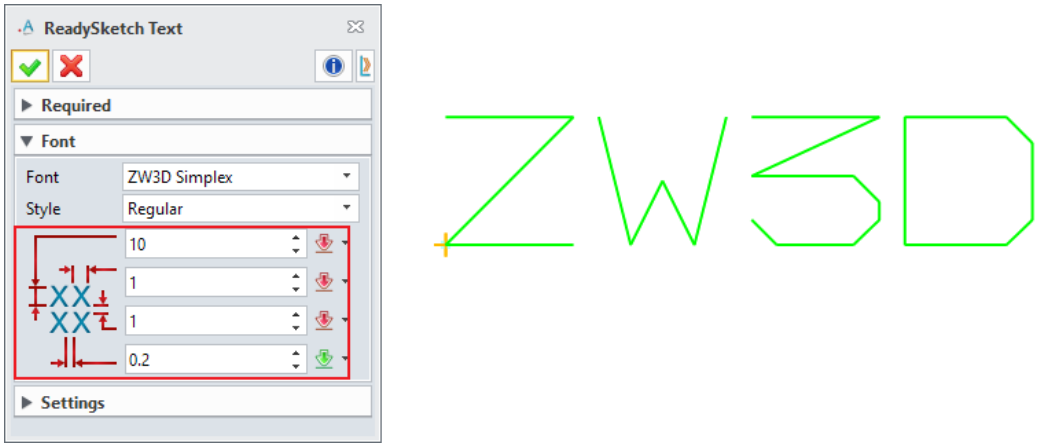
In the new version, user can utilize Dimension or Constraint commands to constrain the position of spline control point while drawing the sketch with Control point spline.



3.1.13 ReadySketch Text Improvement

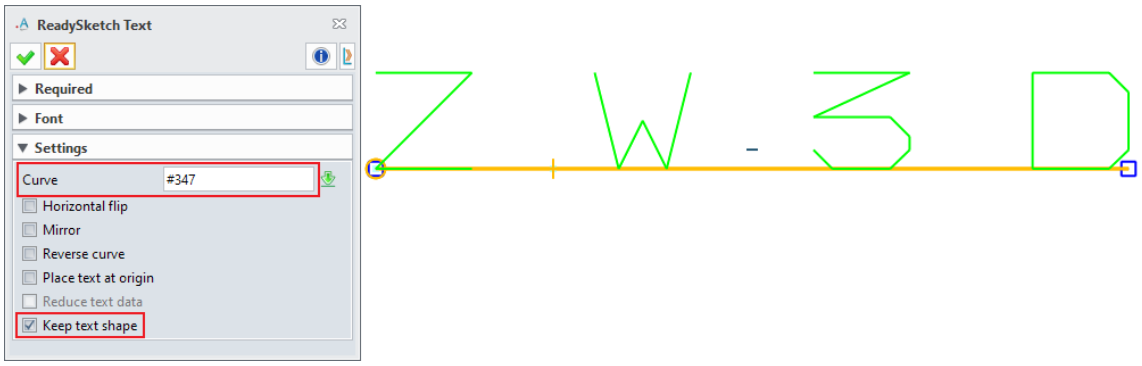
3.1.13.1 New Font Style Setting

In the sketch environment, user often needs to use ReadySketch Text. The settings of height of text, width of text, vertical spacing between text, and horizontal spacing between text are supported in ZW3D 2023, which allow user to set the text font more freely.



3.1.13.2 New Keep Text Shape

We added “Keep text shape” for setting the length of text. After checking the option, the total length of ReadySketch text and the length of the selected curve keep the same. Therefore, user can control the total length of text through adjusting the length of curve.

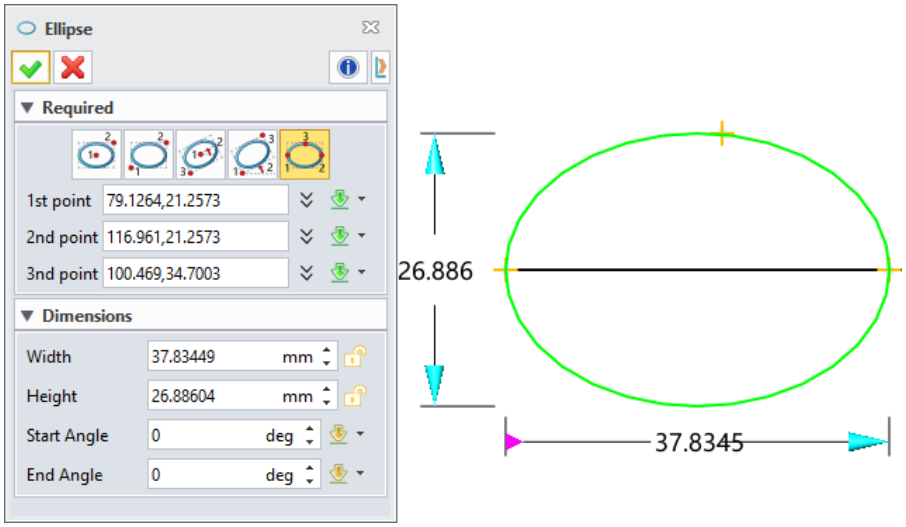


➔ **Where it is**

Sketch Environment >> Sketch >> Drawing >> ReadySketch Text

3.1.14 Ellipse Improvement

In the sketch environment, user can create an ellipse by defining a long endpoint and a short endpoint.

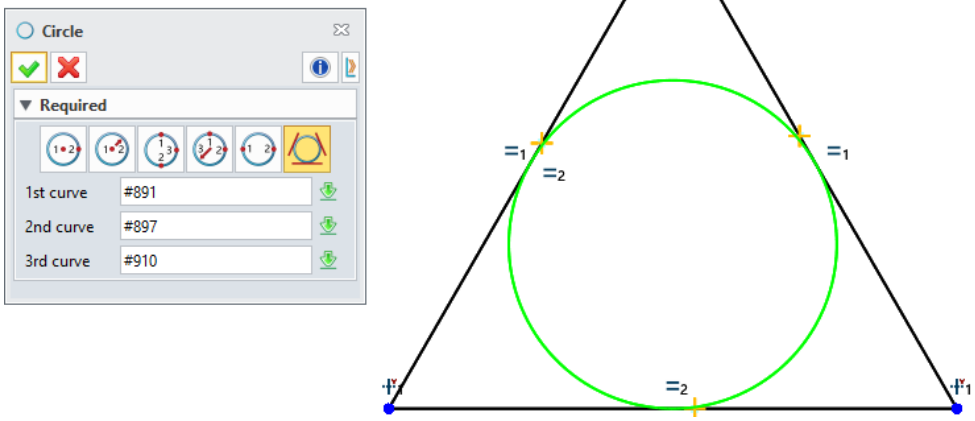


➔ **Where it is**

Sketch Environment >> Sketch >> Drawing >> Ellipse

3.1.15 New 3 Tangent Circle

In the sketch environment, we added a 3 Tangent command by which user can quickly draw a three-line-tangent circle.

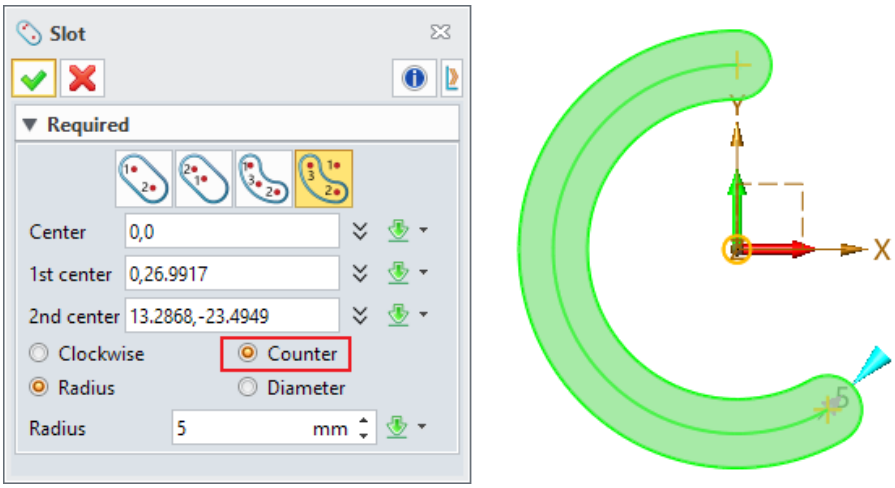


➔ **Where it is**

Sketch Environment >> Drawing >> Circle

3.1.16 Support Creating Direction in Center Arc Slot

In the sketch environment, we added “Clockwise/Counter” in center arc slot so that user can draw a slot according to different direction.

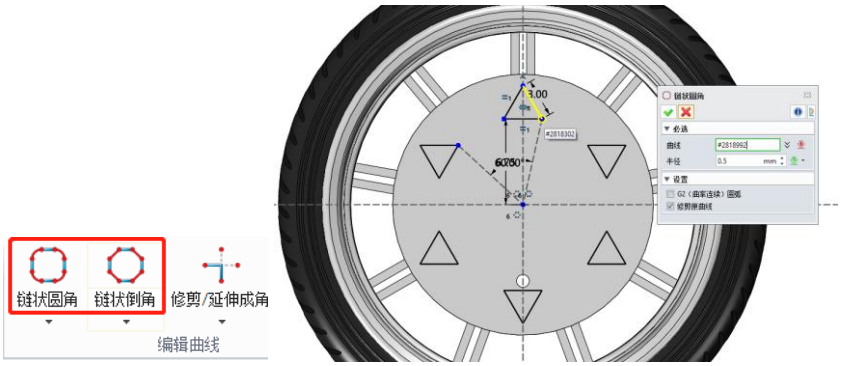


➔ **Where it is**

Sketch Environment >> Drawing >> Slot >> Center arc slot

3.1.17 Fillet Chain/Chamfer Chain Support Picking End

“**Fillet Chain/Chamfer Chain**” support picking an end so that the system can draw a fillet or chamfer based on the two edges of the end.



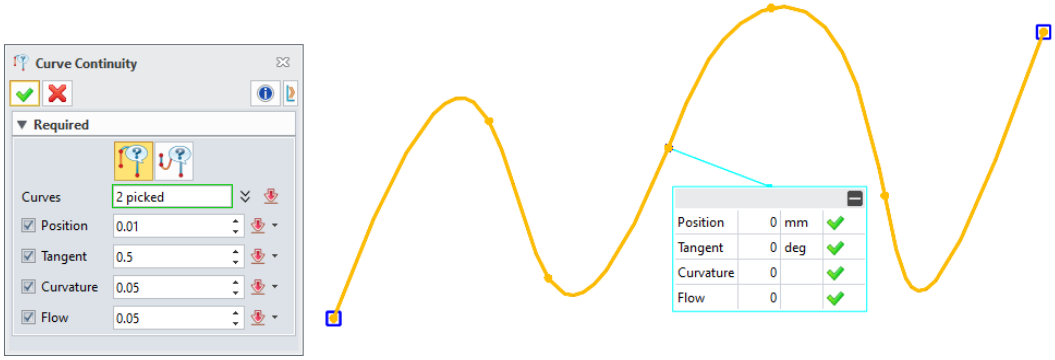
➔ **Where it is**

2D/3D Sketch Environment >> Sketch >>Edit Curve >> Fillet Chain/Chamfer Chain

Part Environment >> Wireframe >>Edit Curve >> Fillet Chain/Chamfer Chain

3.1.18 ★Sketch Curve Continuity Inquire Improvement

In the sketch environment, we improved the "Curve Continuity" inquire which supports inquiring G0/G1/G2/G3 continuity. The inquiring result can be displayed in the drawing area. User can set the inquire type and tolerance value. The items that meet the inquiring results will be shown in green and the items that do not meet will be shown in red. Meanwhile the deviation values and units of the inquiring results will be displayed.

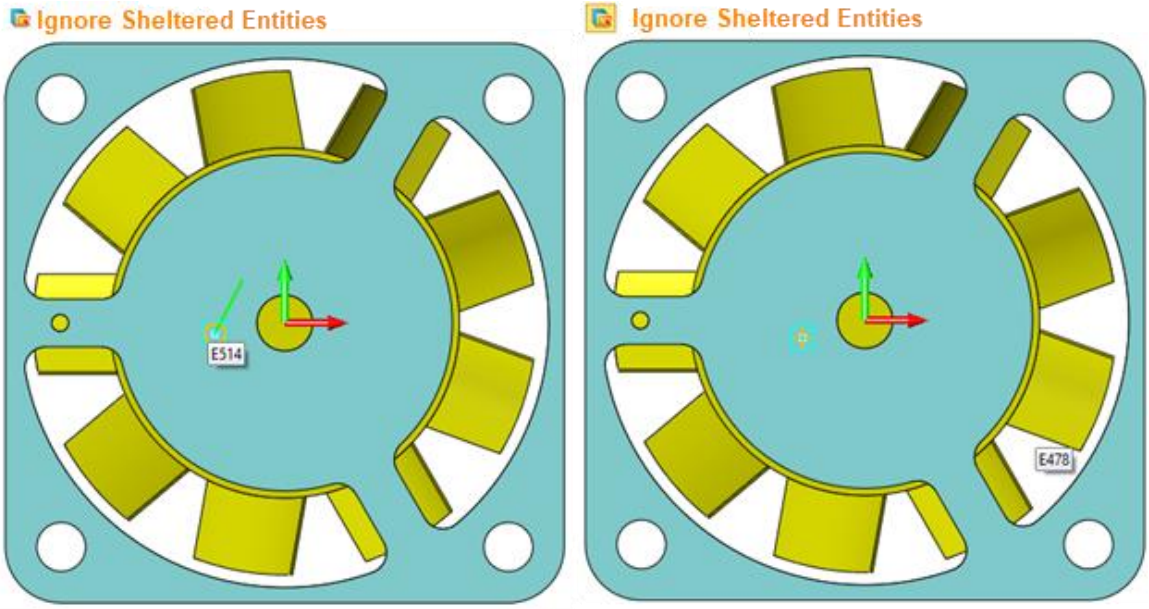


➔ **Where it is**

Sketch Environment >> Inquire >> Inspect Entities >> Curve Continuity

3.1.19 New Ignore Sheltered Entities

In the sketch environment, we added a function of Ignore Sheltered Entities. After enabling, the sheltered entities cannot be selected while executing box selection so that to reduce the interference from the external object in drawing.

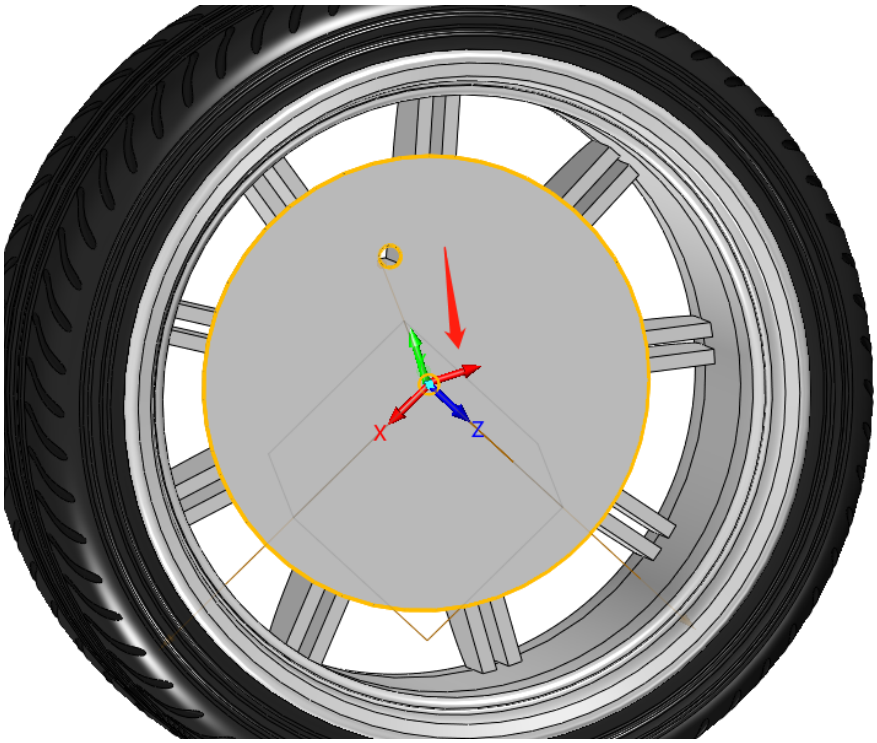


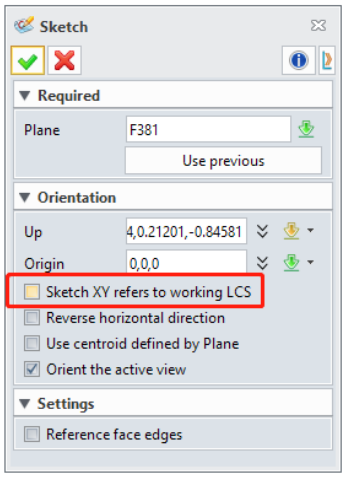
➔ **Where it is**

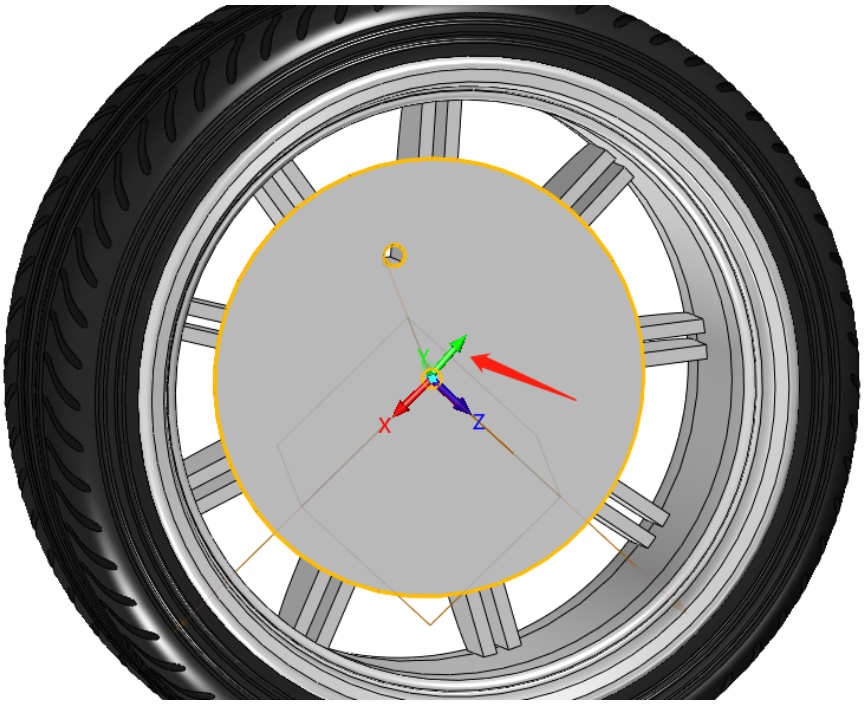
Sketch Environment >> DA Toolbar >> Ignore Sheltered Entities

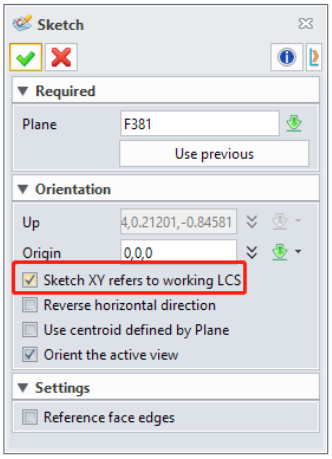
3.1.20 New Sketch XY Refers to Working LCS

We added an option of “Sketch XY refers to working LCS” to Sketch. When it is checked, XY direction will be automatically adjusted according to the current screen and LCS. Otherwise, take UV direction of referring placed plane as the created sketch orientation.







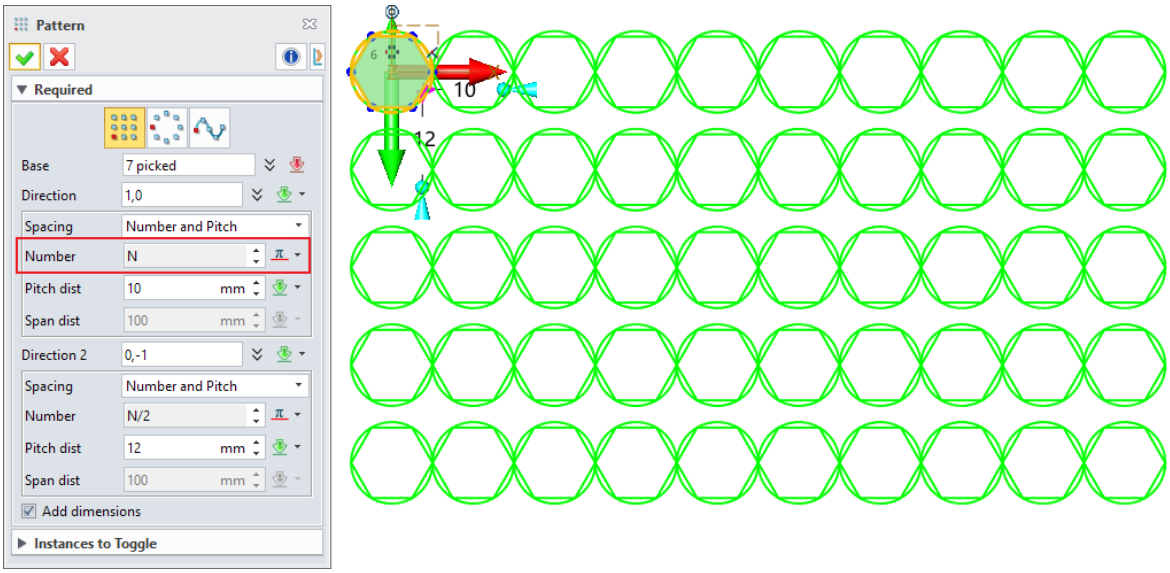


➔ **Where it is**

Part Environment >> Shape >> Basic Shape >> Sketch

3.1.21 Sketch Pattern Improvement

In the sketch environment, user often needs to set pattern number with parameter. To control pattern through parameterization, we improved the pattern of sketch in ZW3D 2023 so that user can control the pattern number by expression.



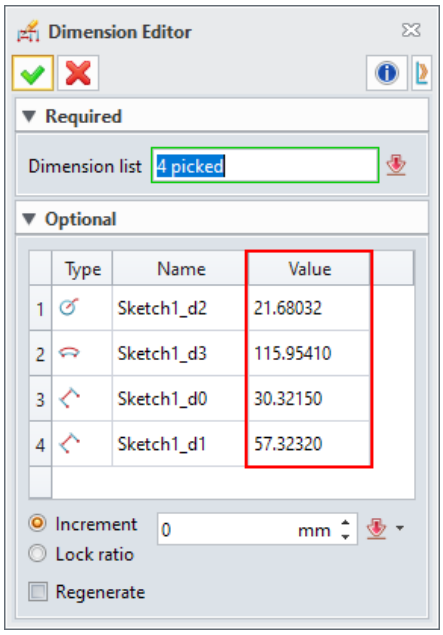
➔ **Where it is**

Sketch Environment >> Sketch >> Basic Edit >> Pattern

3.1.22 Dimension Editor Improvement

We improved the “Dimension Editor” function and supported batch editing of higher accuracy (10e-5) dimension, so that user can define sketch dimension more precisely and clearly.

The “Dimension Editor” command is not only applied in the 2D sketch environment but also in the 3D sketch environment.



➔ **Where it is**

Sketch Environment >> Sketch >> Settings >> Dimension Editor

3.1.23 Power Trim Improvement

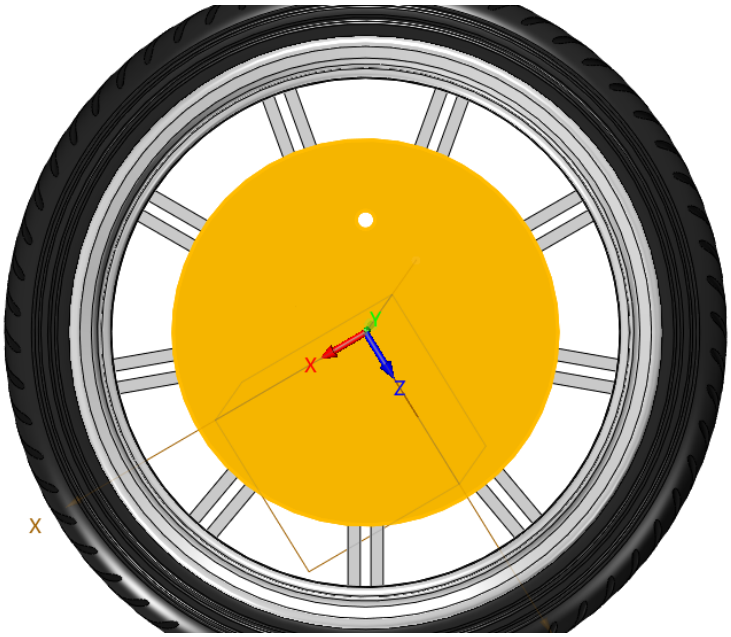
In the sketch environment, user often utilizes “Power Trim” to trim redundant primitives after drawing the basic primitives. As for complicated primitives, it may take multiple mouse actions to complete trimming. ZW3D 2023 supports continuously using of “Power Trim” to reduce repeated action and advance the drawing efficiency. After finishing a trimming, user can continue to trim the primitives by the mouse.

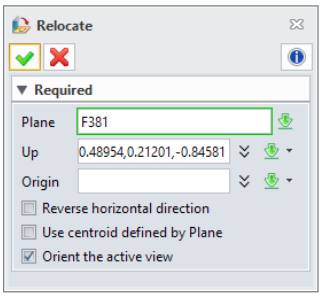
➔ **Where it is**

Sketch Environment >> Sketch >> Edit Curve >> Power Trim

3.1.24 Sketch Relocate Default Activation Update to “Plane”

The common scene of relocating sketch is to change the reference datum plane. Thus, the default activated input box of Relocate is changed to “plane input box”.





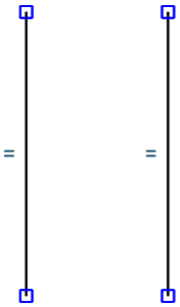
➔ **Where it is**

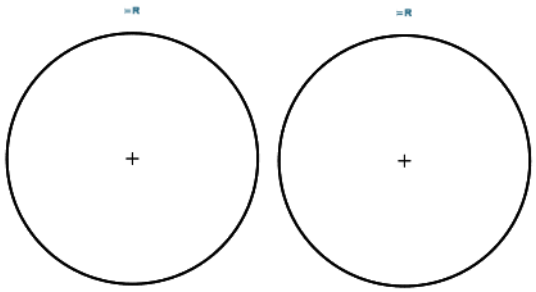
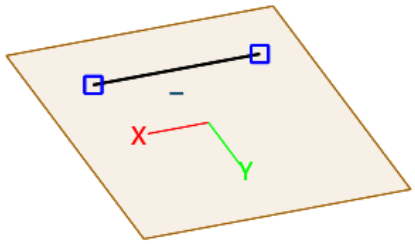
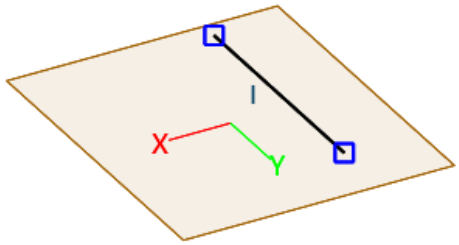
Sketch Environment >> Sketch >> Settings >> Relocate

3.1.25 ★3D Sketch Improvement

3.1.25.1 New Constraint Type

We added four new constraint types：Equal length, Equal Radius, Along Plane X, and Along Plane Y to ZW3D 2023.

Constraint	Effect
Equal Length	

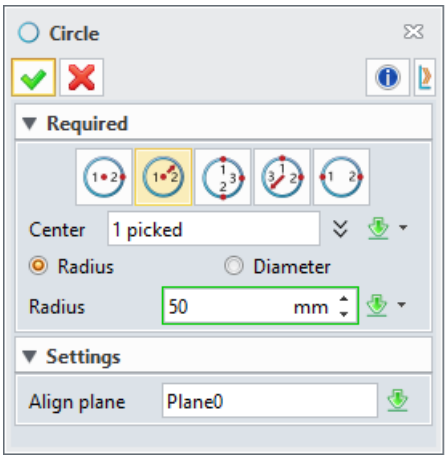
Equal Radius	
Along Plane X	
Along Plane Y	

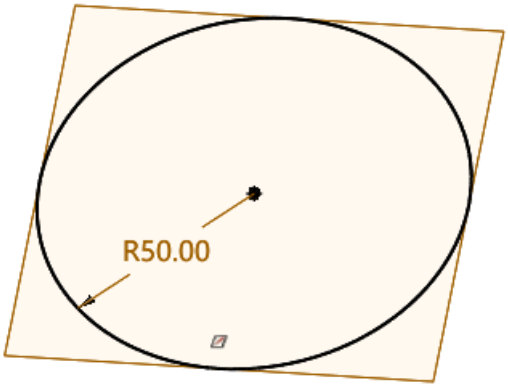
➔ **Where it is**

3D Sketch Environment >> Sketch >> Constraint

3.1.25.2 Support Automatically Creating Constraint

In the 3D sketch environment, when user utilizes “Align Plane” during creating lines, arc, circles, rectangles, polylines, ellipses, splines, and points, the system will automatically create the “**On Plane**” constraint.



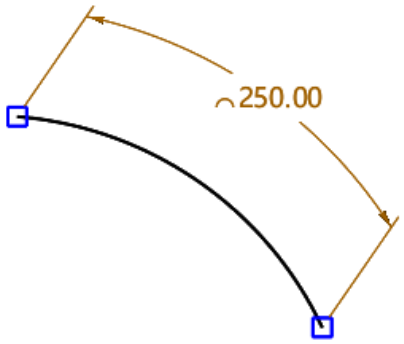
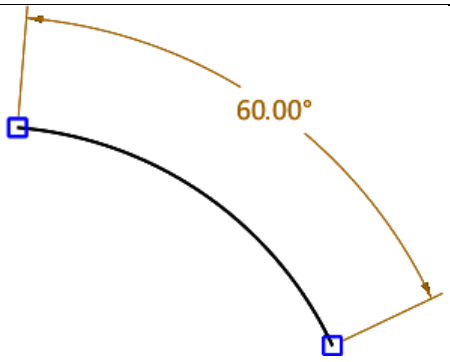


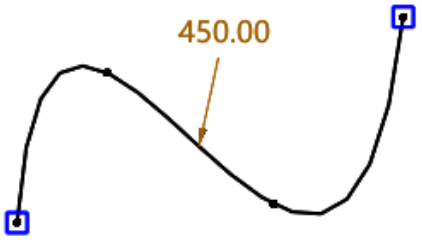
➔ **Where it is**

3D Sketch Environment >> Sketch >> Drawing

3.1.25.3 New Dim Modes

We added three new dimension modes: Arc length, Arc angular, and Curve length to ZW3D 2023.

Dimension Mode	Effect
Arc Length Dimension	
Arc Angular Dimension	

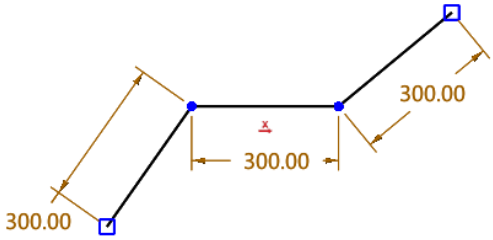
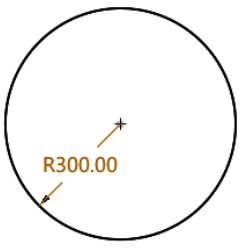
Curve Length Dimension	

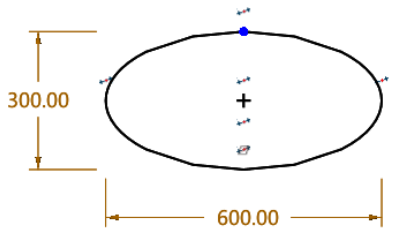
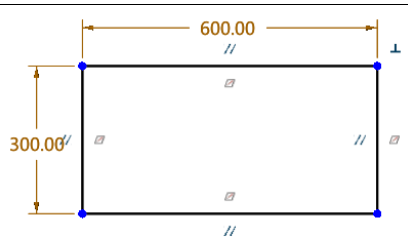
➔ **Where it is**

3D Sketch Environment >> Sketch >> Dimension

3.1.25.4 Support Automatically Creating Dimension

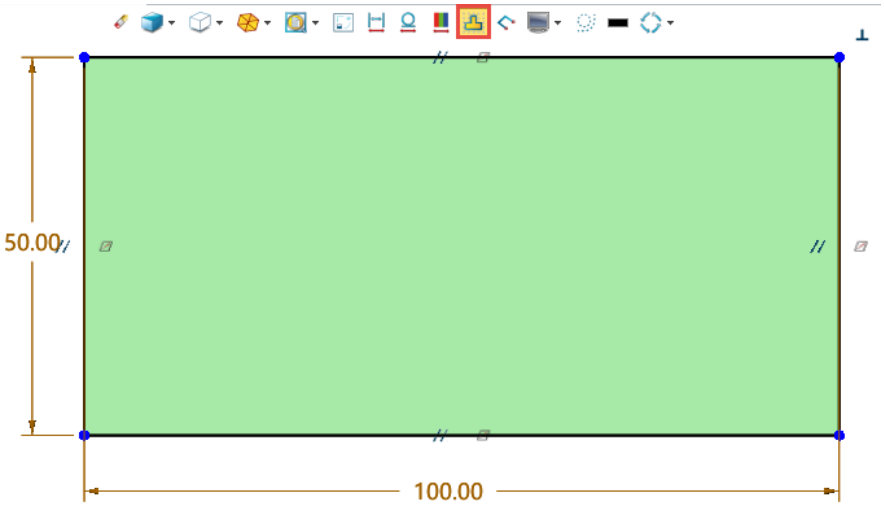
In the 3D sketch environment, the systema will automatically create corresponding dimensions during creating lines, polylines, arcs, circles, ellipses, and rectangles.

Type	Effect
Line	
Polyline	
Arc	
Circle	

Ellipse	
Rectangle	

3.1.25.5 New Closed Rings On/Off

We added “Closed Rings On/Off” to the 3D sketch environment in ZW3D 2023. Only when the closed rings locate on the same plane can the function be used.

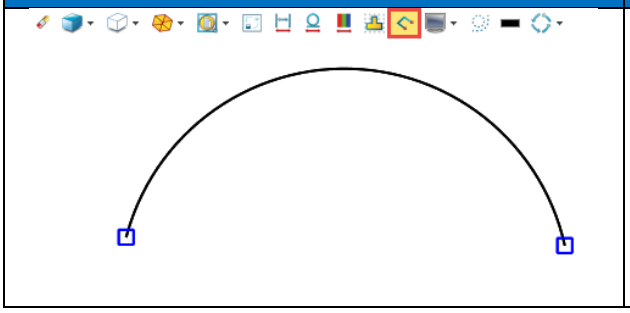
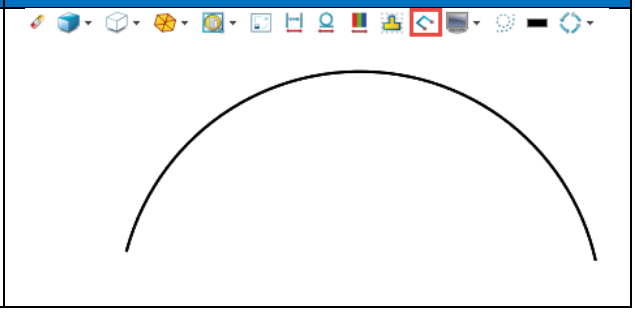


➔ Where it is

3D Sketch Environment >> DA Toolbar >> Closed Rings On/Off

3.1.25.6 New Display Open Ends On/Off

We added the “Display open ends on/off” to the 3D sketch environment in ZW3D 2023. Display open ends in the drawing area when the function turns on.

Display Open Ends (On)	Display Open Ends (Off)
	

➔ **Where it is**

3D Sketch Environment >> DA Toolbar >> Display open ends on/off

3.1.25.7 New Drag

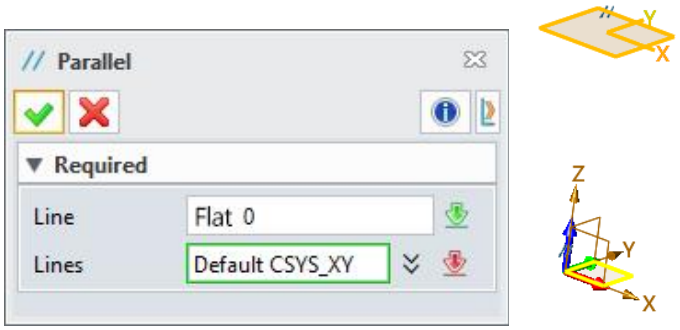
We added the "Drag" command to the 3D sketch environment, which can be used to drag the geometries in the drawing area.

➔ **Where it is**

3D Sketch Environment >> Sketch >> Basic Editing >> Drag

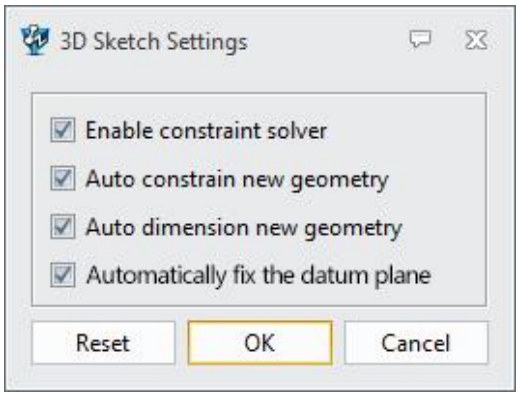
3.1.25.8 3D Sketch Datum Plane Participates in Constraint

The 3D sketch datum plane was brought into the 3D sketch constraint solver system. We implemented it as a driven element driven by the constraint system along with other 3D sketch objects.



The key points as below:

1. When "Auto fix datum plane" is unchecked in 3D Sketch Settings, the datum planes built in 3D sketch can be dragged freely.



2. The origin of 3D sketch datum plane can also be picked. If take the origin as key point (including endpoint, midpoint, circle center, spline point, control point, etc.) to deal with after picking the origin, and then it will take as the datum plane to deal with; if pick the face, it can only be dealt with as the datum plane.

3. We added the option “Auto fix datum plane” in 3D Sketch Settings panel which is checked to the old 3D sketch by default, so that the built-in datum plane can keep position unchanged when solving the old 3D sketch in the new version. Meanwhile, this option can also cover the unexpected situation that the built-in datum plane can be moved. The option is checked by default and the windows selection will not record the last state.

4. A lock icon will appear after adding fix constraint to a datum plane. But when the option “Auto fix datum plane” is checked, the datum plane without adding fix constraint will not display the lock icon.

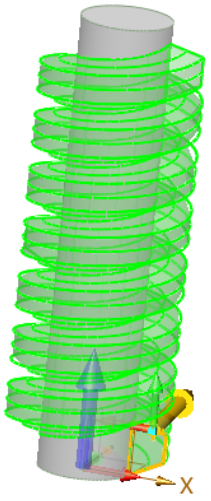
➔ Where it is

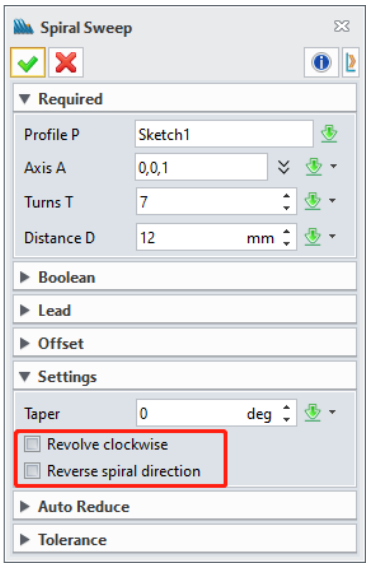
3D Sketch Environment >> Constraint

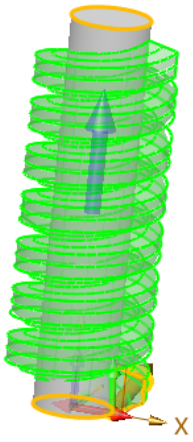
3.2 Shape Design

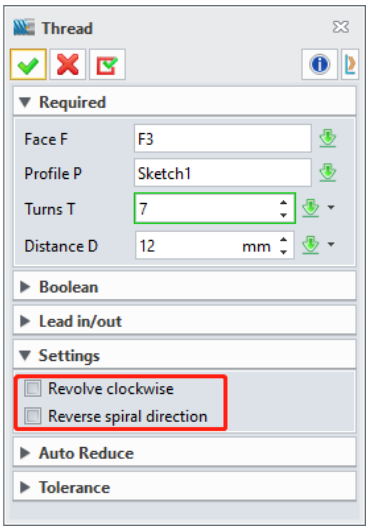
3.2.1 Spiral Sweep and Thread Improvement

Spiral Sweep and Thread both support clockwise and anti-clockwise settings.









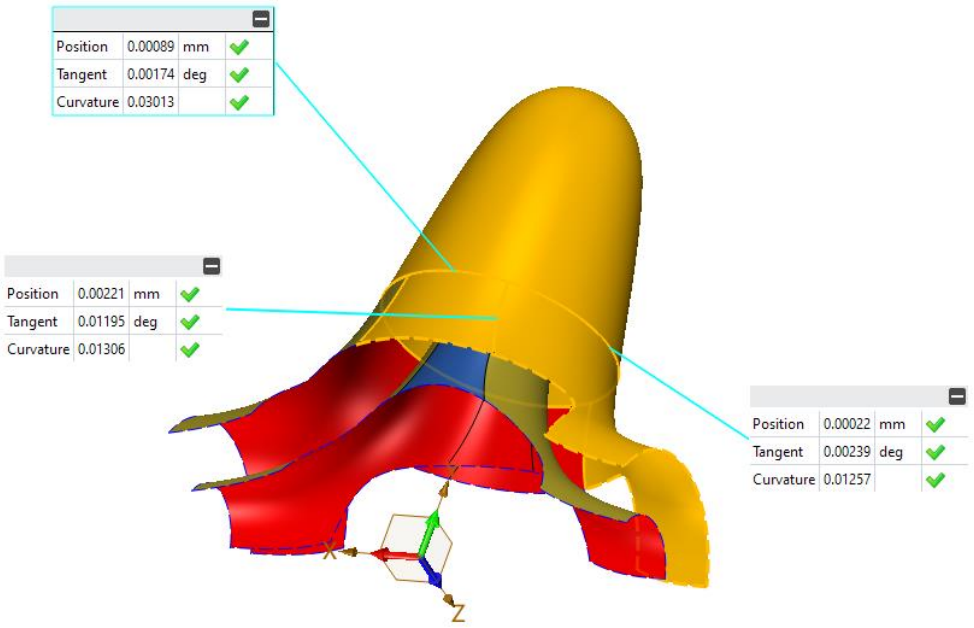
➔ Where it is

Part Environment >> Shape >> Basic Shape >> Spiral Sweep

Part Environment >> Shape >> Engineering Feature >> Thread

3.2.2 ★Face Offset Supports G2

Face Offset supports G1 and G2 continuity and keeps the offset results as G1 and G2 like the following figure shown.

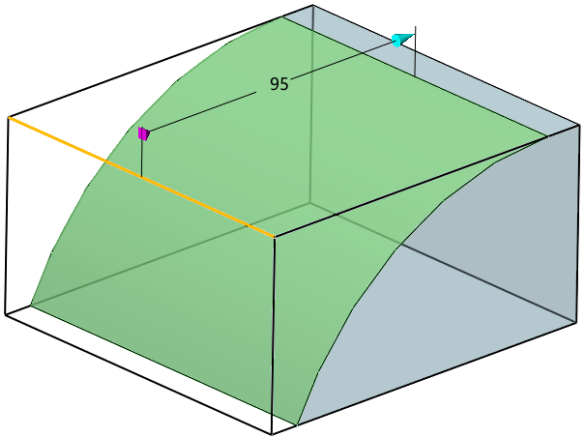


➔ **Where it is**

Part Environment >> Shape >> Edit Shape >> Face Offset

3.2.3 Fillet Improvement

There was a case that the adjacent face of the picked edge disappeared due to the increase of radius during filleting, which has been improved in this version. When user unchecks “Hold fillet to edge”, a fillet can also be successfully created when it exceeds an edge.

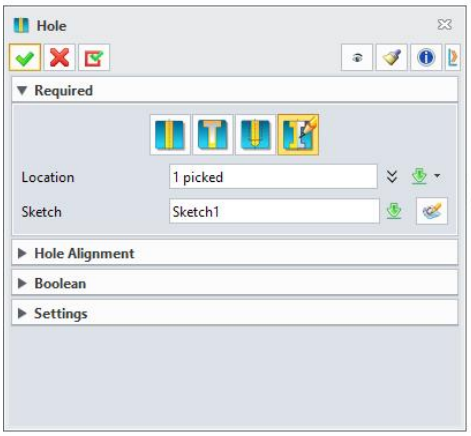


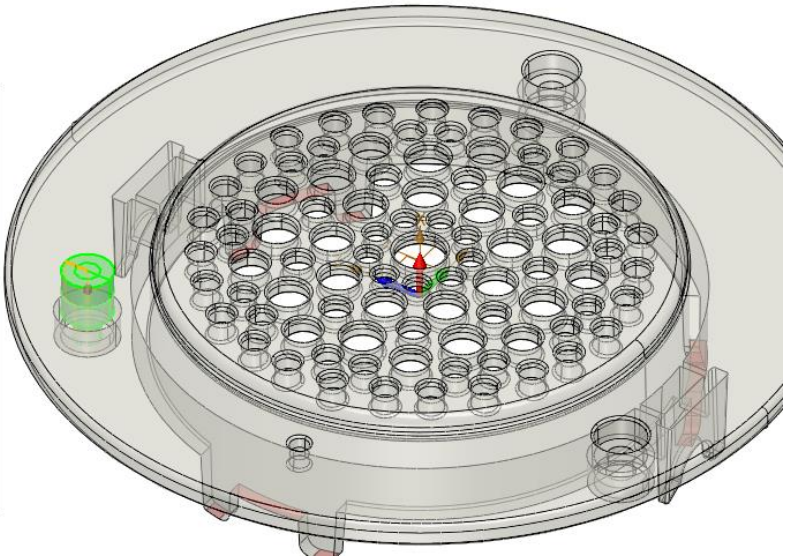
➔ **Where it is**

Part Environment >> Shape >> Engineering Feature >> Fillet

3.2.4 New Profile Hole

The new self-defined sketch profile hole can meet the actual mechanical design. User sometimes needs to create hole feature according to the custom profile, such as multiple counter-bore hole.



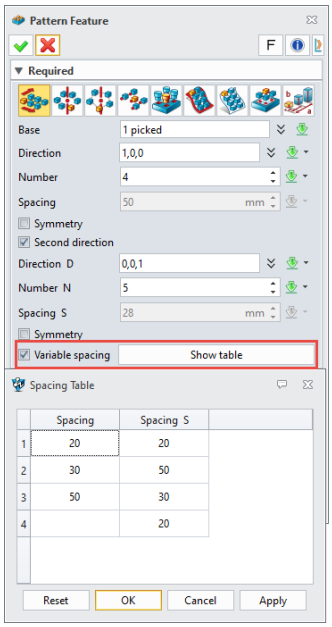


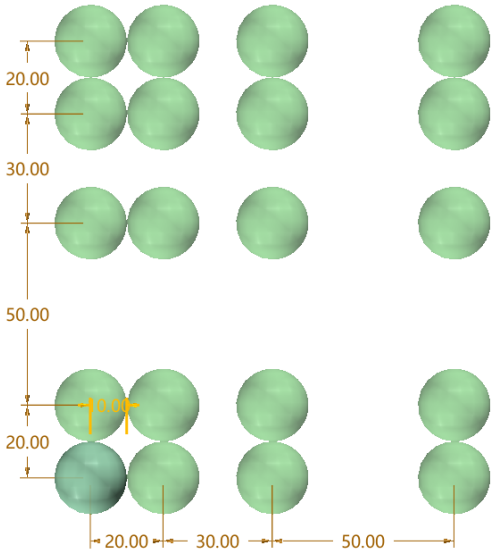
➔ **Where it is**

Part Environment >> Shape >> Hole

3.2.5 Pattern Improvement

We added “Variable spacing” to achieve pattern in unequal spacing which displayed pattern parameters as a table. It supports spacing in two directions and drives the variable spacing through parameters.



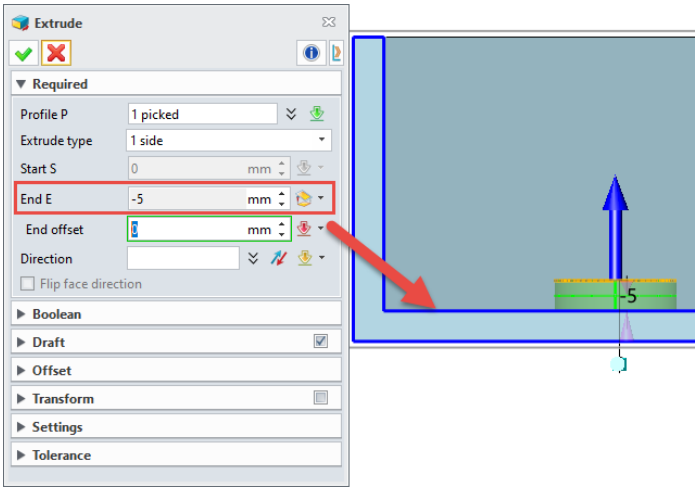


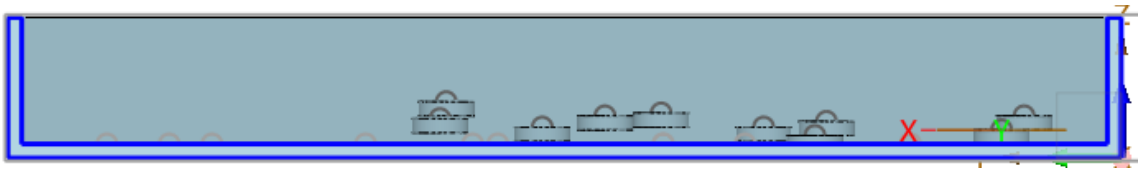
➔ **Where it is**

Part Environment >> Shape >> Basic Editing >>Pattern Geometry/Pattern Feature

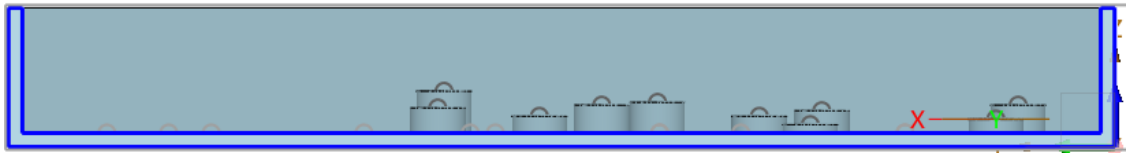
3.2.6 Extrude Point to Point Pattern Improvement

In ZW3D 2023, when extrude features to a face, all features of pattern can be kept to the extruded face in the point to point pattern.





Before Improvement



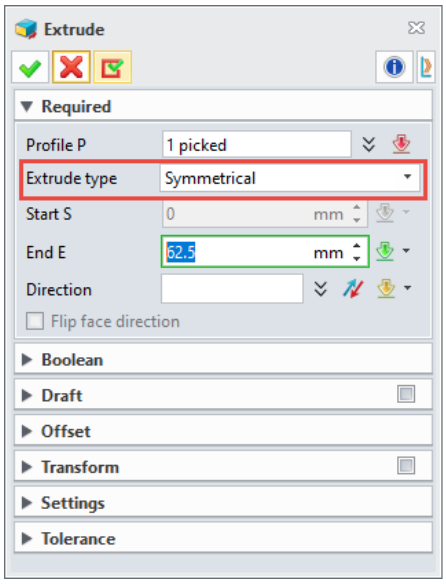
After Improvement

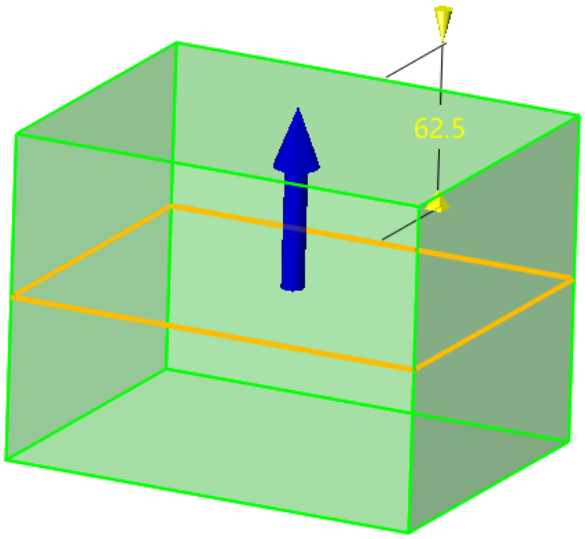
➔ **Where it is**

Part Environment >> Basic Editing >> Pattern Feature >> Point to point

3.2.7 Extrude Improvement

In product design, the total length of product is usually first to know. We added “Total symmetrical” to input the total length in the extrude type.





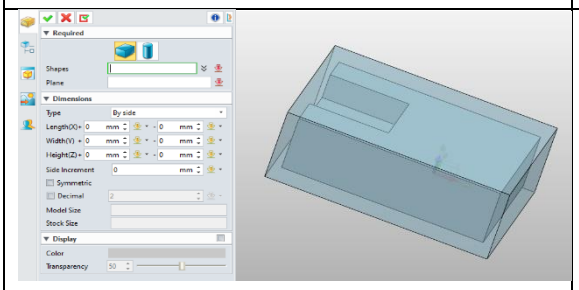
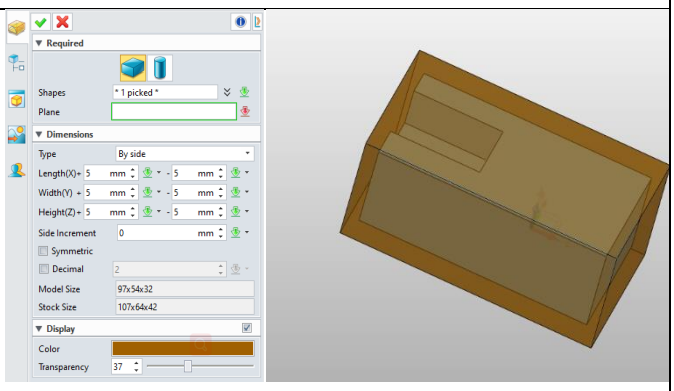
➔ **Where it is**

Part Environment >> Shape >> Basic Shape >> Extrude

3.2.8 Support Display Stock Color and Transparency

We added the function of Display to adjust the stock color and transparency through this option.

If the option is unchecked, the color and transparency are not able to edit; if the option is checked, user can adjust the stock color and transparency according to realistic need.

By Default	Custom Display
	

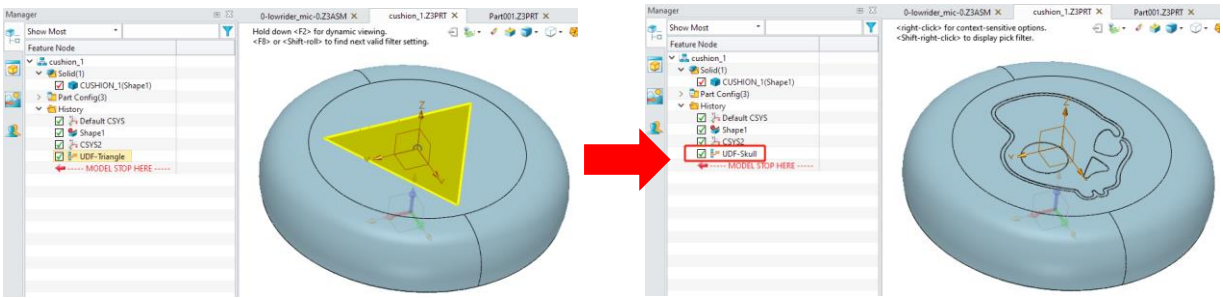
➔ **Where it is**

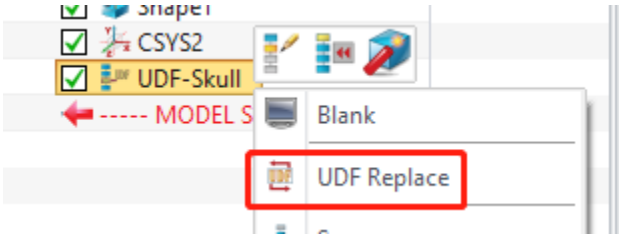
Part Environment >> Shape >> Engineering Feature >> Stock

3.2.9 UDF Improvement

When designers are designing parts, they often need to repeat some features, such as screw holes and ladder holes on box parts, keyway of shaft parts, LOGO on cover parts, etc. Although these features are not complicated, it takes lots of time and energy to repeat.

We added “UDF Replace” in ZW3D 2023, which can replace “UDF”. But the new “UDF” reference type and quantity must be the same as the original one.



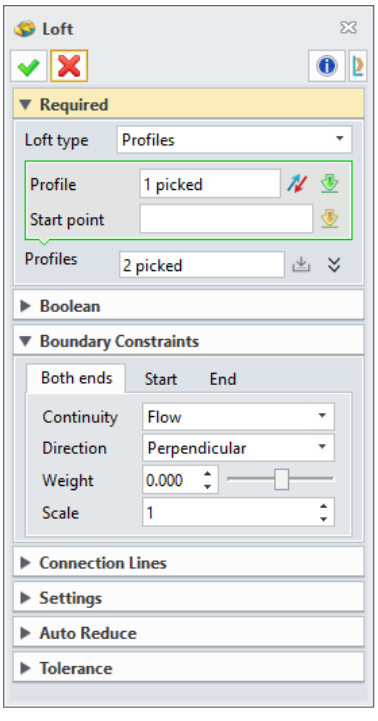


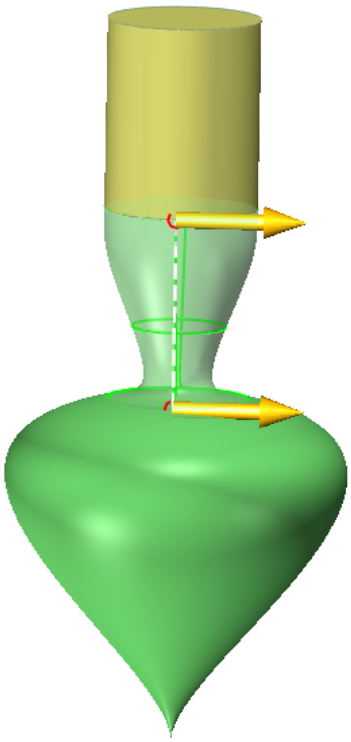
➔ **Where it is**

Context Menu >> UDF Replace

3.2.10 ★ Loft Improvement

We added a fourth continuity method “Flow” in the boundary constraint of Loft command. The boundary constraint of “Flow” has achieved the continuity of G3, which means two lofted faces share common edges.





➔ **Where it is**

Part Environment >> Basic Shape >> Loft >> Boundary Constraints >> Continuity >> Flow

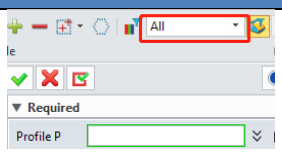
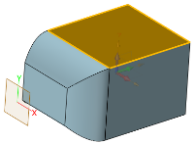
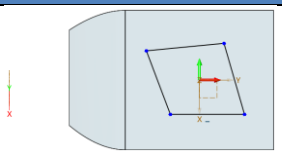
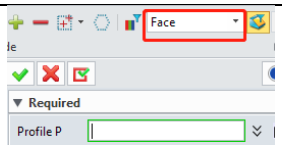
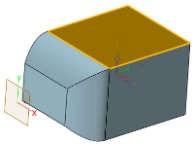
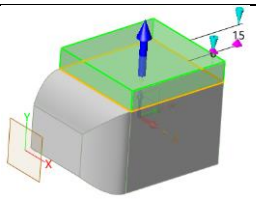
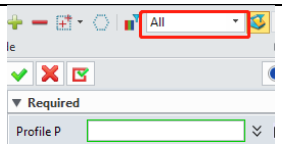
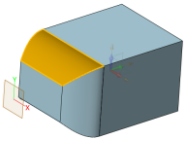
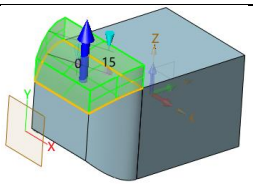
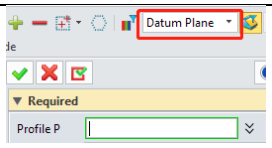
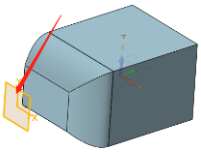
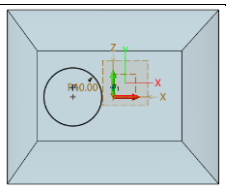
3.2.11 Extrude/Revolve Workflow Improvement

We simplified the steps to enter the built-in sketch in **Extrude/Revolve** commands. User can directly enter the sketch environment by selecting a shape surface/datum plane or directly extrude the selected surface, which will be dependent on the selection of filter of "**Profile P**" in the command.

- When the filter is set as "All" or "Datum Plane", user can directly enter the sketch environment by selecting the plane.
- When the filter is set as "Face", user can extrude/revolve the face after selecting.

When the filter is set as "All" and surface (non-plane) is selected, user can extrude/revolve the surface.

Here take the extrude feature as examples.

Attribute Filter	Profile P	Result
		
		
		
		

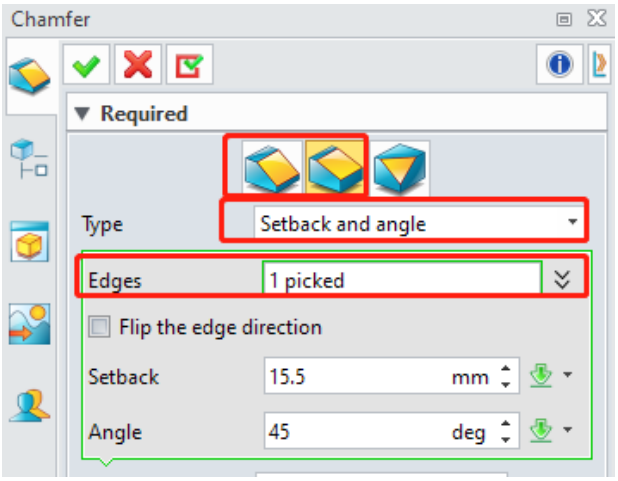
➔ Where it is

Part Environment >> Shape >> Basic Shape >> Extrude

Part Environment >> Shape >> Basic Shape >> Revolve

3.2.12 New Memory Input Object in Chamfer

After activating the command, user often directly selects chamfer edges and selects the chamfer type. In previous version, the system will automatically clear the objects selected earlier, which leads to user's repeat selecting. To improve efficiency, we added a function of memory input object in ZW3D 2023, which can remember the earlier input object when user is switching the chamfer and unsymmetric chamfer.

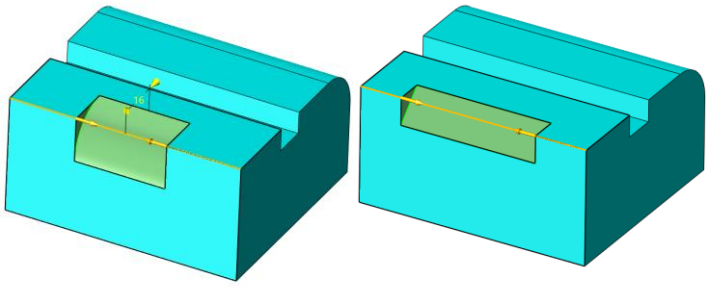


➔ **Where it is**

Part Environment >> Shape >> Engineering Feature >> Chamfer

3.2.13 New Stop Short of Corner in Fillet/Chamfer

We added the "**Stop Short of Corner**" to the "**Fillet/Chamfer**" commands, which can control stopping on some point of the edge in fillet/chamfer so to reach the purpose of local fillet/chamfer.

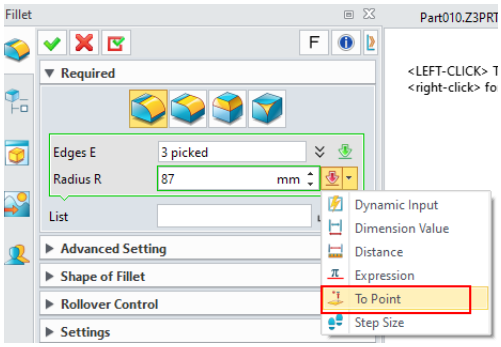


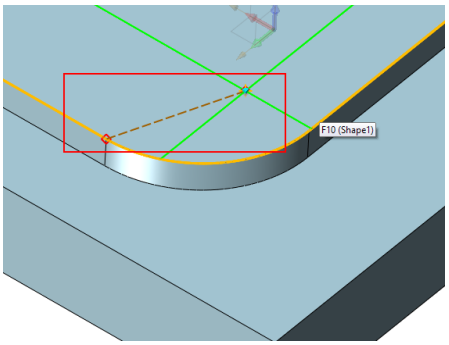
➔ **Where it is**

Part Environment >> Shape >> Engineering Feature >> Fillet/Chamfer

3.2.14 New Dotted Line in To Point

As for Radius/Setback settings in “Fillet/Chamfer” commands, user can set the length through “To Point” in the drop-down list. We added the dotted line to represent the length “To Point” in ZW3D 2023.



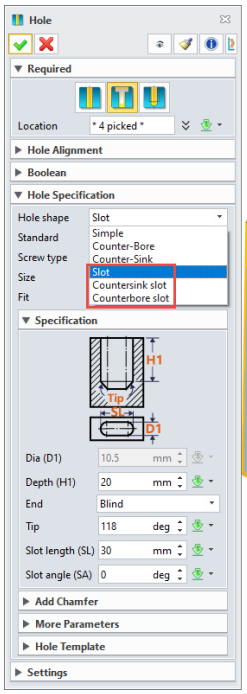


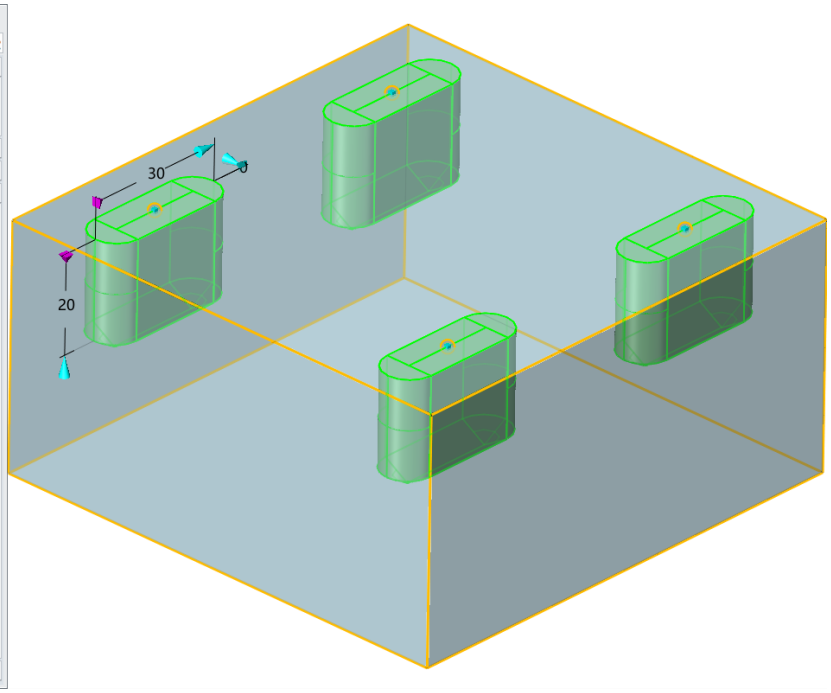
➔ **Where it is**

Part Environment >> Shape >> Engineering Feature >> Fillet/Chamfer >> Radius/Setback Drop-Drown List >> To Point

3.2.15 New Slot Hole

We added three new hole shapes: **Slot**, **Countersink slot**, and **Counterbore slot** to the “Hole” command in ZW3D 2023.





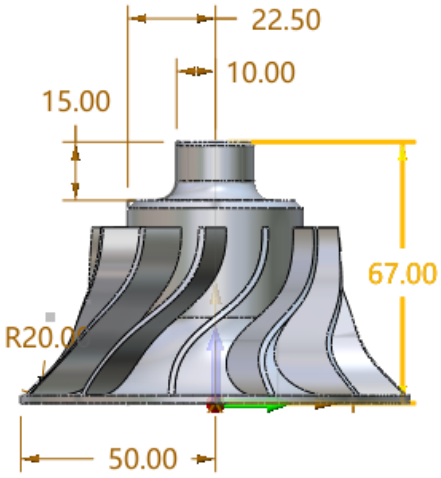
➔ **Where it is**

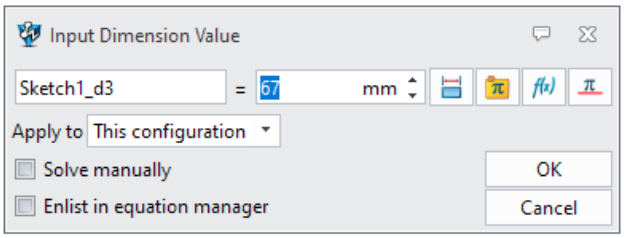
Part Environment >> Shape >> Engineering Feature >> Hole

Assembly Environment >> Assembly >> Basic Editing >> Hole

3.2.16 Show Dimensions Improvement

In the part environment, the internal sketch dimension can be displayed when user enables “Show Dimensions” onto features in ZW3D 2023. When the internal sketch parameter dimensions are showed, user can double-click the dimensions to modify the internal sketch dimensions and quickly modify the model geometry.





➔ **Where it is**

Part Environment >> Feature/Sketch Context Menu >> Show Dimensions

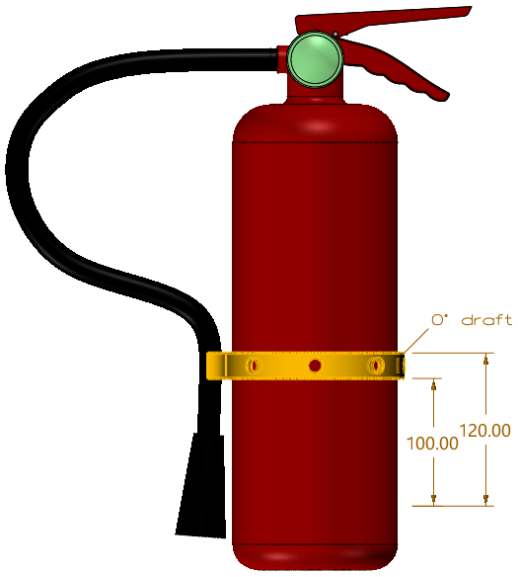
3.2.17 Dimensions On/Off Improvement

We optimized Dimensions On/Off to assist user better controlling the feature parameters in need.

1) When user chooses the features before the Dimensions On/Off, the function can only show/hide the parameter dimensions of corresponding features, which means to support selecting feature first and then enable command.

2) When user does not choose any feature and enable the Dimensions On/Off, the default show/hide all features dimensions.

3) When user selects an entity object with the Dimensions On/Off, then show/hide the corresponding its parameters.

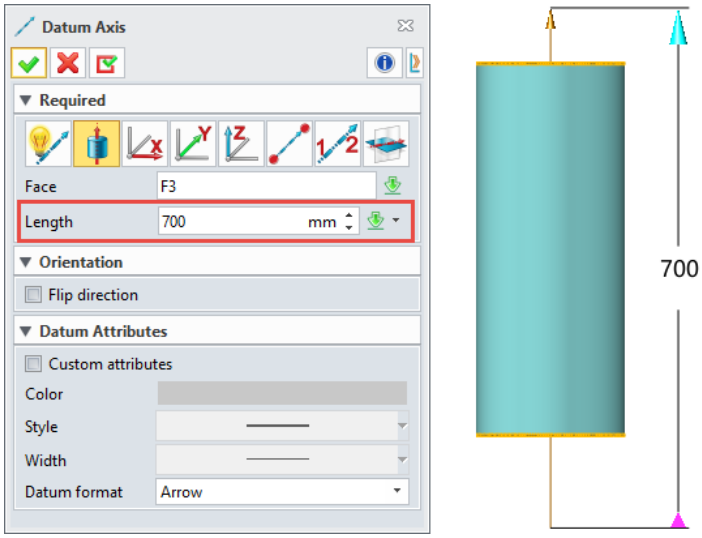


➔ **Where it is**

Part Environment >> DA Toolbar >> Show/Hide Dimensions

3.2.18 Datum Axis Improvement

In Datum Axis, we added a new length setting in mode of face center axis.

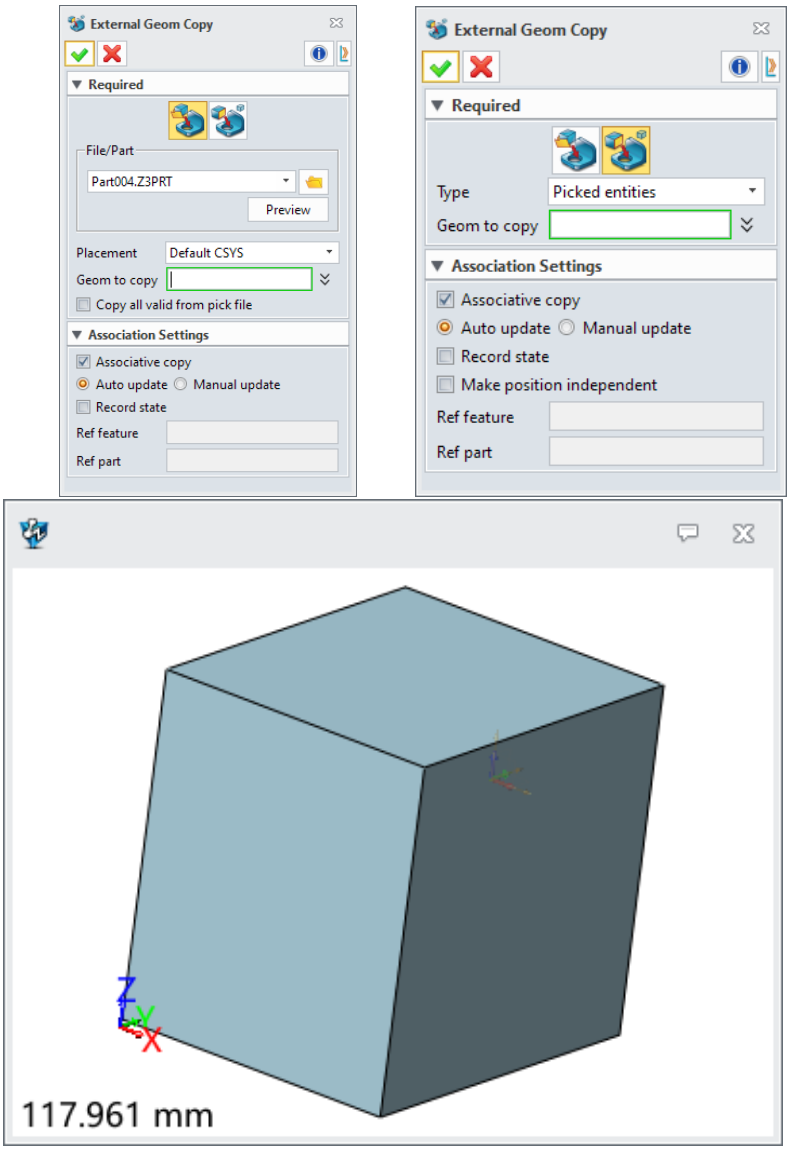


➔ **Where it is**

Part Environment >> Shape >> Datum >> Datum Axis

3.2.19 ★New External Geom Copy

External Geom Copy is an important top-down design tool, which can be used to pass design standard on another. User can copy reference geometry to pass information and remain association between models through the tool. Considering that some functions are missing, and the command is scattered in the geometry reference of the original system, we added External Geom Copy which supports small window.



1) Support referring to the geometry objects of external parts such as points, lines, edges, sketches, faces, entities, datum axes, datum planes and datum CSYS. Meanwhile, user can locate the external geometry and target object by Default Datums and Select Datums.

2) Support referring to geometry object of other components in the assembly tree.

3) Support the disconnection and association of reference object as well as the manual and automatic update.

4) Support the widget picking the reference object.

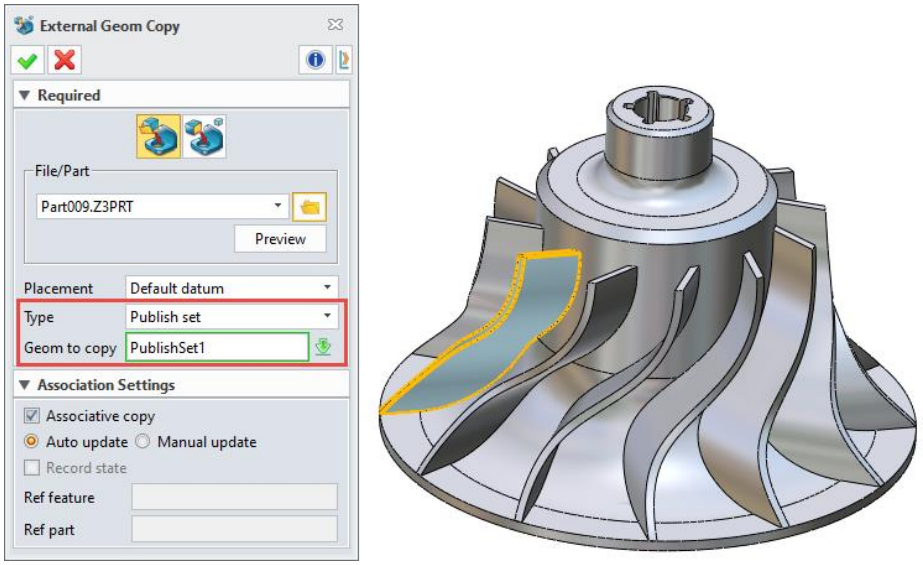
5) Support referring to the feature with time stamp.

➔ **Where it is**

Part Environment >> Data Exchange >> External Geom Copy

3.2.20 New Publish Set

We added “Publish Set” in ZW3D 2023 to help user pick the geometric objects to be published. A publish set folder will be generated in the history tree and the “Publish Set” feature will be listed in the folder. User can refer the published geometry objects of other parts to the current file through the command “External Geom Copy”. When the original published geometry object is updated, the referenced published objects are updated simultaneously.

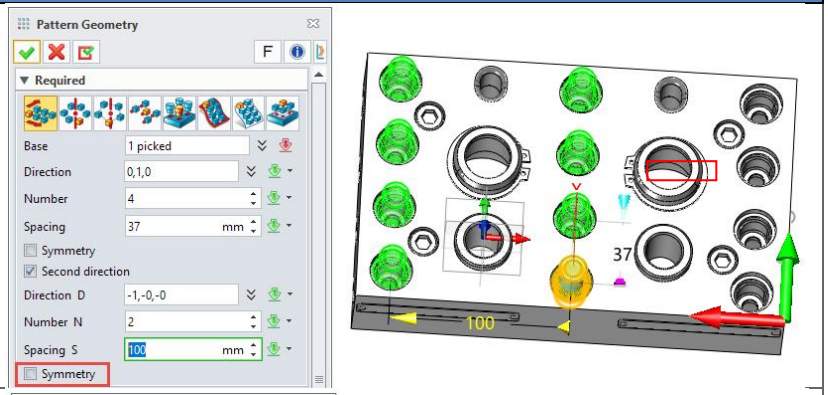
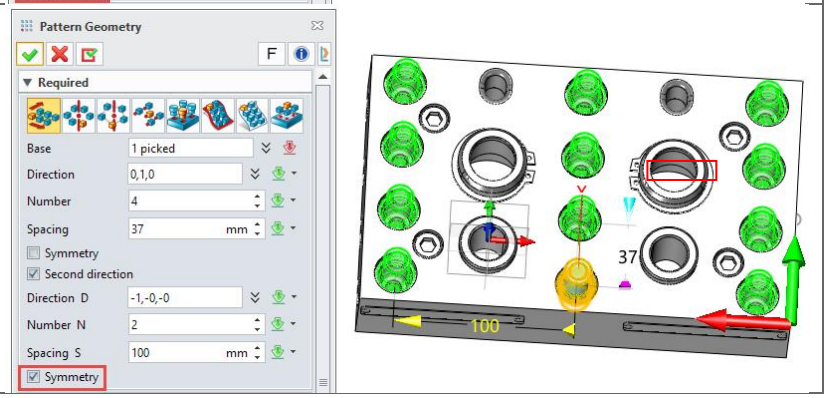


➔ **Where it is**

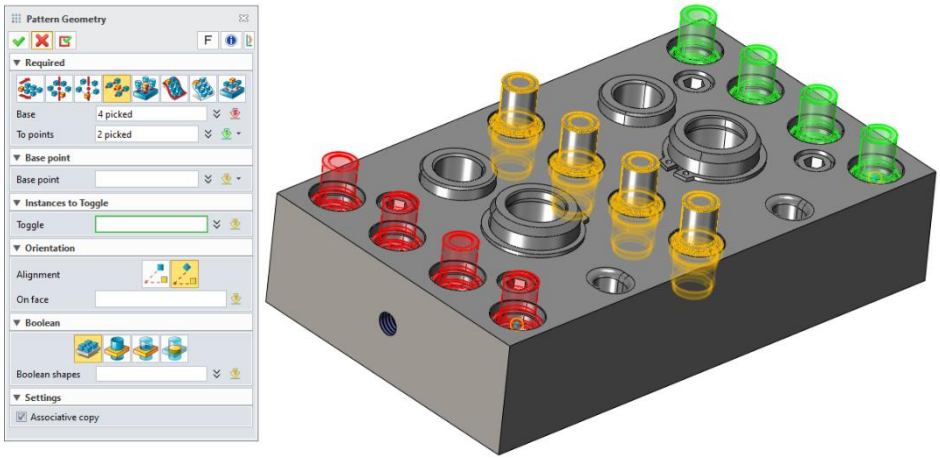
Part Environment >> Data Exchange >> Export >> Publish Set

3.2.21 Pattern Improvement

(1)The option “Symmetry” is added to the command “Pattern Geometry/Pattern Feature”, which supports geometry creating pattern in the specified direction and creating pattern object symmetrically in the opposite direction based on the geometry position.

Option	Status	Sample
Symmetry	Unchecked	
	Checked	

(2)We added “Instance to Toggle” to “Point to Point” to “**Pattern Geometry/Pattern Feature**”. “Instance to Toggle” supports opening and closing instance in the pattern. According to instance opening or closing, the response mode will display instance in a red dotted line box. The function in this command is like that in other modes.

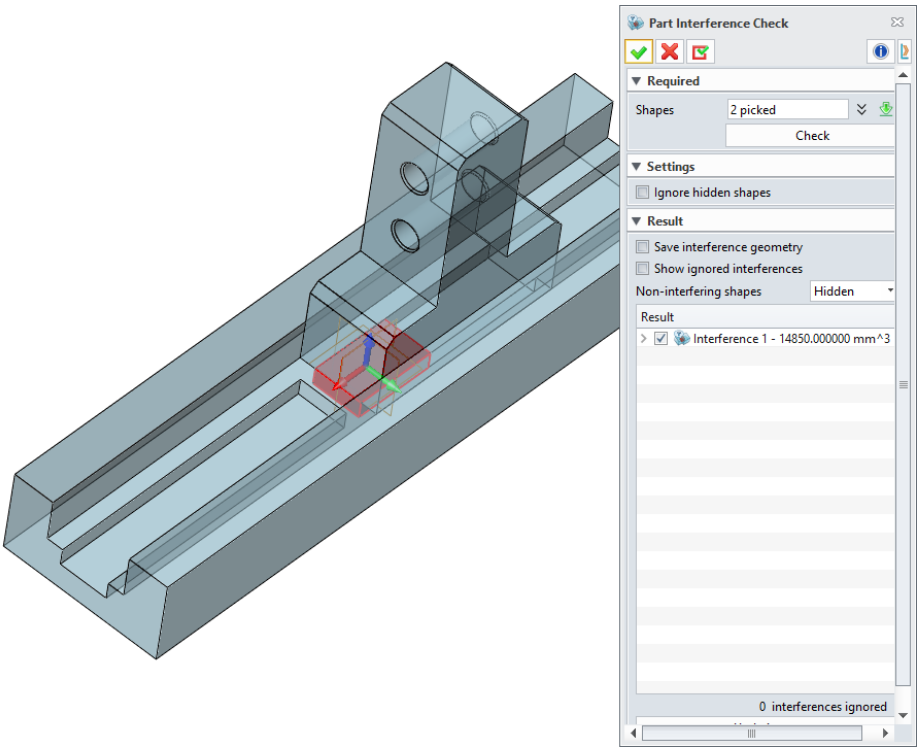


➔ **Where it is**

Part Environment >> Basic Editing >> Pattern Geometry/Pattern Feature

3.2.22 New Interference Check

To clearly understand the interference between shapes, user can use this command to check the interference status between parts.



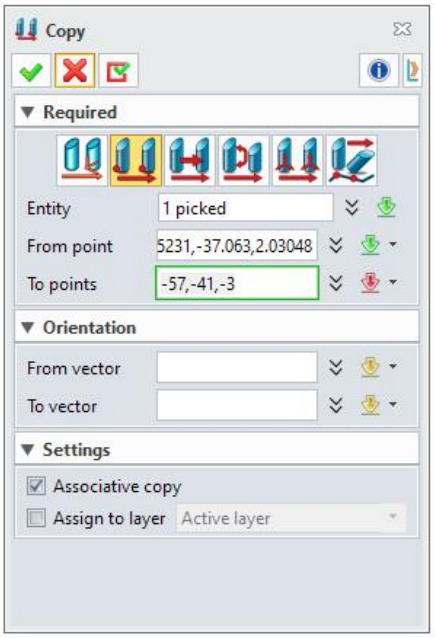
➔ Where it is

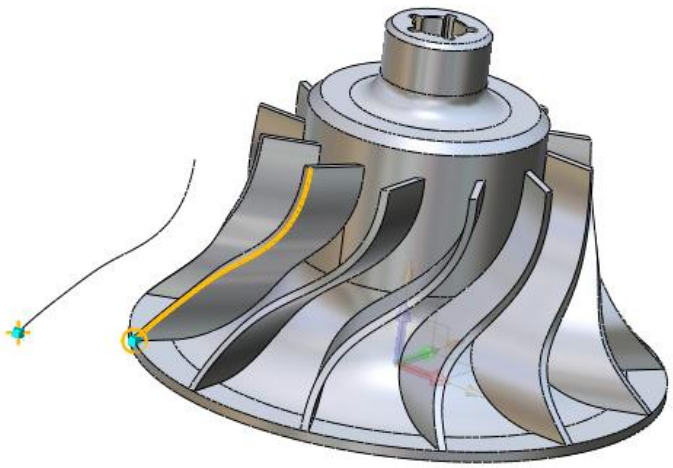
Part Environment >> Inquire >> Inspect Model >> Part Interference Check

3.2.23 New Copy Edge

The “Copy” command can support copying multiple entities including shapes, surfaces, sketches, datum planes, curves, points, and texts, etc. Copying edges are supported by ZW3D 2023.

In addition to copy edge through “Copy” command, it can be copied and pasted through shortcut keys Ctrl+C/Ctrl+V.





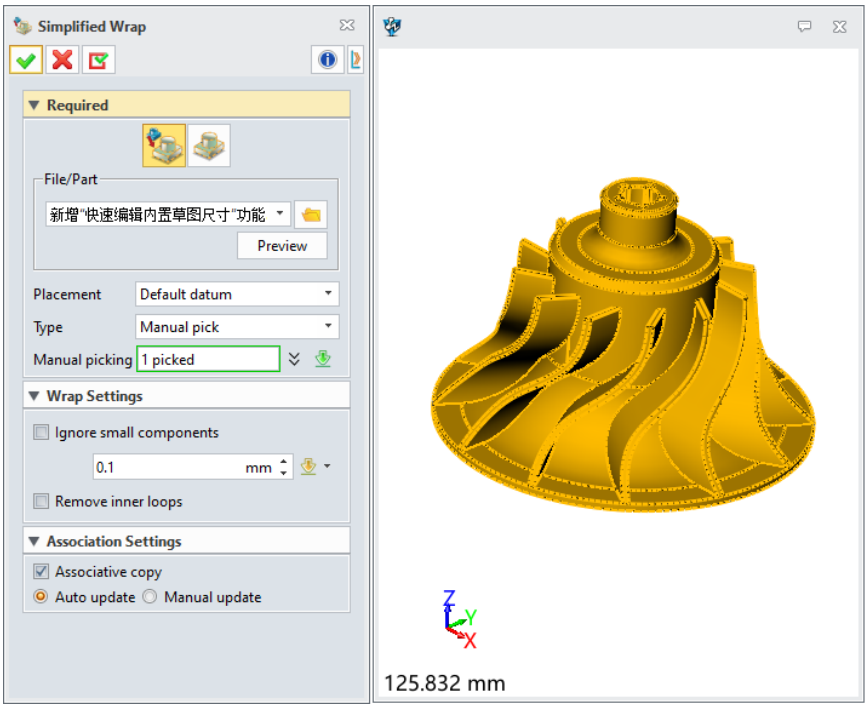
➔ **Where it is**

Part Environment >> Shape >> Basic Editing >> Copy

3.2.24 New Simplified Wrap

Use the “Simplified Wrap” command to extract the outmost profiles and copy them to the current files. The command supports extracting shapes from external and local parts.

- Support extracting wrap shapes from the small windows.



- Support ignoring small components and defining the size of small components to ignore.
- Support moving inner rings.
- Support associative copy. A local “Simplified Wrap” feature after copying will be generated.

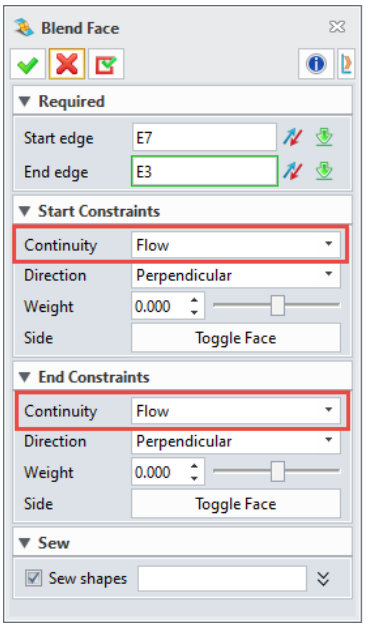
➔ Where it is

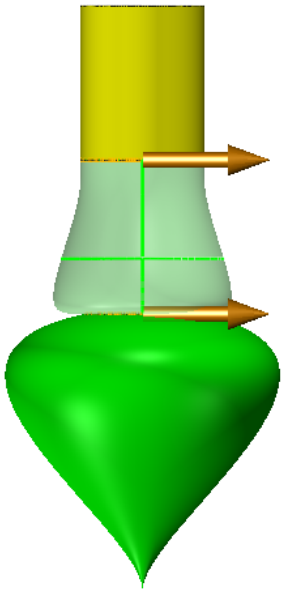
Part Environment >> Data Exchange >> Import >> Simplified Wrap

3.3 Free Form Design

3.3.1★Blend Face Improvement

We added a fourth continuity method of “Flow” in Start/End Constraints of Blend Face. The boundary constraint “Flow” realizes the continuity of G3, which means two blend faces share the common edges.



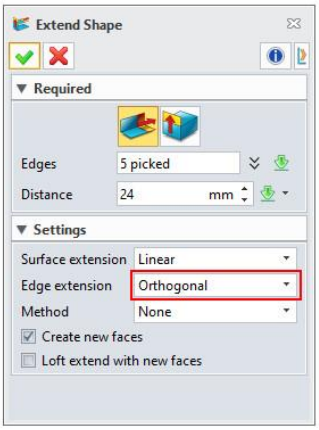


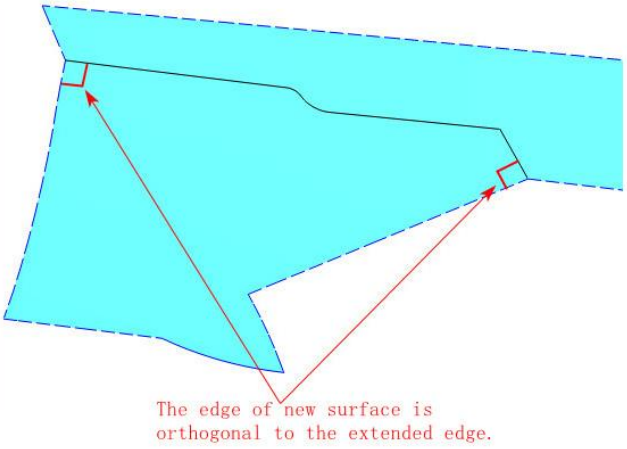
➔ **Where it is**

Part Environment >> Free Form >> Basic Face >> Blend Face >> Start Constraints >> Continuity >> Flow

3.3.2 Extend Shape Improvement

We added “Orthogonal” method to the command “External Shape”. With this method, the boundary and the extended edge of the extension surface will form perpendicular.





➔ **Where it is**

Part Environment >> Free Form >> Edit Face>> Extend Shape

3.3.3 New Disjoint Entities as New U/V

We added “Disjoint entities as new U/V” in the command “Curve Mesh” in ZW3D 2023. When it is checked, as Figure A shows, support user selecting all U curves during inputting U curves at one time. The system will automatically identify as disjointed curves and take them as new U curves adding to the curve list. When it is unchecked, as Figure B shows, user can select multiple disjointed curves at one time and keep error message popup. The application of V curves is similar.

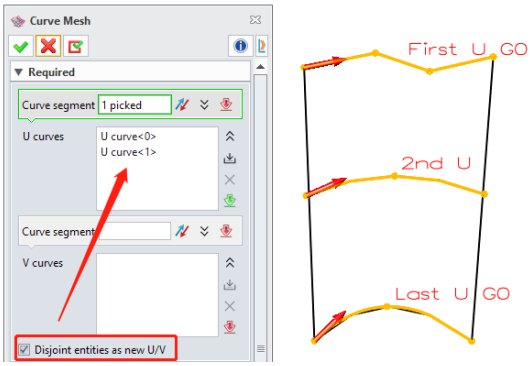


Figure A Check “Disjoint entities as new U/V”

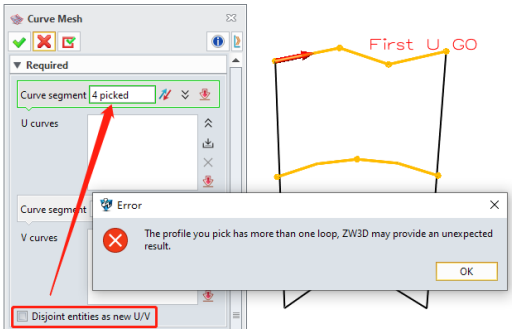


Figure B Uncheck “Disjoint entities as new U/V”

➔ Where it is

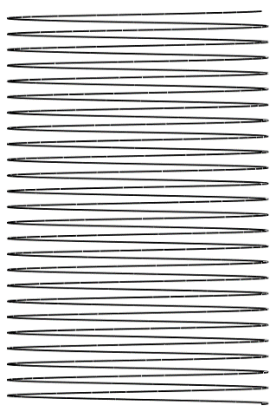
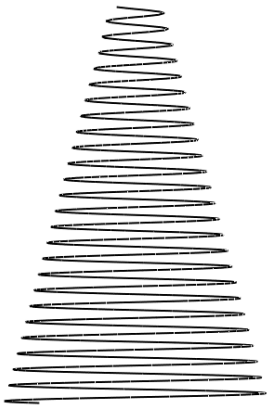
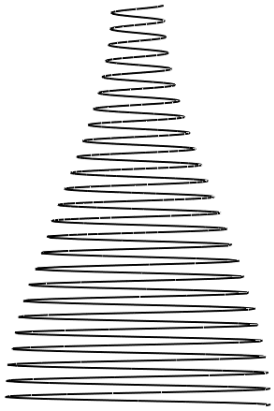
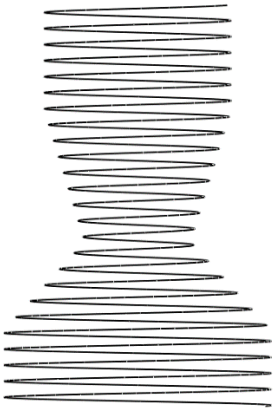
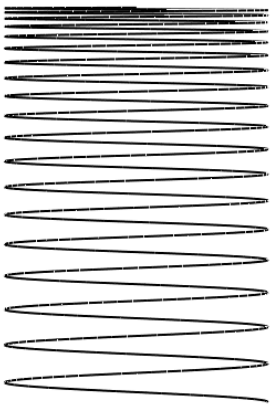
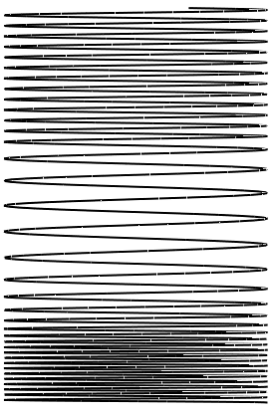
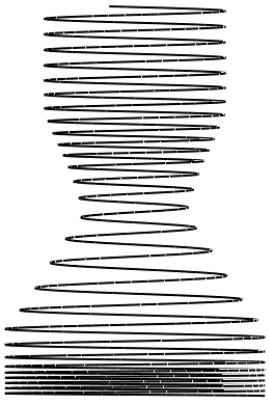
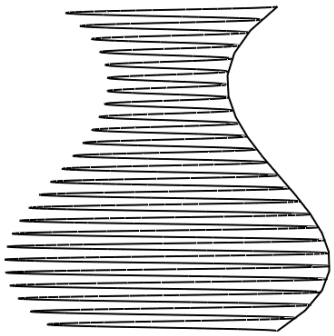
Part Environment >> Free Form >> Basic Face>> Curve Mesh

3.4 Wireframe Design

3.4.1 ★Spiral Helix Improvement

ZW3D 2023 optimizes the way of creating spiral helix, supports variable radius and variable pitch adjustment, and supports any two of the number of turns, pitch, and length as variables to control the shape of spiral helix. New support for creating spirals as law curves.

Constant radius	Linear radius	3 degree radius	Custom radius

			
Linear pitch	**Custom pitch**	**Variable radius and pitch**	**Law curves**
			

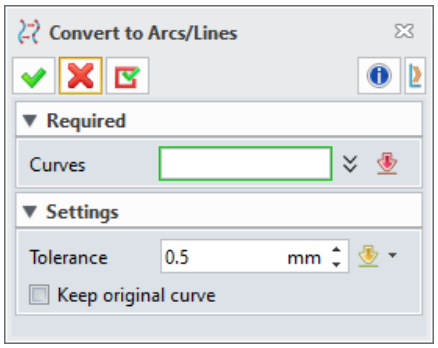
➔ **Where it is**

Part Environment >> Wireframe >> Curve >> Spiral Helix

3D Sketch Environment >> Sketch >> Curve >> Spiral Helix

3.4.2 New Keep Original Curve in Curve to Arcs/Lines

In Wireframe/Sketch environment, we added an option “Keep original curve” to the command Covert to Arcs/Lines. When a spline curve is converted to an arc/line, if this option checked, the original curve can be kept; if unchecked, the original curve will not be kept. Default the option is unchecked.

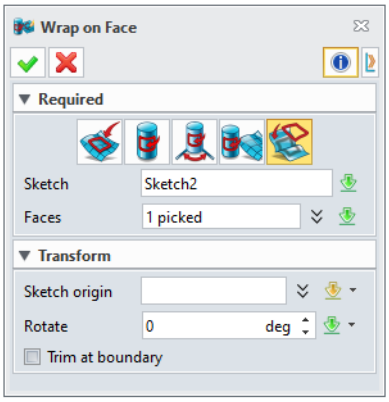


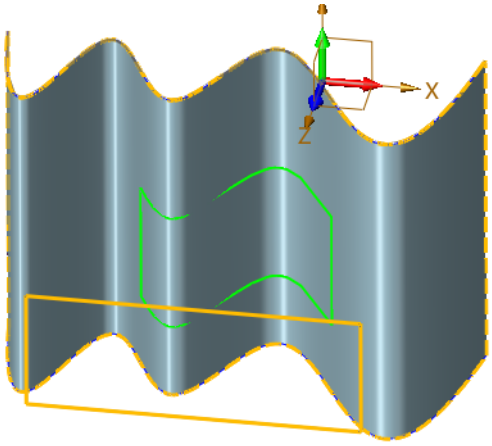
➔ **Where it is**

Wireframe/Sketch Environment >> Edit Curve >> Convert to Arcs/Lines

3.4.3 New Wrap Curve on Developable Face

We added the method "**Wrap curve on developable face**" in the command "**Wrap on Face**", which can map the sketch profile to the developable face in equal length. When the option "**Trim at boundary**" is checked, user can trim the wrap curves that exceed the range of developable face.



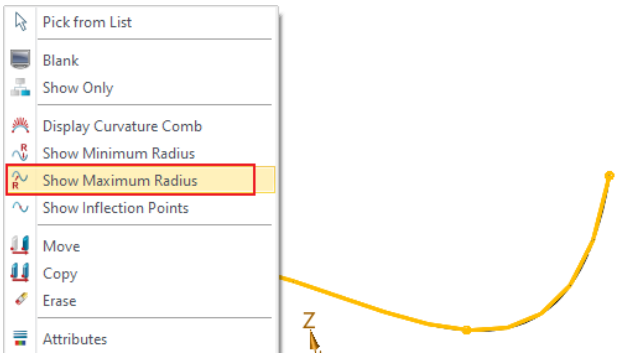


➔ **Where it is**

Part Environment >> Wireframe >> Curve>> Wrap on Face

3.4.4 Show Maximum Radius Improvement

In the part environment, user often needs to search the maximum radius of curves. Therefore, we added the function "Show Maximum Radius" in the context menu of curve in ZW3D 2023.



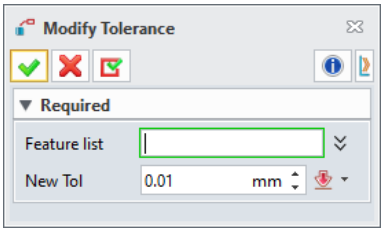
➔ **Where it is**

Part Environment >> Select Curve >> Context Menu>> Display Maximum Radius

3.5 History Feature

3.5.1 Modify Tolerance Improvement

Modify Tolerance is changed to a command with UI panel. Feature list is used to select the feature that needs modifying while the default value in New Tol is the tolerance of the current file. User can modify the existing tolerance by setting New Tol through this command.

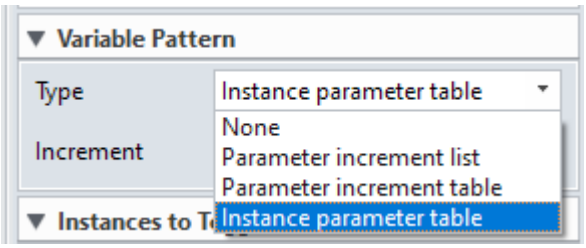


➔ **Where it is**

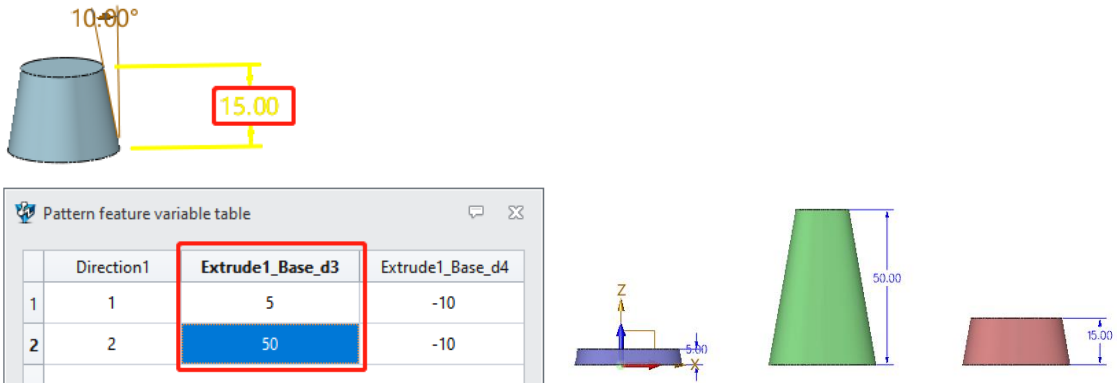
Part Environment >> Edit >> History Operation >> Modify Tolerance

3.5.2★ Pattern Feature

(1) In Variable Pattern of Pattern Feature, the type **“Parameter list”** is renamed as “**Parameter increment list**”, the type “**Parameter table**” is renamed as **“Parameter increment table”**, and a new type “**Instance parameter table**” is added.

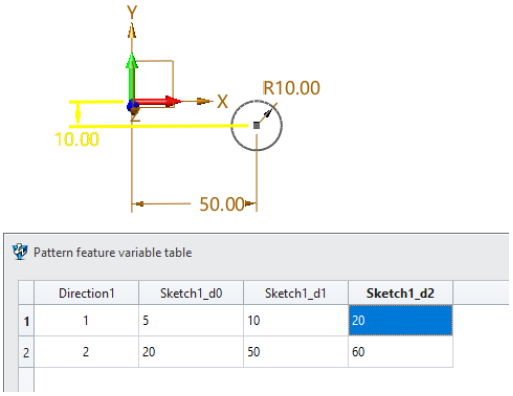


"**Instance parameter table**" supports direct defining of all parameters of each feature.

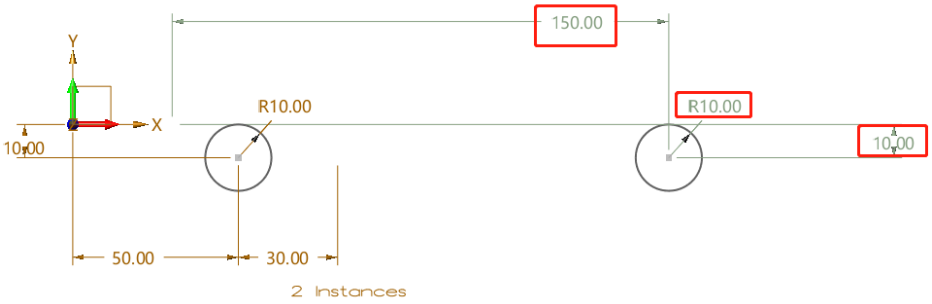


(2) Pattern feature supports selecting sketch, by which user can select whether to pattern the sketch as request.

(3) Support changing parameters of the sketch dimensions of each type of variable pattern in the feature of pattern. When user is selecting some parameter variable in the parameter table, the corresponding original dimension in the view will be highlighted.



(4) Support modifying the displayed dimension in pattern sketch.

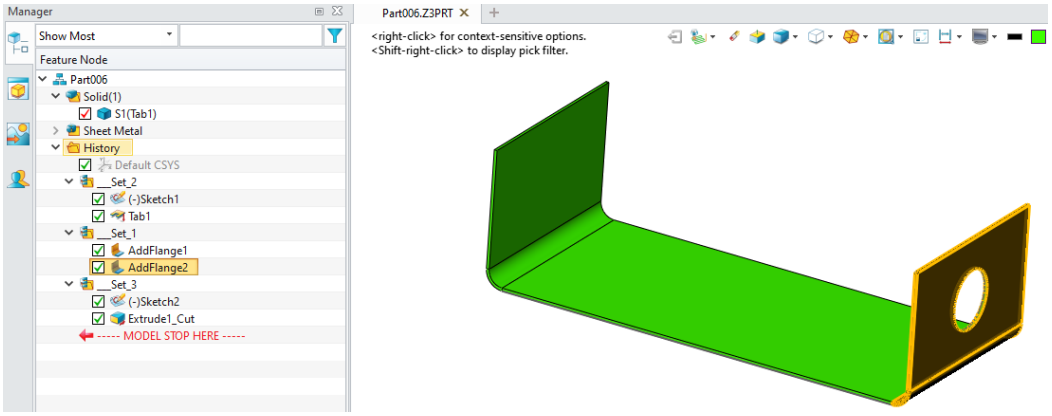


➔ **Where it is**

Part Environment >> Shape/Free Form/Wireframe/Weldments >> Basic Editing >> Pattern Feature

3.5.3 Location History Node Improvement

When the mouse is selecting the features, the history manager will automatically jump to the nodes and highlight them. If there exists a history feature folder, the folding folder will be automatically unfolded and highlight the history feature node.

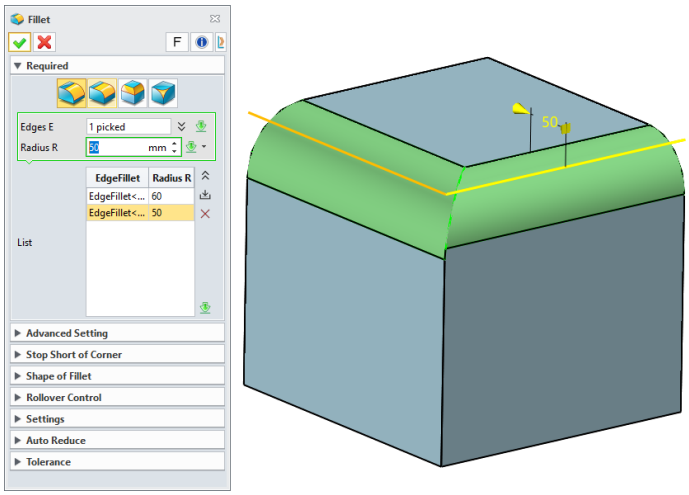


3.5.4★ UDF Improvement

3.5.4.1 UDF Supports List

If a feature uses “List”, when it is added to UDF, the “List” will be unfolded according to the selected number and the inputs of sub-commands. Each item is an independent input of UDF, so user can configurate independently.

As the figure shows, it contains two groups fillet edges of chamfer feature.



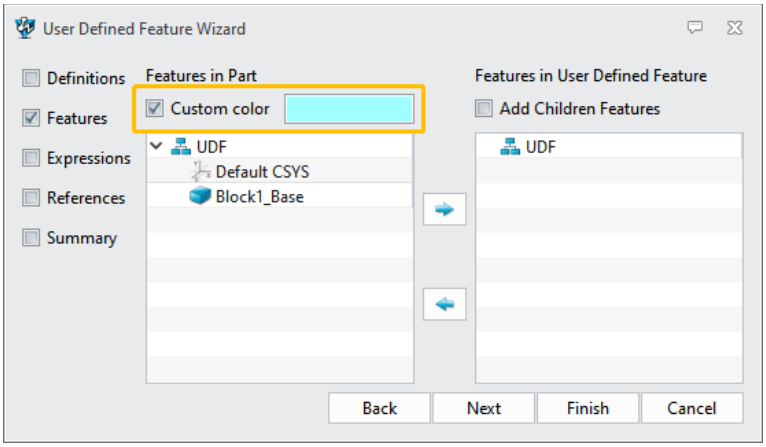
It can be independently controlled while inserting when take feature as “UDF”.

3.5.4.2 UDF Supports More Commands

UDF support more commands including **Internal Sketch, Datum CSYS, Datum Axis, Draft, Loft, Rib, Rib Network, Lip, Curve Mesh, Offset, Extrude, Louver, Normal Pocket, Pattern Feature, and Mirror Feature**, etc.

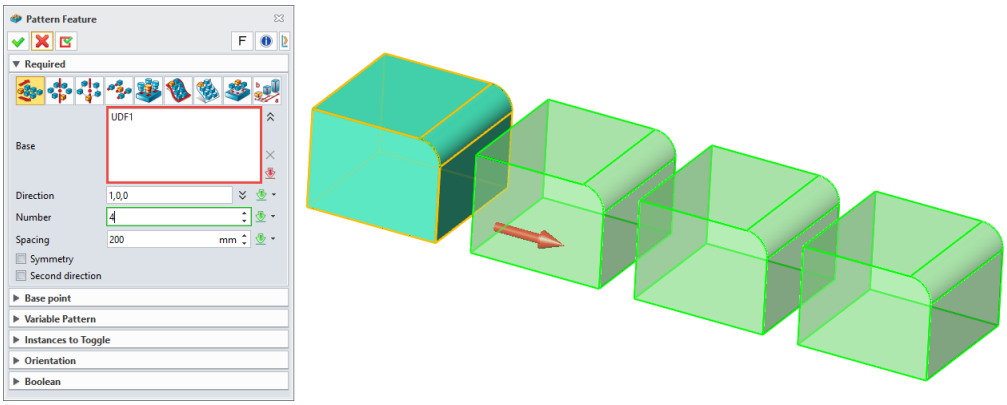
3.5.4.3 UDF Supports Custom Color

We added “Custom color” in the UDF panel, which is used to generate the color of UDF feature.

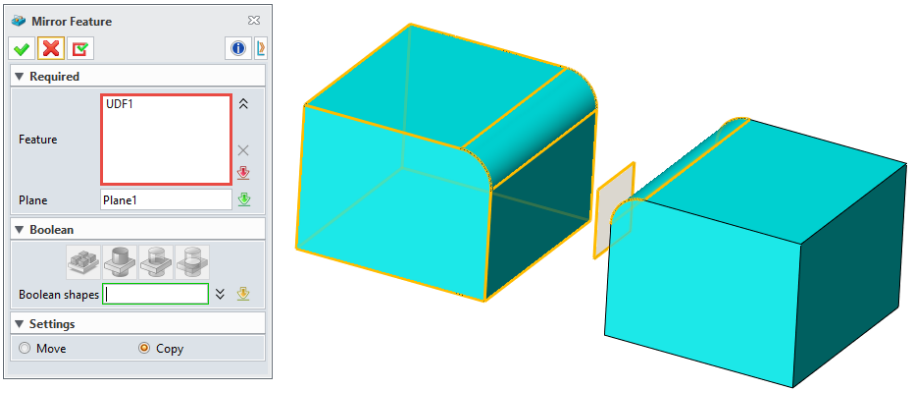


3.5.4.4 “UDF” Pattern Feature/Mirror Feature Support Selecting UDF

The command “Pattern Feature’ supports selecting UDF as input objects.

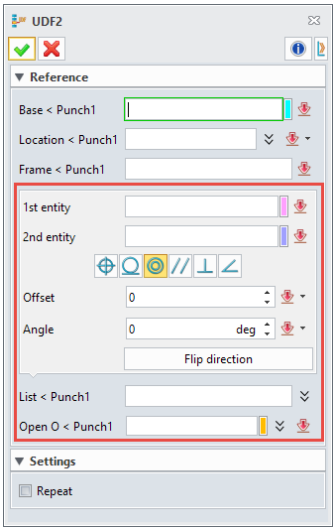


The command "**Mirror Feature**" supports selecting UDF as input objects.

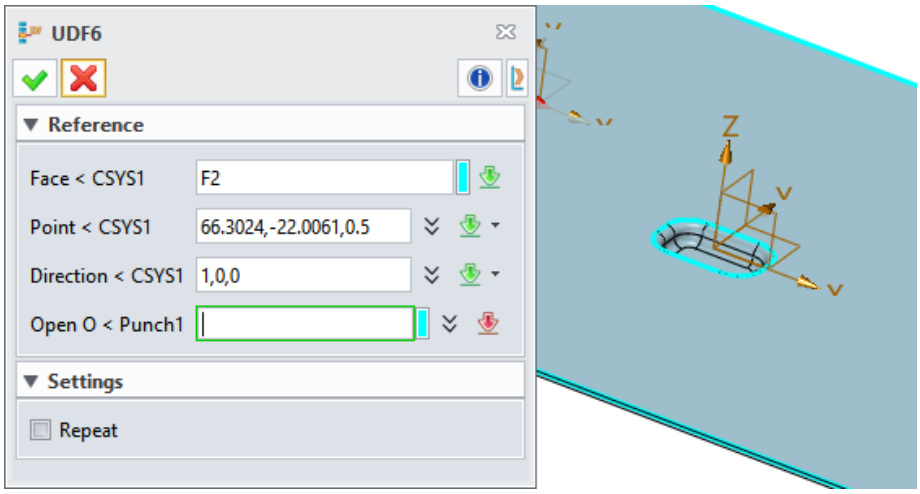


3.5.4.5 UDF Placing Method Improvement

Take creating a Punch as an example, the current user locates punch through constraint method and then generates the feature into UDF. When inserting, the past user needs to define too many options, as the following figure shows:



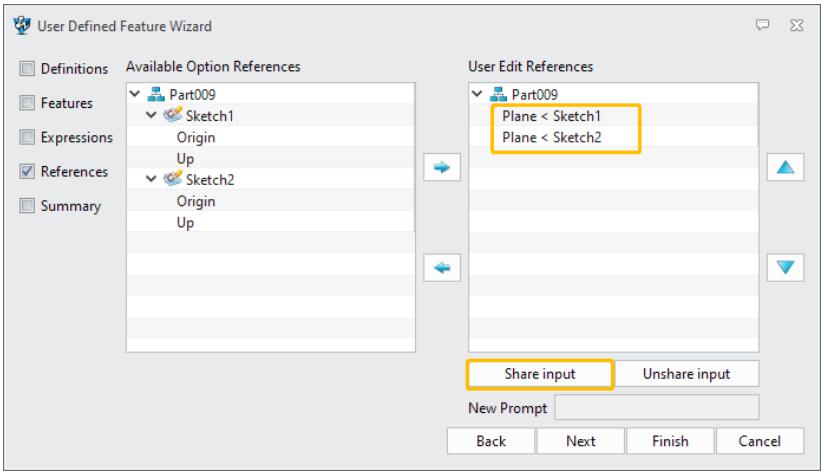
After improvement, we simplified the placing low by optimizing the features of UDF.



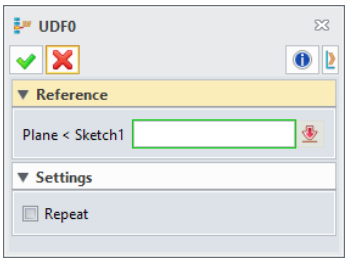
3.5.4.6 New User Editable Expressions

As the following figure shows, we added "**User Edit References**" in UDF panel to simplify the placing method of UDF. When user defines reference, the system will list the references in "**User Edit References**" that may need to be modified, where user can add or delete them. When user inserts UDF, only the references in "**User Edit References**" can be modified and the rest references will read the original values without user's additional modification.

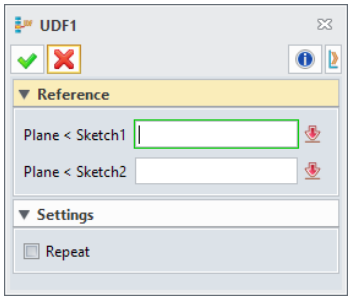
The function of "Share input" is added to "User Editable Expressions" list. When the UDF contains two sketches and the two sketches' planes are the same, during the wizard, user can share the planes. There only needs selecting one plane instead of two planes while inserting UDF.



If "Share input" is clicked, the user's input fields will decrease. As the following figure shows, there only needs one sketch plane inputting.



If not shared, the user's input fields will increase. As the following figure shows, there needs inputting two sketch planes.



➔ Where it is

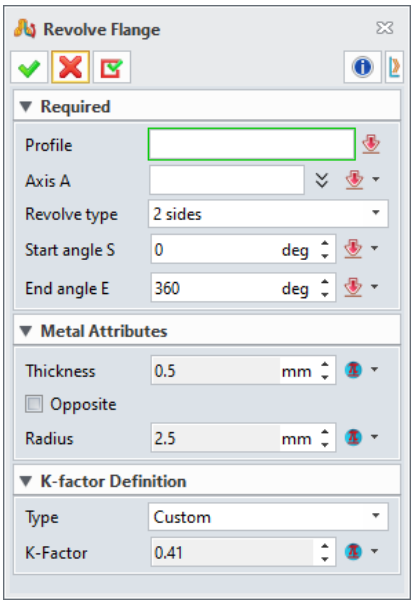
Part Environment >> Tools >>Library >> UDF Wizard & Insert UDF

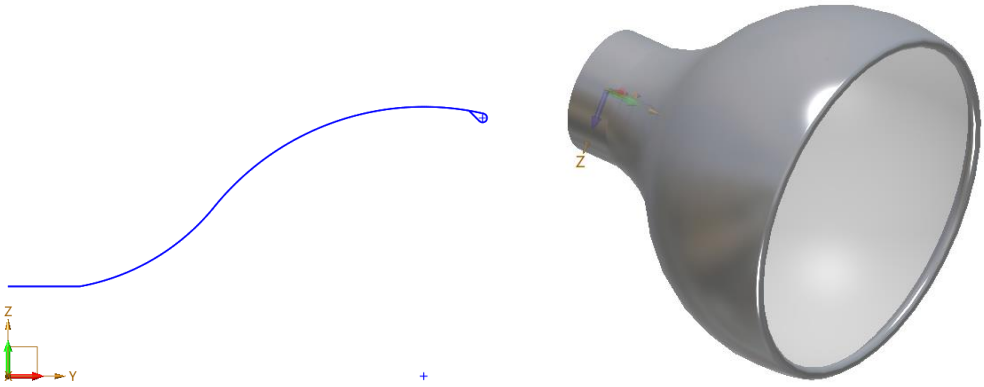
3.6 Sheet Metal Design

3.6.1 ★New Revolve Flange

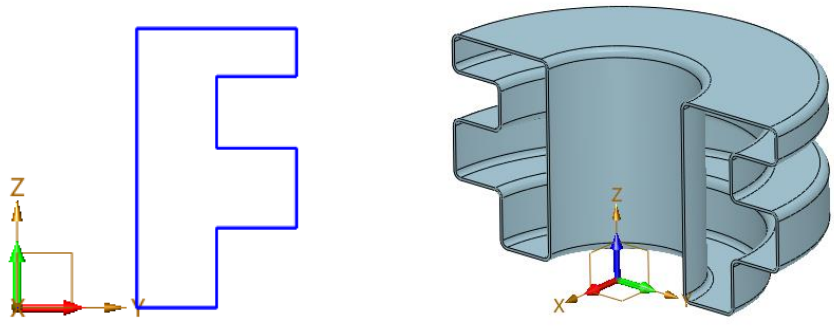
In the product design of metal forming, its product needs to use the function of revolve flange. To speed up the workpiece design process, we added "Revolve Flange" in ZW3D 2023 to support more application scenarios of sheet metal.

Revolve Flange command supports picking an open profile as well as a closed profile. The picked profile object will create a sheet metal solid by revolving an axis. User can set the rotation angle, thickness, bend radius, and K-factor of the sheet metal. An opposite direction is added to the thickness. Besides, the sheet metal thickness, bend radius and K-factor support reading the sheet metal attribute values.





Open Profile



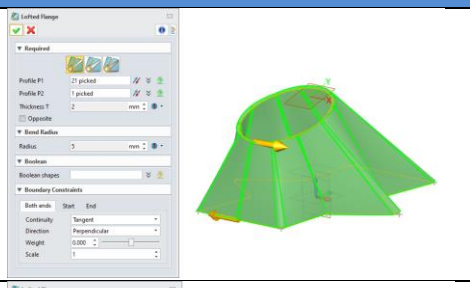
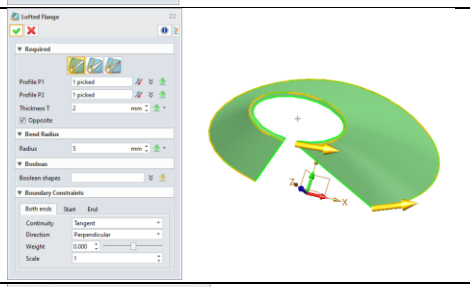
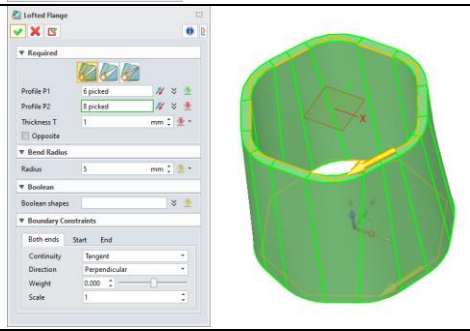
Closed Profile

➔ **Where it is**

Part Environment >> Sheet Metal >> Base >> Revolve Flange

3.6.2 ★ Lofted Flange Improvement

Lofted Flange supports non-parallel profile. Moreover, it supports lofting to solid edges and some complicated curves including arcs, spline curves and equation curves (2D and 3D). Meanwhile, the flattenable lofting and the bending lofting are both supported for better modeling of fully closed final parts after welding.

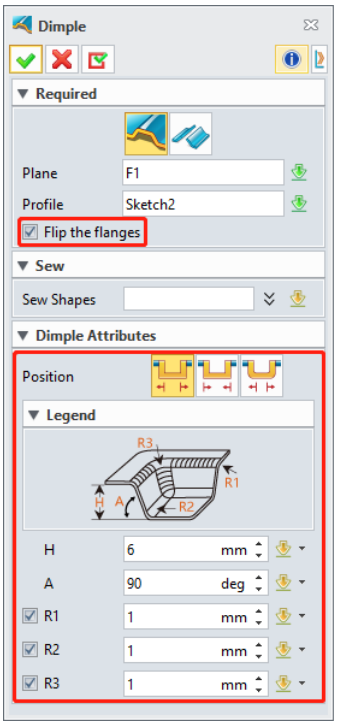
Improvement	Main improvement	Case
Non-parallel profile lofting	Support the sheet metal lofting design of the non-parallel profile	
Complicated curve lofting	Support the complex curve sheet metal lofting such as splines	
Closed curve lofting	Support the sheet metal lofting of the closed curves	

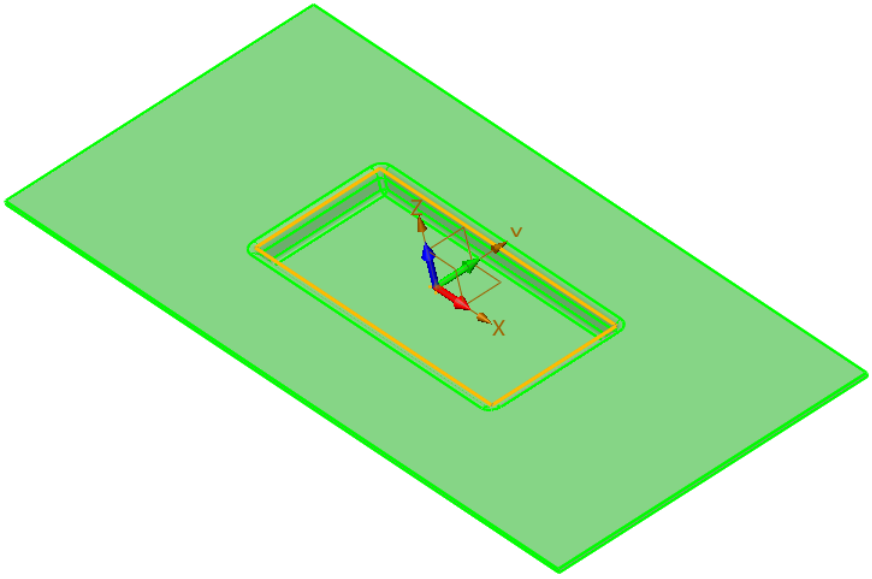
➔ **Where it is**

Part Environment >> Sheet Metal >> Flange >> Lofted Flange

3.6.3 Dimple Improvement

Dimple command newly supports defining material side and flipping the flange direction. The cases of non-tangent sketch, cross boundary, and cross bending are newly supported. Besides, user can define filleting sidewall.



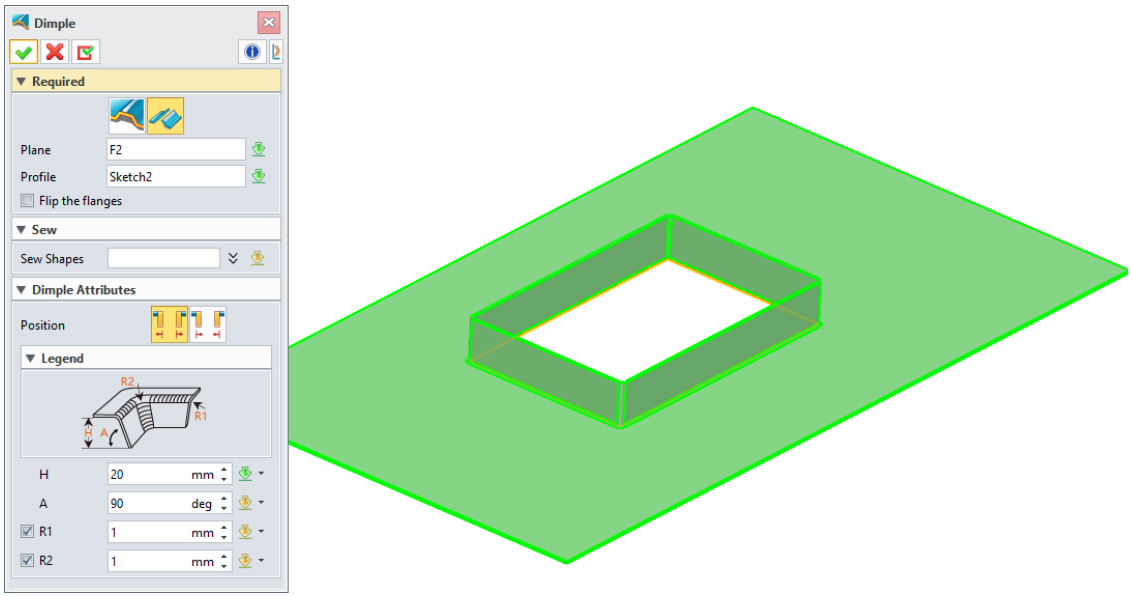


➔ **Where it is**

Part Environment >> Sheet Metal >> Form >> Dimple

3.6.4 Flared Hole Improvement

Flared Hole command supports defining the material side and flipping the opposite direction. The non-tangent sketch, cross boundary, and cross bending are newly supported. Besides, user can define filleting sidewall.

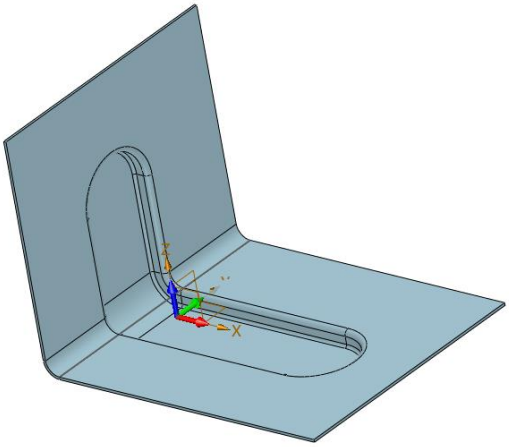


➔ **Where it is**

Part Environment >> Sheet Metal >> Form >> Dimple

3.6.5 Fold by Line Improvement

Fold by Line command supports folding sheet metal with dimple.

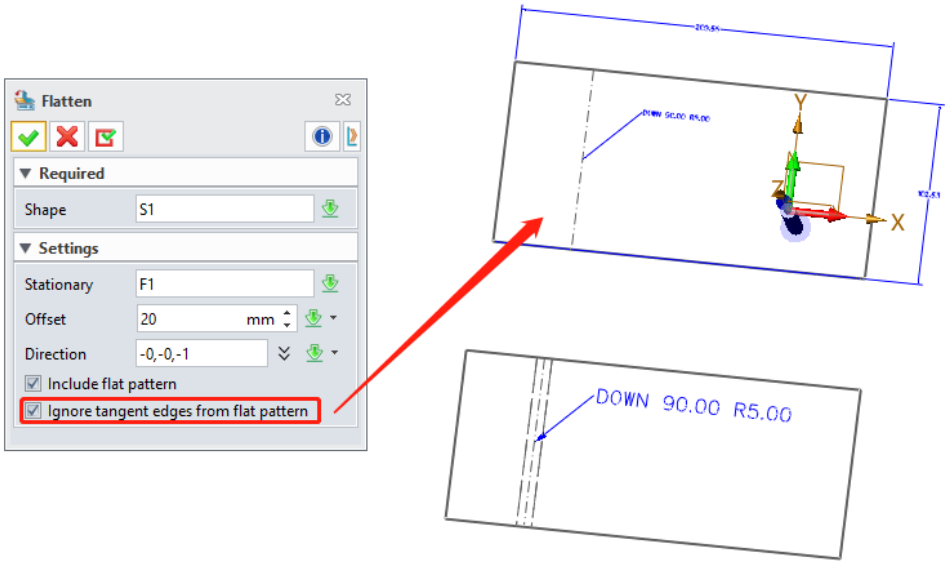


➔ **Where it is**

Part Environment >> Sheet Metal >> Flange >> Fold by Line

3.6.6 Flatten Improvement

Flatten command supports ignoring tangent edges from flat pattern.



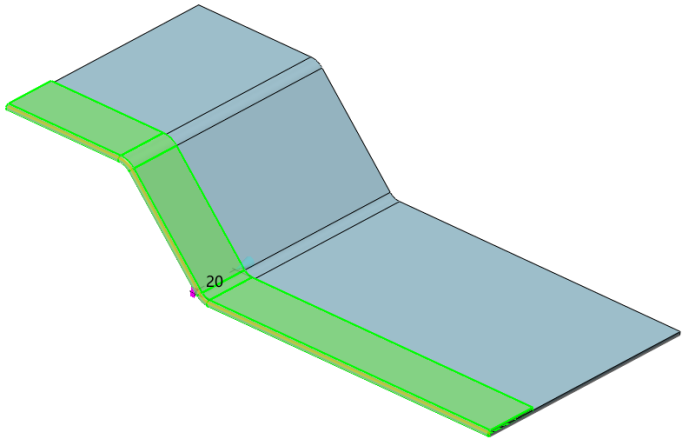
➔ Where it is

Part Environment >> Sheet Metal >> Bend >> Flatten

3.6.7★New Double Bend Flange

It takes two times of folding operations to construct a flange on circular edges before forming a shape as well as two times of unfolding before folding. Therefore, such flanges are called double bend flange. In addition to support the construction of double bend flange, it also supports unfold the double bend flange.

Only the line can be selected as the sketch drawn by Flange with Profile is 2D sketch. Swept Flange can only be constructed on the stationary face. Thus, neither Flange with Profile nor Swept Flange can support double bend flange. Among the modeling commands of sheet metal, only **Full Flange**, **Partial Flange** and **Hem Flange** support double bend flange.

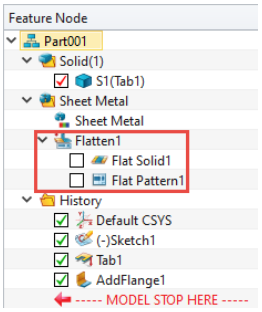


➔ **Where it is**

Part Environment >> Sheet Meal >> Flange >> Full Flange/Partial Flange/Hem Flange

3.6.8 Automatically Add Flatten Attribute

To facilitate acquiring the profile dimension data after product unfolding, namely the minimum length, width, and height in ZW3D 2023, the system will automatically add flat nodes when user is creating a sheet metal. The nodes include **Flat Solid** and **Flat Pattern** which both are under the depressed and user can modify their status as request.

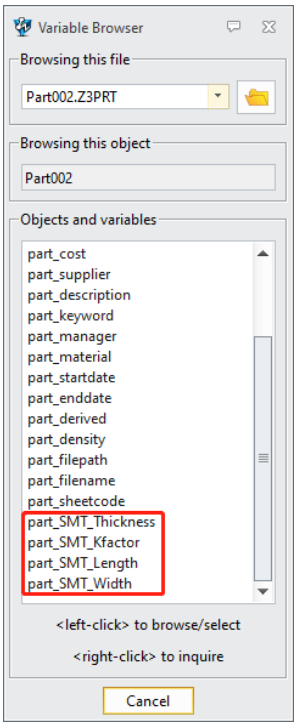


➔ **Where it is**

Part Environment >> Manager >> Sheet Metal >> Flatten

3.6.9 New Sheet Metal Variables

We added four system variables: **part_SMT_Length**, **part_SMT_Width**, **part_SMT_Thickness** and **part_SMT_Kfactor** as new sheet metal variable in ZW3D 2023.

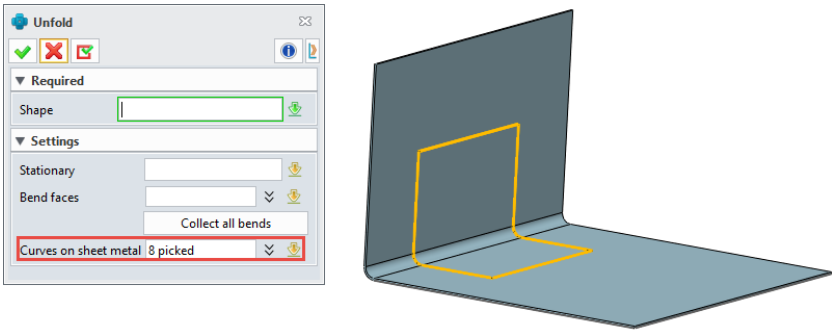


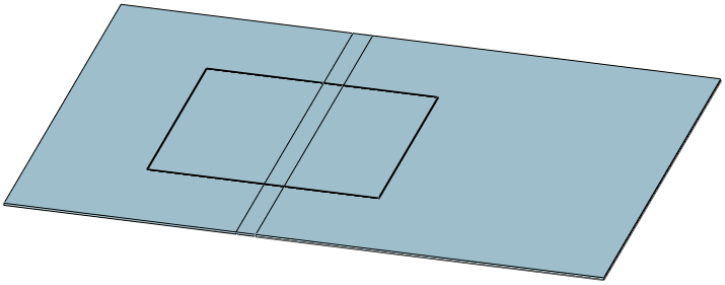
➔ **Where it is**

Part/Sketch/Drawing Sheet Environment >> Tools >> Variable Browser

3.6.10 New Curves on Sheet Metal in Unfold/Fold

If geometry objects of wireframe are placed on the flat plane of sheet metal, the picked curves will be folded as the sheet metal when it is being folded. The geometry objects of wireframe being unfolded with sheet metal is also supported, as following figure:



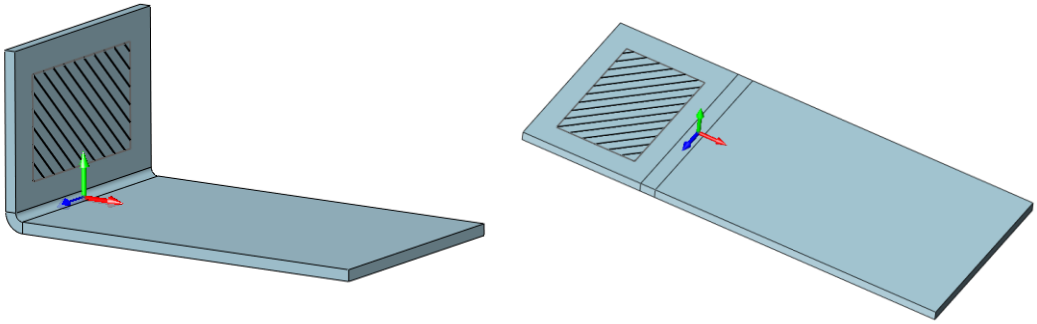


➔ **Where it is**

Part Environment >> Sheet Metal >> Bend >> Unfold/Fold

3.6.11 Cosmetic Sketch Fold as Sheet Metal

When sketch plane is on the sheet metal plane, the cosmetic sketch will be automatically folded while opening the sheet metal, so that ensure the sketch keeping on the sheet metal plane.



➔ **Where it is**

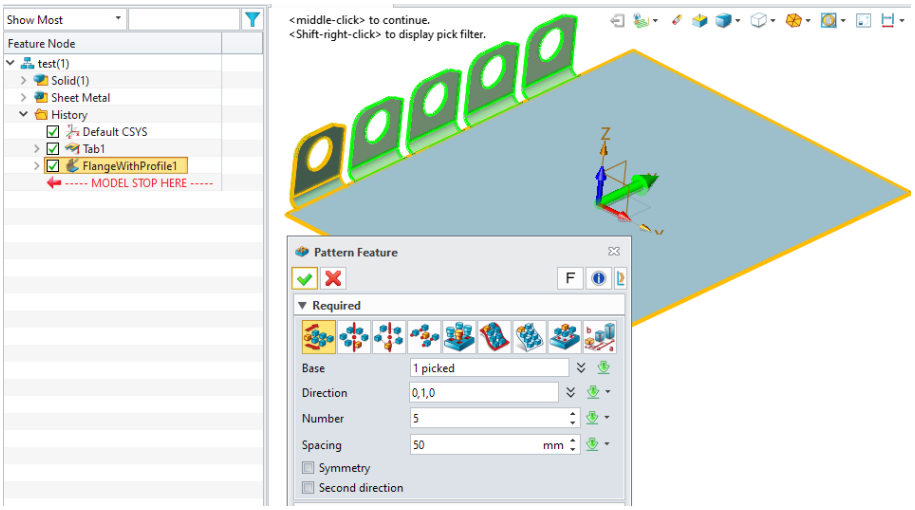
Part Environment >> Shape >> Basic Shape >> Cosmetic Sketch

3.6.12 Sheet Metal Flange Improvement

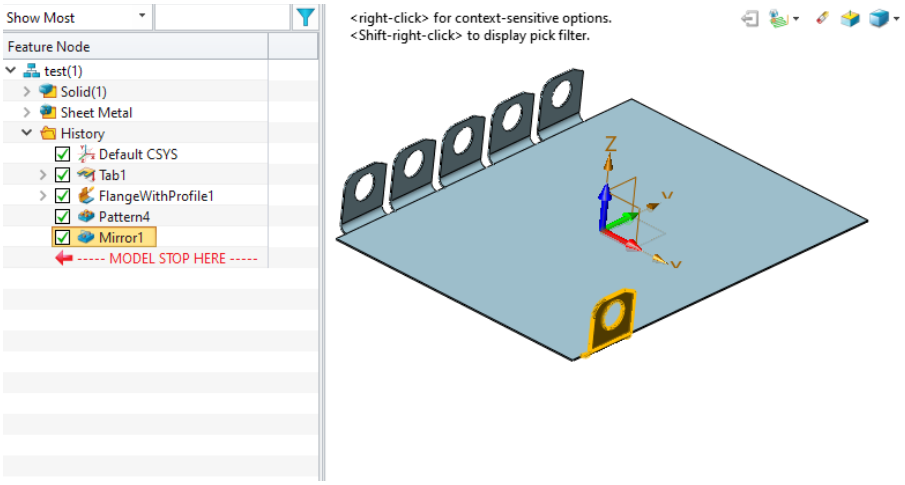
3.6.12.1 Flange with Profile Supports Pattern Feature/Mirror Feature

ZW3D 2023 supports pattern feature and mirror feature of “Flange with Profile”. So far, there are three flange commands: **Full Flange**, **Partial Flange** and **Flange with Profile** that support pattern feature/mirror feature.

Flange with Profile supports pattern feature during sheet metal design.



Flange with Profile supports mirror feature during sheet metal design.

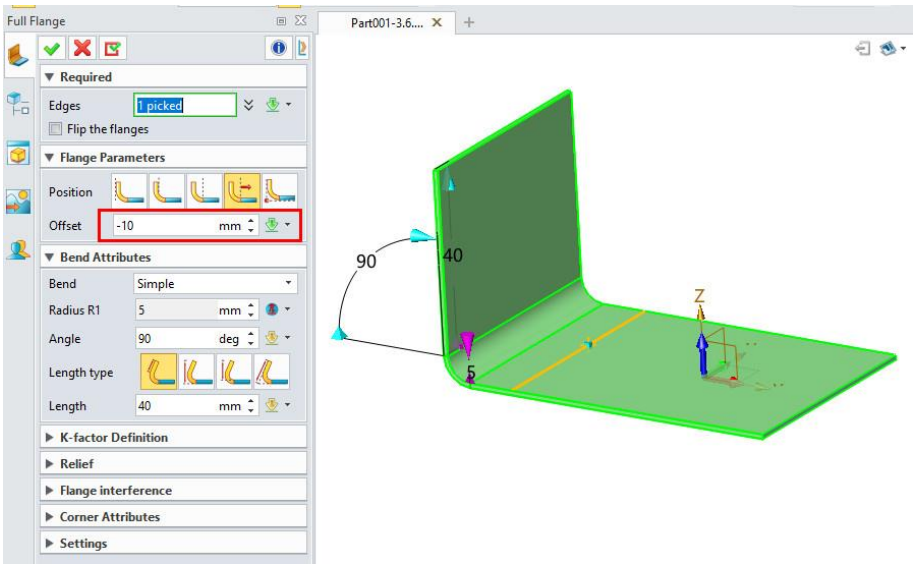


➔ Where it is

Part Environment >> Shape >> Basic Editing >> Pattern Feature/Mirror Feature

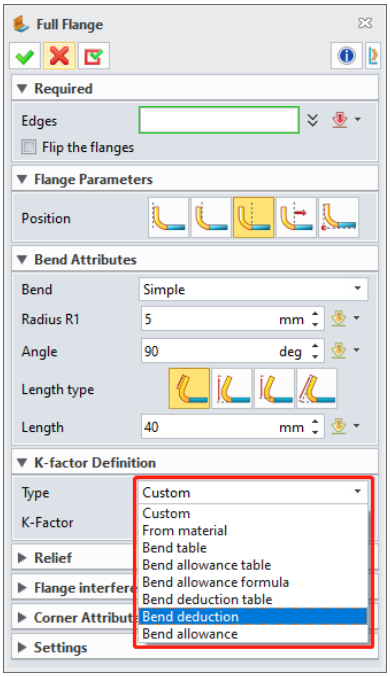
3.6.12.2 Offset Supports Negative Value

When creating a flange, user can input a negative value in the offset while flange position type is set as Offset.



3.6.12.3 New Bend Deduction

In realistic technics, there is a case that use the bend deduction method to calculate the length of sheet metal while folding the bend. We added “Bend deduction” as a type in K-factor definition while creating flange in ZW3D 2023.

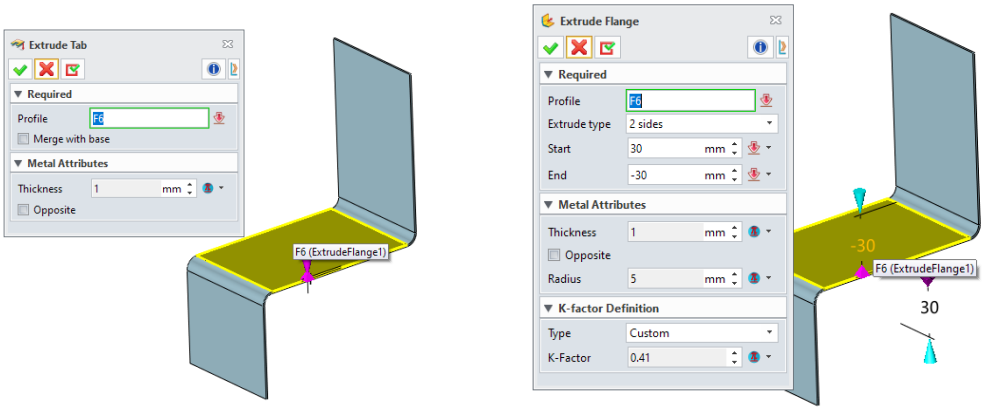


➔ Where it is

Part Environment >> Sheet Metal >> Flange

3.6.13 Extrude Tab and Extrude Flange Improvement

We improved the profile input in commands “Extrude Tab” and “Extrude Flange”, which supports selecting surface as an internal sketch plane.

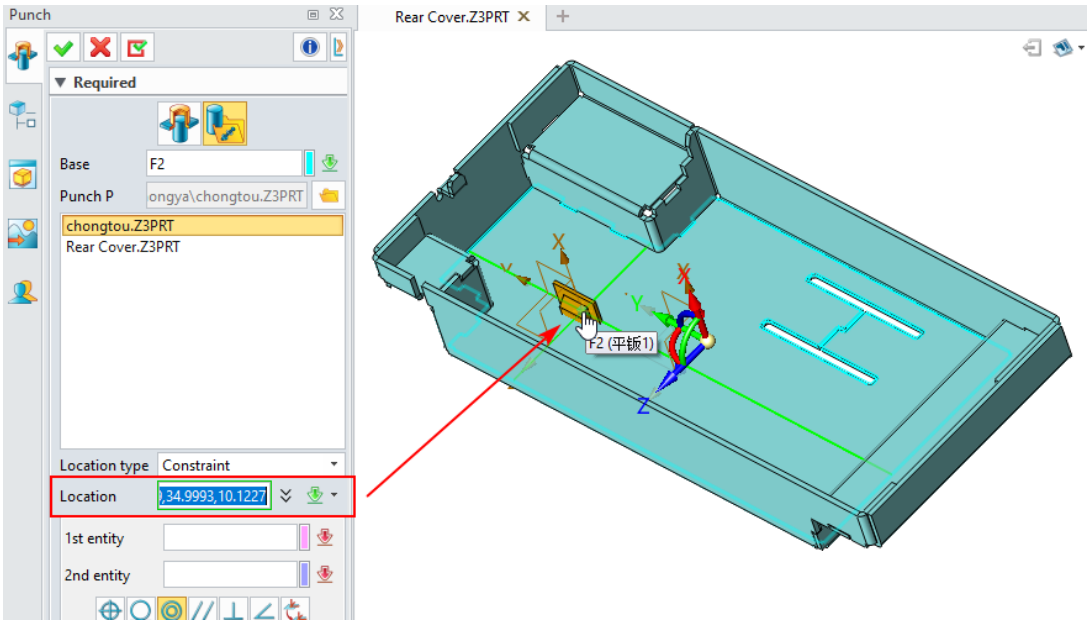


➔ Where it is

Part Environment >> Sheet Metal >> Base >> Extrude Tab/Extrude Flange

3.6.14 Punch Improvement

In the method of “Punch from file”, we added the field “Location” in the constraint. When the punch is placed in constraint way, the punch locates at origin of coordinate by default. At this point, user can activate the location input field and select any point to place the punch position, and then the corresponding constraint can precisely locate the position.

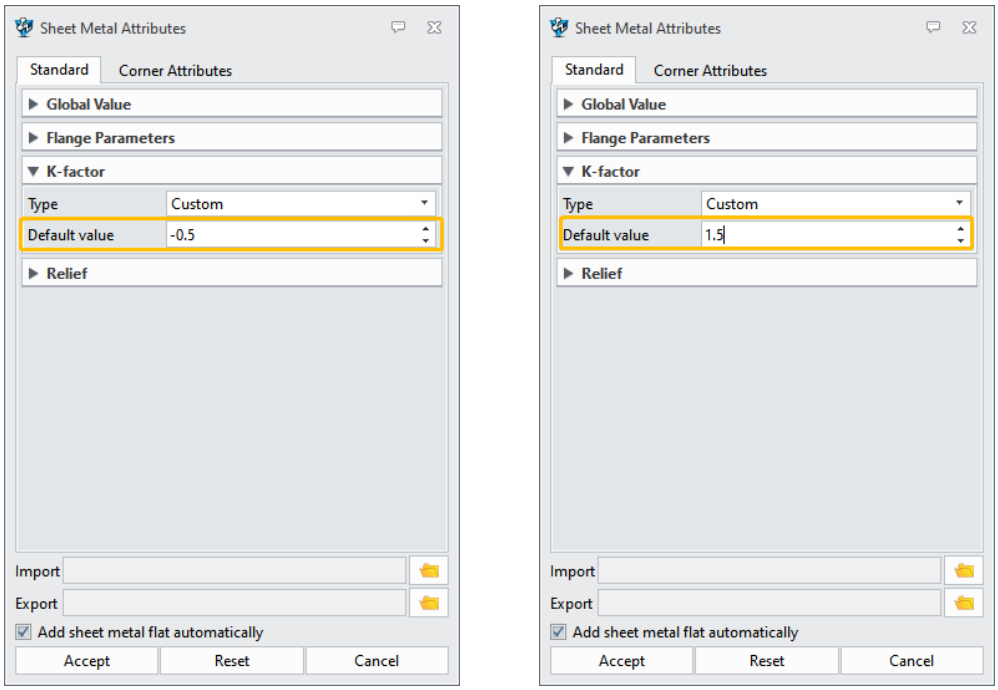


➔ **Where it is**

Part Environment >> Sheet Metal >> Form >> Punch

3.6.15 K-factor Supports Greater Range

The K-factor value allows to set the number that is smaller than 0 or greater than 1 while designing sheet metal.



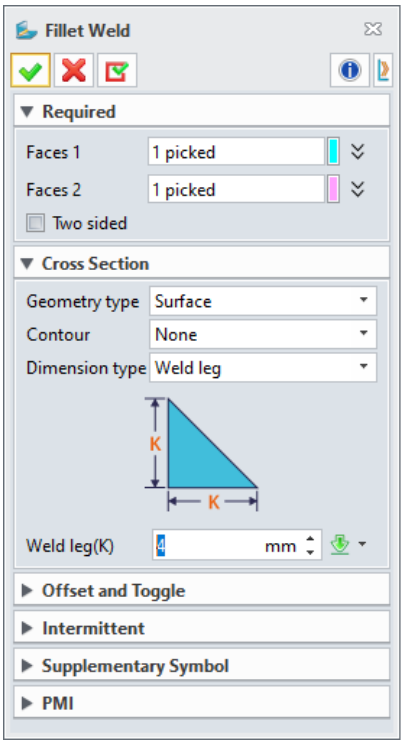
➔ **Where it is**

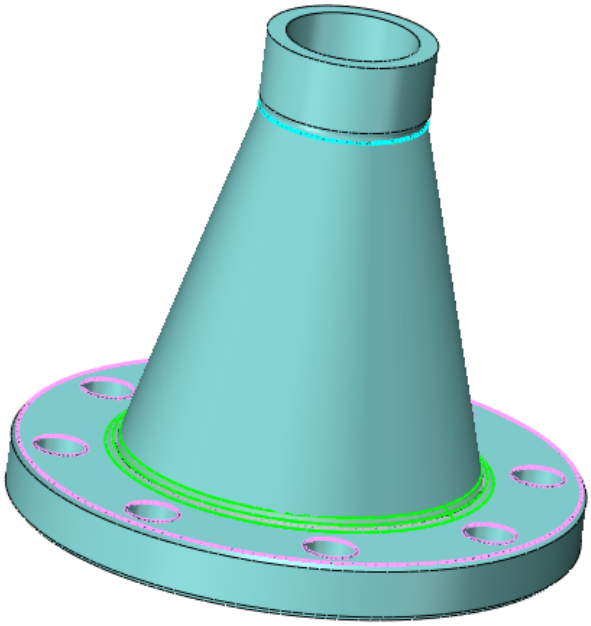
Part Environment >> Sheet Metal >> Sheet Metal Attributes & Each Flange Command

3.7 Weldment Design

3.7.1 New Fillet Weldment

With the Fillet Weld function, user can define a Fillet Weldment based on two groups of referred faces. User can also determine the created weldment's Cross Section, Offset and Toggle, Intermittent, Supplementary Symbol, PMI by modifying parameters in different Collapses.





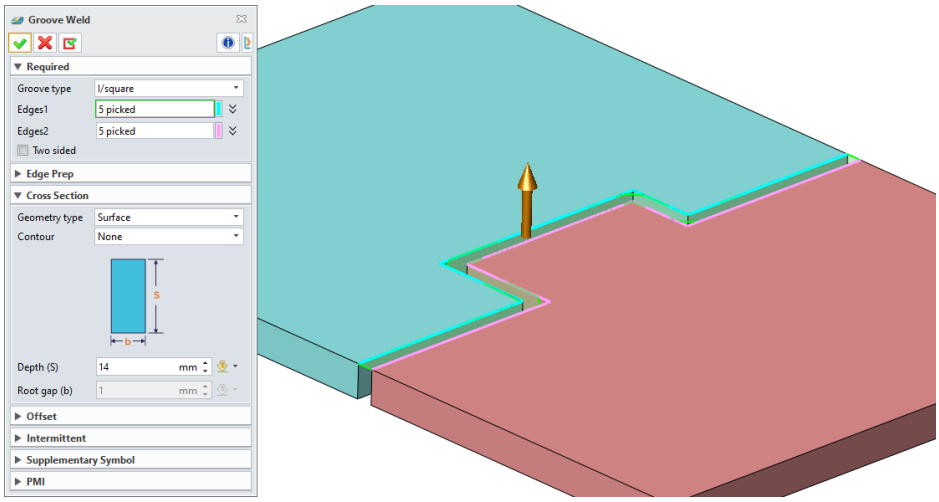
➔ **Where it is**

Weldments >> Fillet Weld

3.7.2 New Groove Weldment

With the Groove Weld function, user can define a Groove Weldment based on two groups of referred Edges. We now support I and V two types of grooves and user can decide whether the selected edges have been prepared for welding.

User can also determine the created weldment's Cross Section, Offset and Toggle, Intermittent, Supplementary Symbol, PMI by modifying parameters in different Collapses.



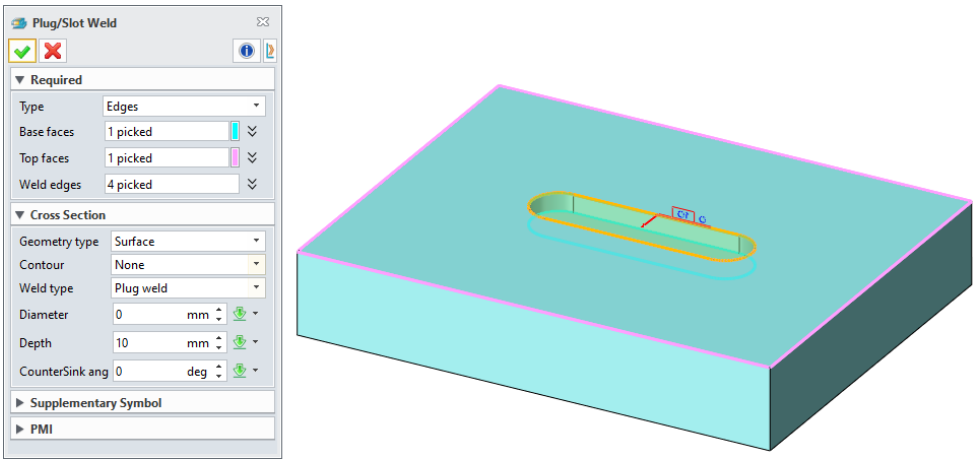
➔ **Where it is**

Weldments >> Groove Weld

3.7.3 New Plug/Slot Weld

Plug/Slot Weld function offers two types of references' selection methods to generate a weldment. For both methods, user need to select the Base faces and Top faces. For method one, user can select "Side faces" in the "Type" drop-down list in the "Required" collapse then the bound of the created Plug/Slot weld will be determined by the Side faces selected. For method two, user can pick the "Edges" option in the drop-down list and the Plug/Slot weld will then be bounded by the edges selected.

User can also determine the created weldment's Cross Section, Supplementary Symbol, PMI by modifying parameters in different Collapses.

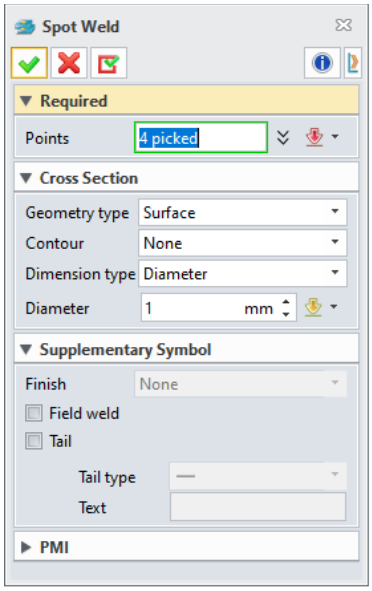


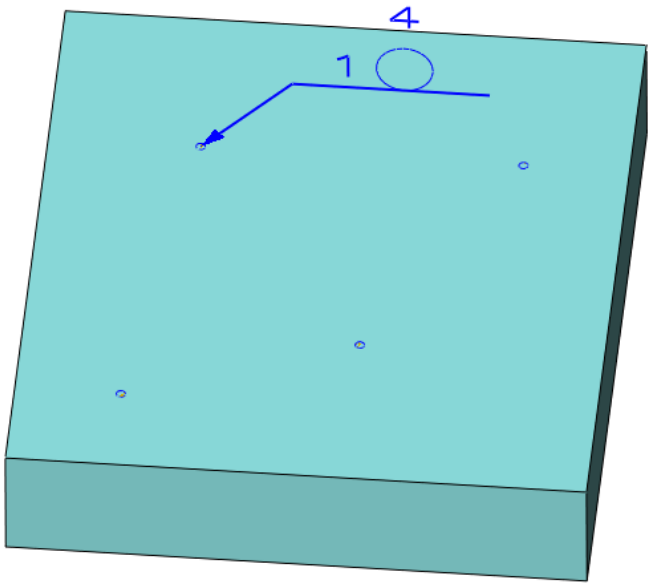
➔ **Where it is**

Weldments >> Plug/Slot Weld

3.7.4 New Spot Weld

With the Spot Weld function, user can define a Spot Weldment based on points selected. User can also determine the created weldment's Cross Section, Supplementary Symbol, PMI by modifying parameters in different Collapses.





➔ **Where it is**

Weldments >> Spot Weld

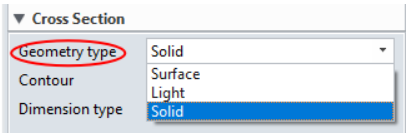
3.7.5 Weldment Improvement

We optimized weldment commands to adjust to more scenarios and meet engineering and customer needs by promoting their usability.

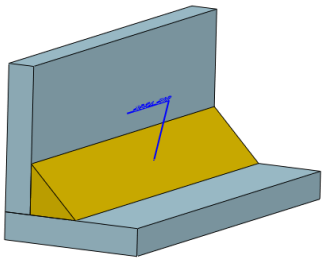
3.7.5.1 Support Solid Weldment

Fillet Weld, Groove Weld, Plug/Slot Weld and Spot Weld support creating solid weldment type as the following shown:

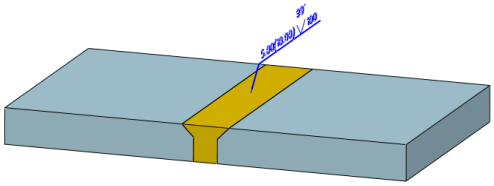
In Cross Section option, user can switch Solid option in Geometry type and create the solid weldment.



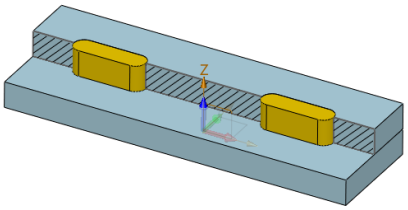
Solid type in Fillet Weldment:



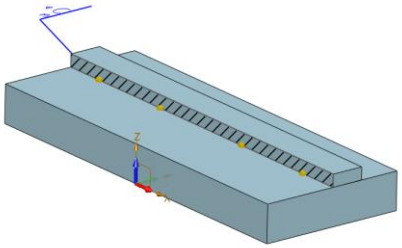
Solid type in Groove Weldment:



Solid type in Plug/Slot Weld



Solid type in Spot Weld:



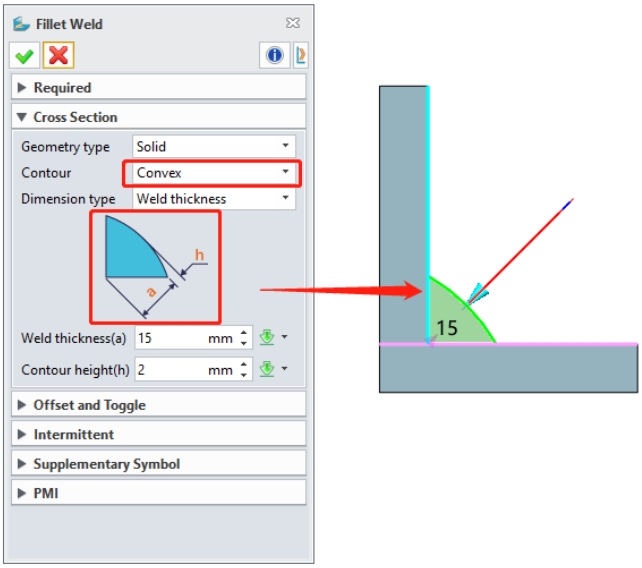
➔ Where it is

Part Environment >> Weldments >> Frame >> Fillet Weld/Groove Weld/Plug/Slot Weld/Spot Weld

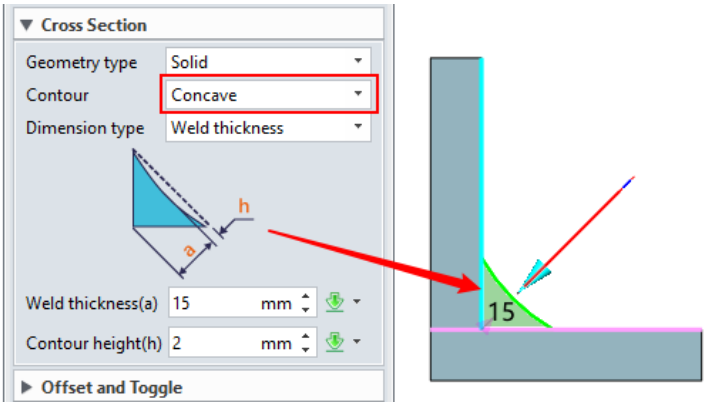
3.7.5.2 Convex Concave Improvement

ZW3D 2023 supports creating the convex or concave type of weldment to generate solid with the actual graphic feature.

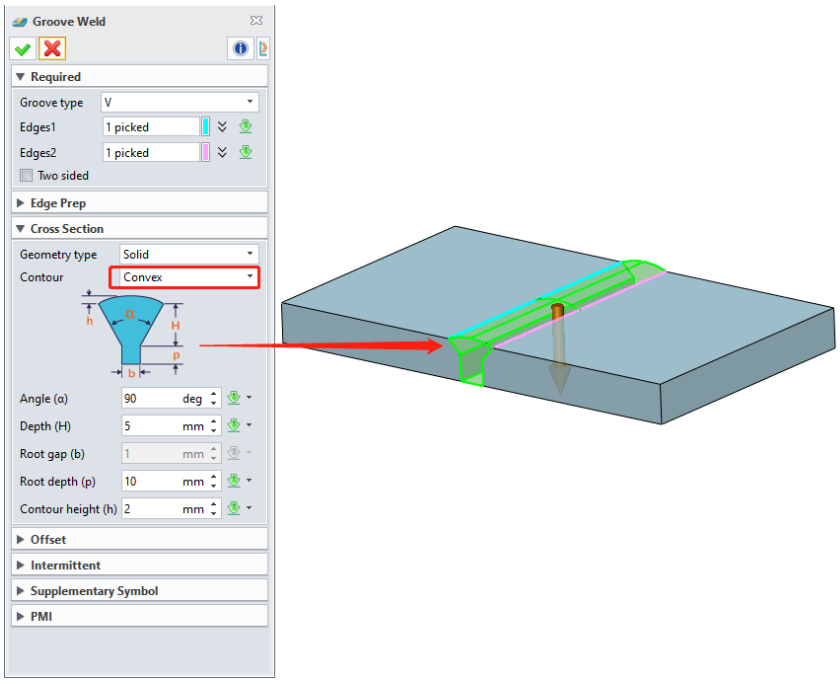
Convex in Fillet Weld:



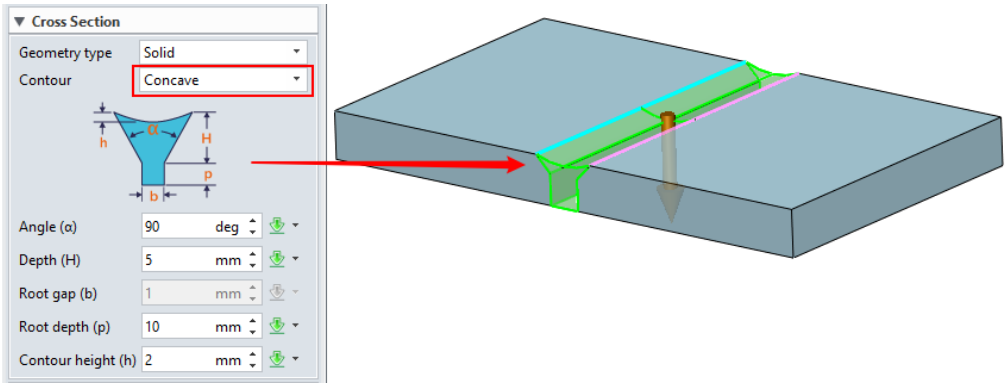
Concave in Fillet Weld:

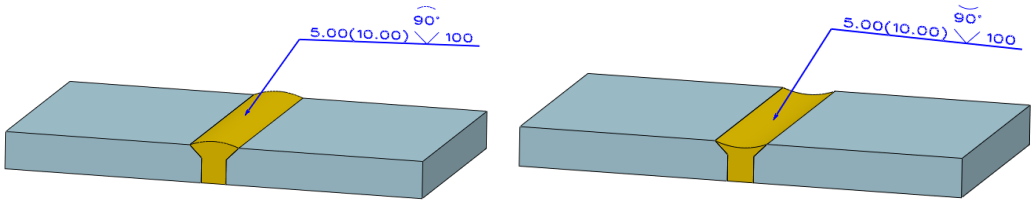


Convex of V type in Groove Weld:



Concave of V type in Groove Weld:



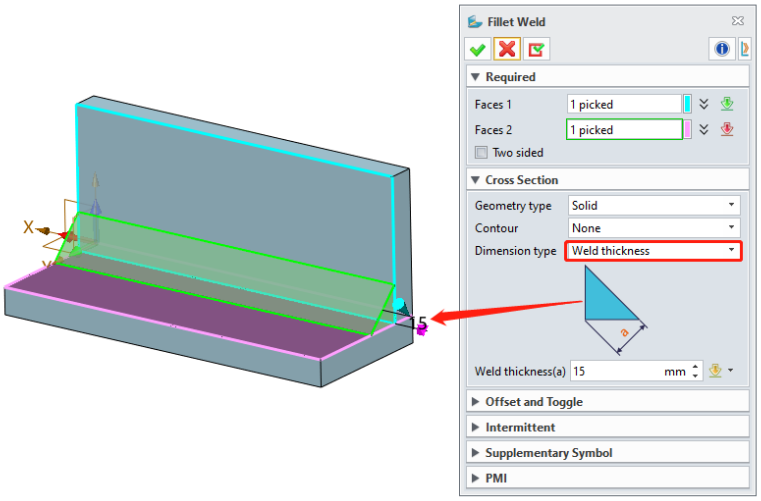


➔ **Where it is**

Part Environment >> Weldments >> Frame >> Fillet Weld/Groove Weld/Plug/Slot Weld/Spot Weld

3.7.5.3 Fillet Weld Improvement

We optimized a way to create fillet weld to adjust to more application scenarios. User can input a weld thickness height to create fillet weld as the following figure shown:



➔ **Where it is**

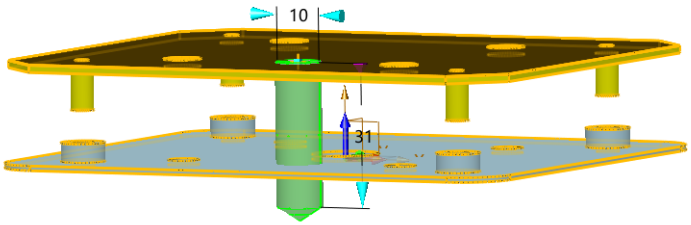
Part Environment >> Weldments >> Frame >> Fillet Weld

3.8 Assembly Design

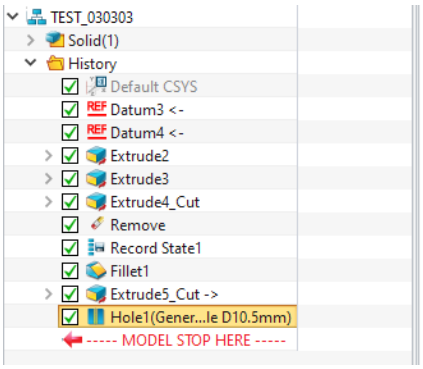
3.8.1 Assembly Feature Improvement

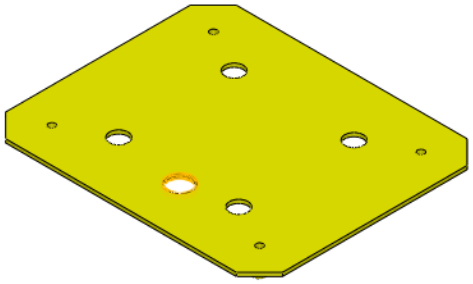
When creating assembly features, if “Propagate feature to components” checked, the assembly features can be transmitted synchronously to the corresponding parts/sub-parts of the assembly components.

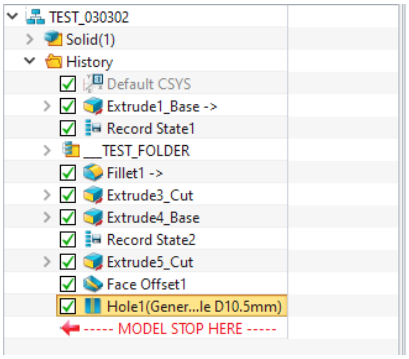
Case: add assembluy “Hole” feature to the assembly.

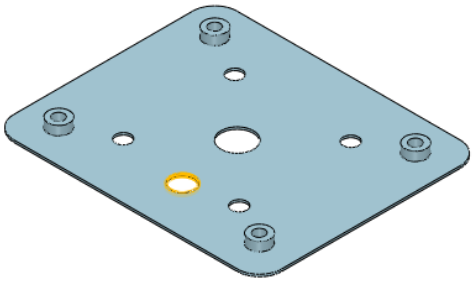


The “Hole” feature in the assembly can add hole features to the corresponding parts/sub-assembly of component through checking option “Propagate feature to components”. The feature nodes from propagated generation cannot be redefined separately in the parts/sub-assembly.









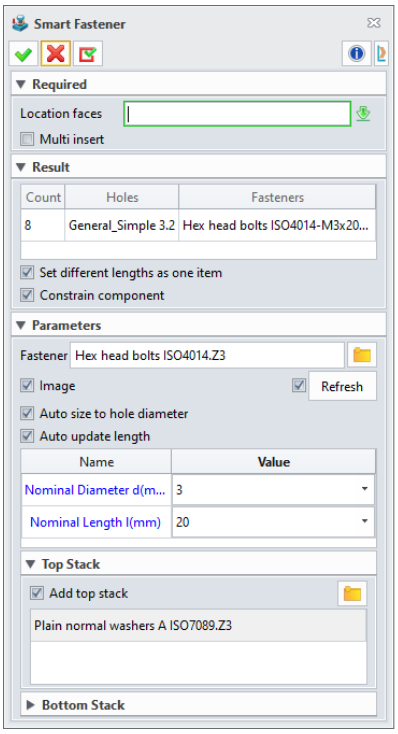
➔ Where it is

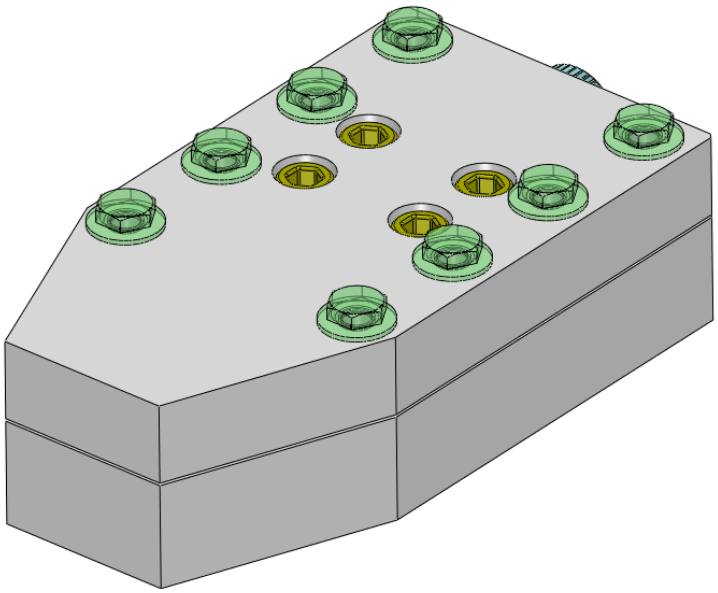
Assembly Environment >> Basic Editing >> Frame >> Cut/Hold/Fillet/Chamfer

3.8.2 ★New Smart Fastener

Smart Fastener command can increase the efficiency to insert standard parts. The command characteristics include:

- Automatically find out the holes that need to insert standard parts based on the feed face.
- Automatically recommend the type of standard parts based on the hole size and depth.
- Insert gaskets or nuts that match with the bolts at the same time.
- Automatically insert constraints without manual operation after adding standard parts.





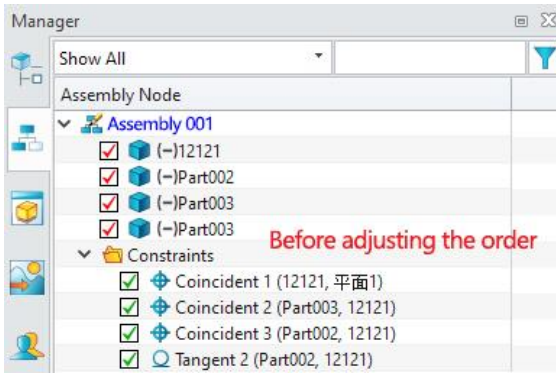
➔ **Where it is**

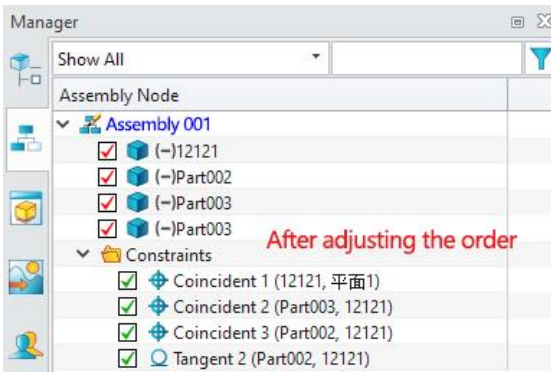
Assembly Environment >> Component >> Smart Fastener

3.8.3 Assembly Constraint Improvement

3.8.3.1 Assembly Constraint Order Adjustment

In the separate mode of assembly constraint, user can directly drag the assembly constraints to adjust their order through the new function of assembly order adjustment.



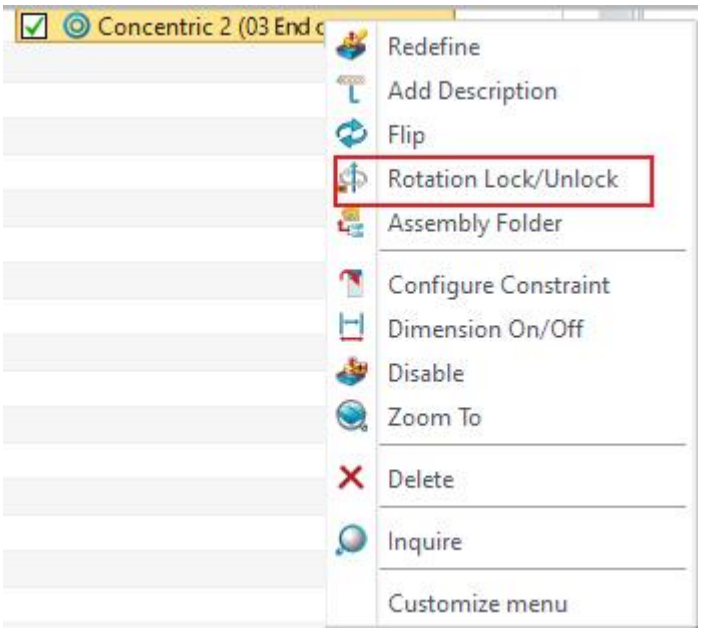


➔ **Where it is**

Assembly Environment >> Manager >> Assembly Node >> Constraint

3.8.3.2 Concentric Constraint Improvement

We added Rotation Lock/Unlock to the context menu for the **concentric constraint**, so that user can directly set the concentric constraint on the right context menu.



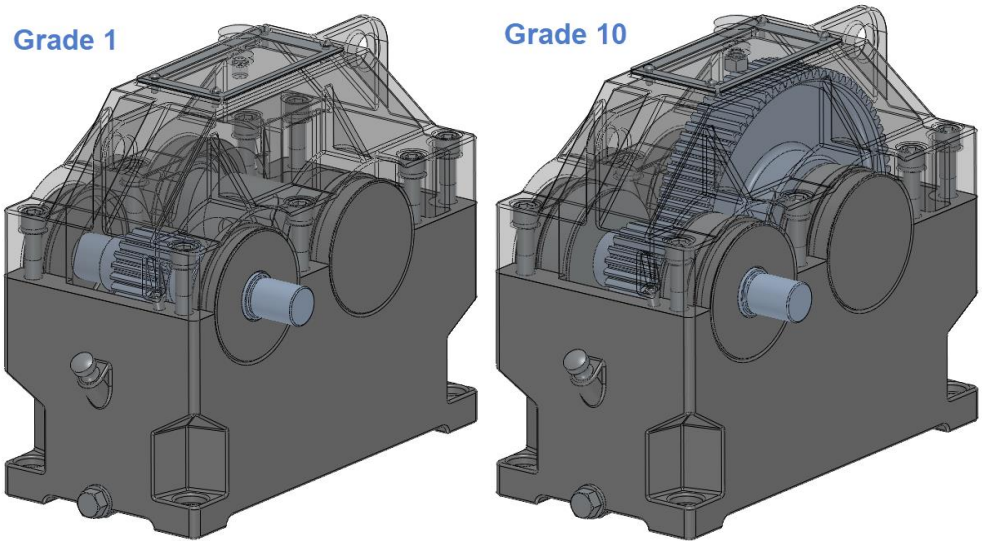
➔ Where it is

Assembly Environment >> Assembly Manager >> Constraint Context Menu >> Rotation Lock/Unlock

3.8.4 Simplified Wrap Improvement

3.8.4.1 New Sensitivity Control

We added Sensitivity to simplify wrap displaying contents more precisely. The rule of sensitivity control is classified into 10 grades. The higher sensitivity, the more details of the simplified wrap mode.



➔ **Where it is**

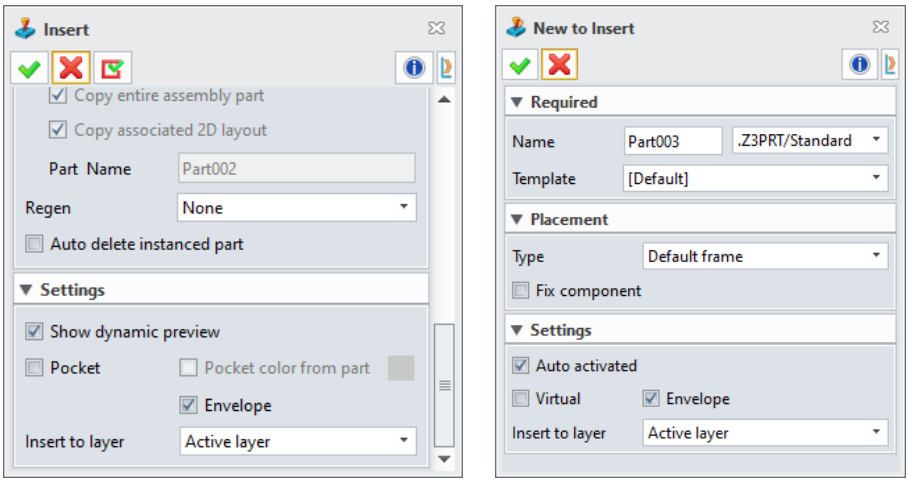
Part Environment >> Data Exchange >> Simplified Wrap

3.8.5 ★New Envelop

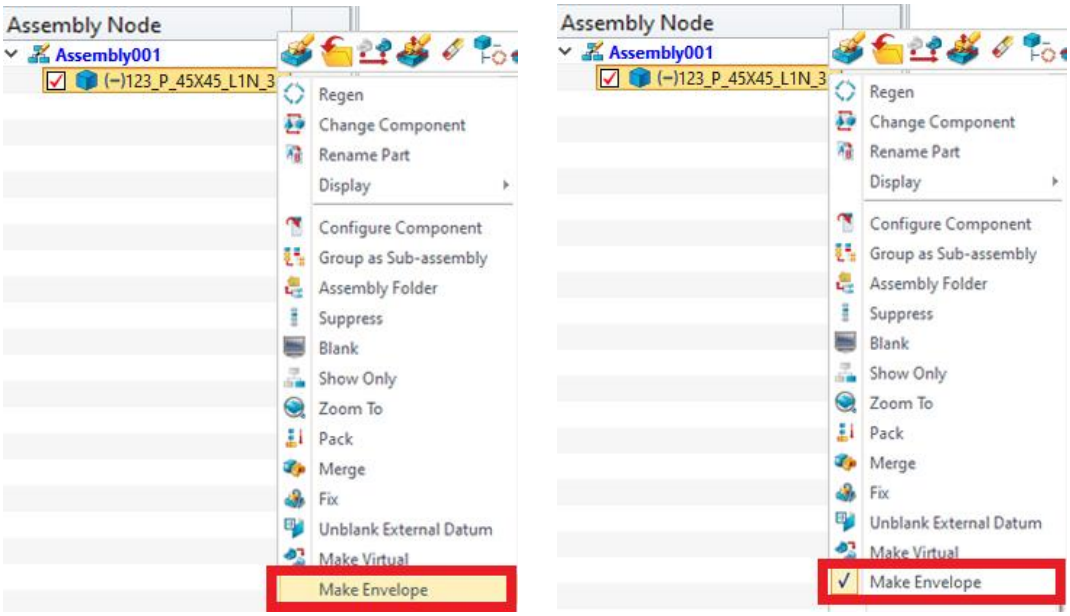
The main purpose of envelope is to set some parts as envelopes in assembly. The enveloped parts will not be calculated when inquiring attribute information such as quality and geometry and exporting BOM tables. The envelope projected view can be displayed or hidden in the drawing sheet as user's need.

The main function is applied to the following cases:

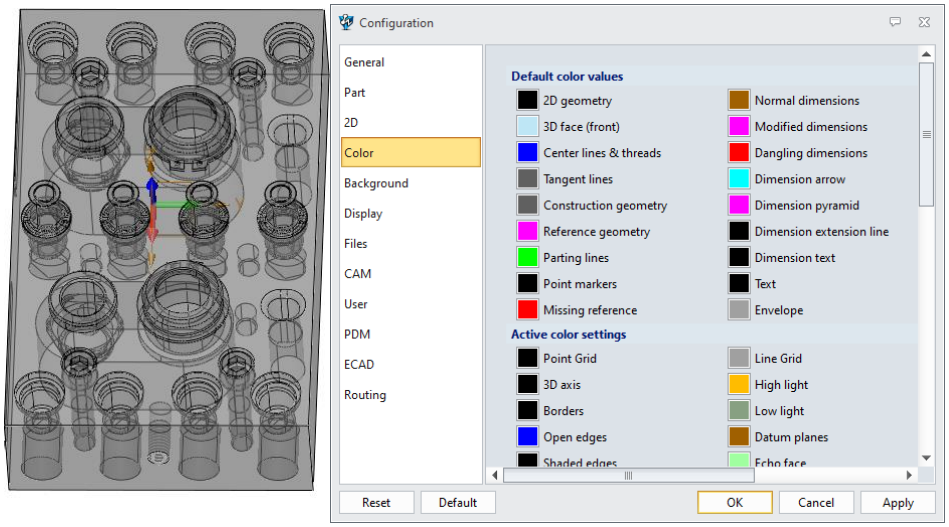
1. When user is inserting or creating a new part in assembly, he or she can specify the components into an envelope.



2. After selecting some components on the assembly tree, user can check “Make envelope” on the right context menus to make the components into envelop.

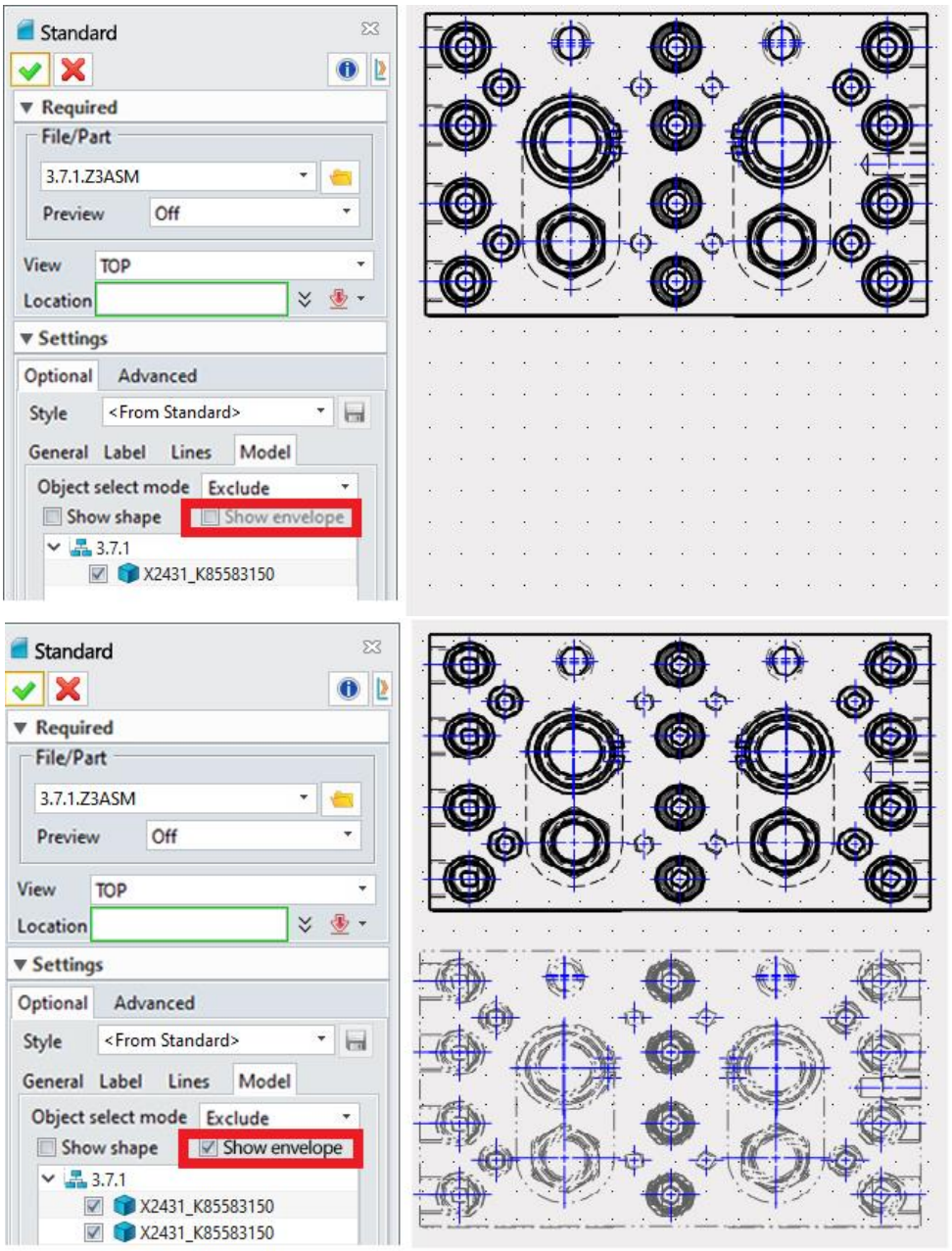


3. The system default transparency 50% light gray (R:160, G:160, B:160) as envelope color. User can change the envelope color in color configuration.



4. The envelope can carry on all relevant assembly editing and assembly constraint operations such as to pattern, mirror, move, and rotate along with other assembly components.

5. When generating the drawing sheet, the envelops are not displayed by default. When selecting to display the envelopes, they will be displayed in double dotted lines and adjust their lines and line width to the view projection.



6. When generating 3D BOM or drawing sheet table, the envelope information is not displayed in the table.

➔ Where it is

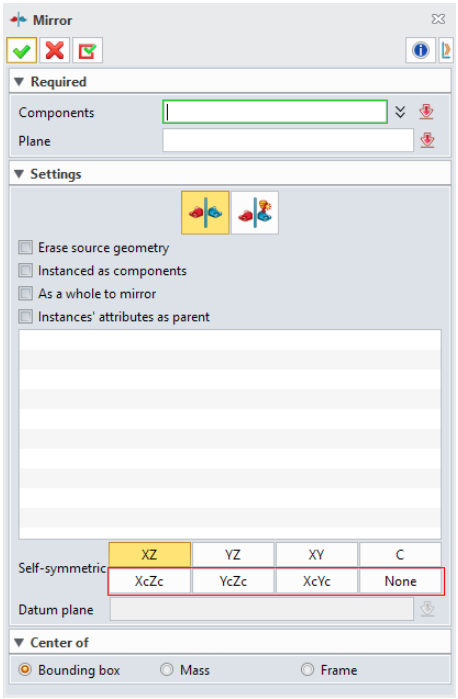
Assembly Environment >> Component >> Insert

3.8.6 Assembly Mirror Improvement

In the assembly environment, the original Move and Copy in Mirror have been upgraded to Copy and Duplicate to solve more complicated mirroring requirements.

3.8.6.1 Self-symmetric Optimization

We added options to self-symmetric datum plane to allow user to select self-symmetric plane of datum about the center of mass inertial coordinate system (XcYcZc) and None for positioning mirror only.

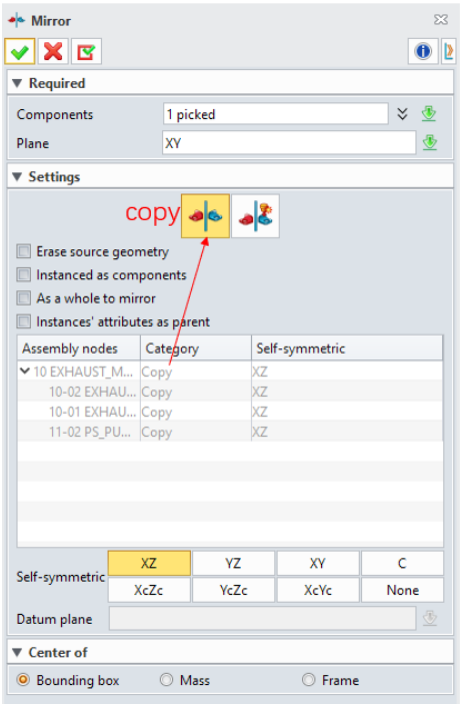


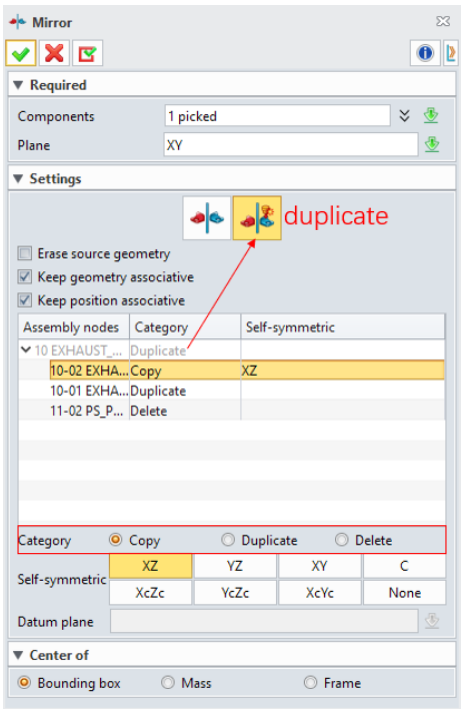
3.8.6.2 Sub-assembly Mirror Optimization

When creating mirror for sub-assembly, user can customize the mirror category for the next components through single selection or multiple selection. User can directly specify mirror state category for each component through single selection button on the mirror operational interface.

When Copy is selected as the mirror category of sub-assembly, its sub-component mirror form can also be copied. It's not supported that user edit sub-component mirror categories and self-symmetric planes.

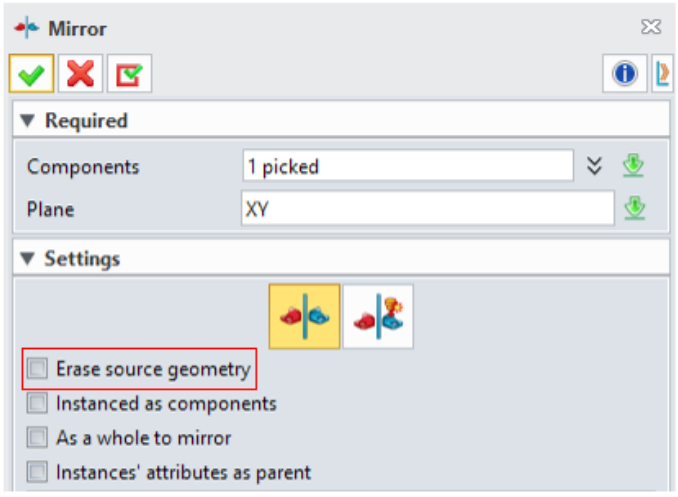
When Duplicate is selected as the mirror category of sub-assembly, its sub-components can be single selection or multiple selection and set mirror category as Duplicate, Copy or Delete.





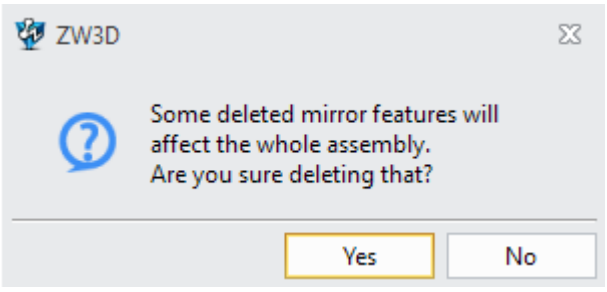
3.8.6.3 Full Mirror Optimization

When completely mirroring components, user can check the option “Erase source geometry” to keep the source component objects.

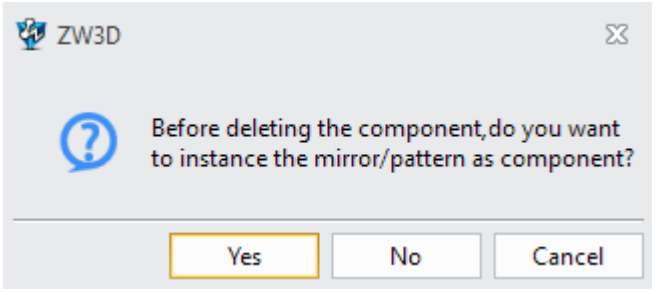


3.8.6.4 Deleting Source File Warning

A warning will pop-up when user is deleting mirror features and mirror source components in case of mistaking operation.



Deleting mirror feature warning



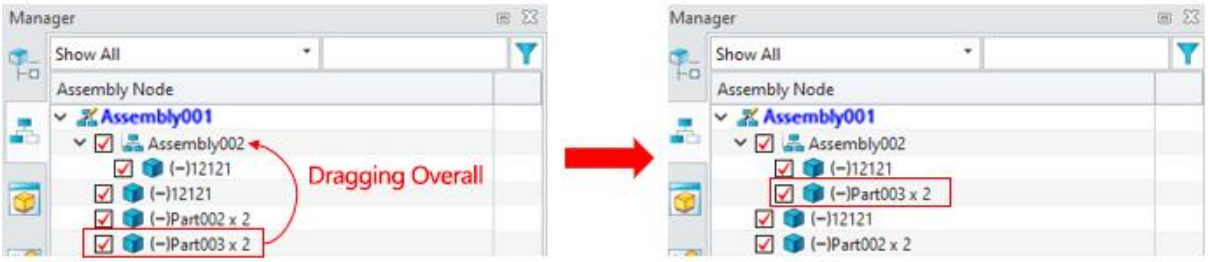
Deleting mirror source file warning

➔ **Where it is**

Assembly Environment >> Assembly >> Basic Editing >> Mirror

3.8.7 Support Dragging Overall Component

In the assembly node, when dragging packaged parts to assembly folder or other assembly level, user can directly drag the overall packaged parts. If unpackaged state, user can only drag a single component.

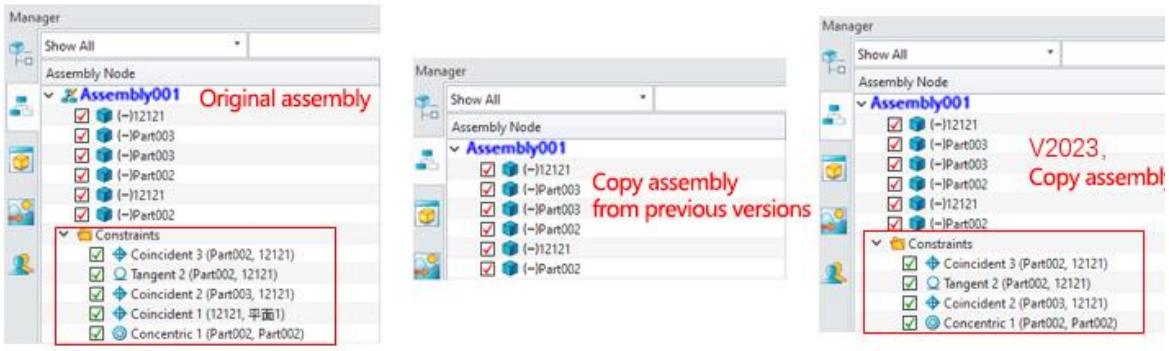


➔ **Where it is**

Assembly Environment >> Assembly Manager >> Component Context Menu >> Pack >> Drag

3.8.8 Copy Component Improvement

In the assembly environment, the original constraint from copying components to assembly can be kept and user does not need to re-assembly constraints.



Note：When copying, the components of constraints must be copied together to keep the original constraint relationship. If only copying one component, the original constraint relationship cannot be kept.

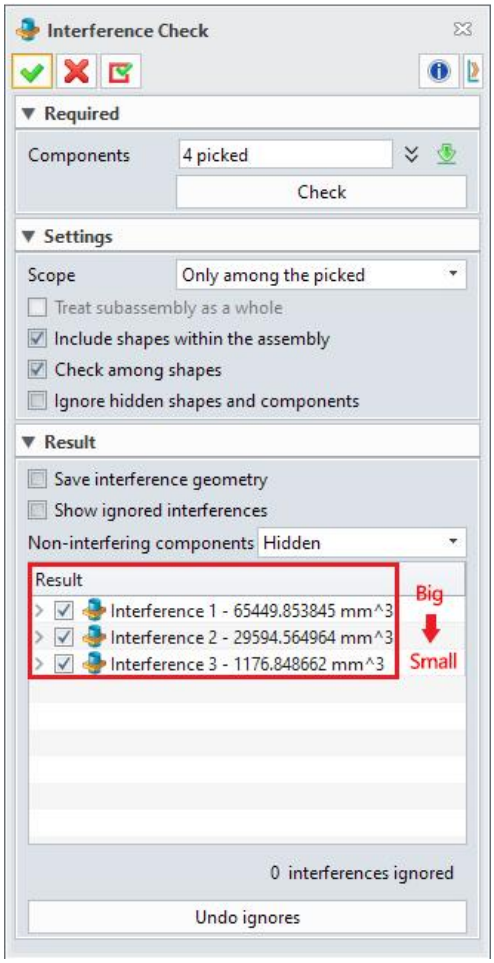
➔ Where it is

Assembly Environment >> Editing >> Copy + Paste

Assembly Environment >> Ctrl C + Ctrl V

3.8.9 Interference Check Result Support Order

The display order of interference check results is changed to sort the size of the interference volume from sort the assembly nodes. Therefore, user can deal with the interference of big volume in priority.

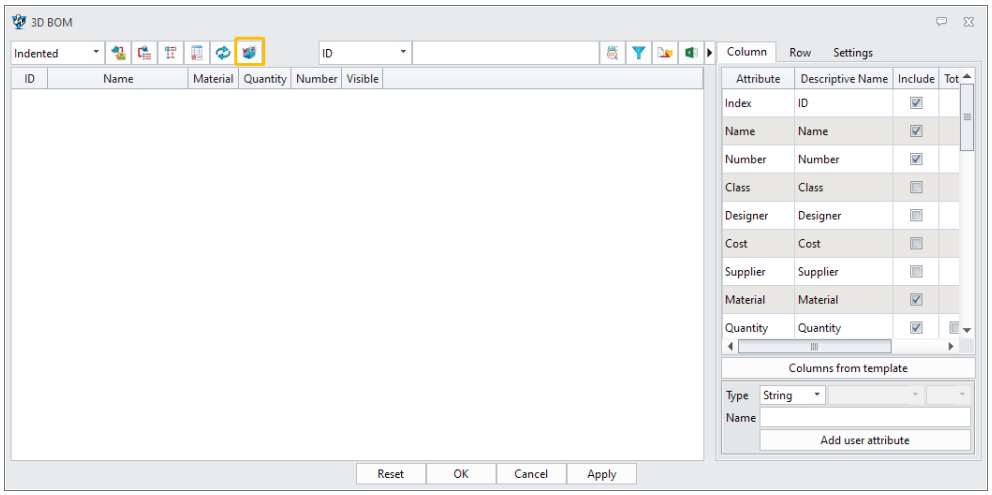


➔ Where it is

Assembly Environment >> Inquire >> Interference Check

3.8.103D BOM Support Ignoring Virtual Components

As for some scenarios, user does not need the virtual components when exporting BOM because user can regard it as a whole and as an assembly. Therefore, we added the option of virtual component excluded in 3D BOM and did not make its BOM calculate the virtual components.

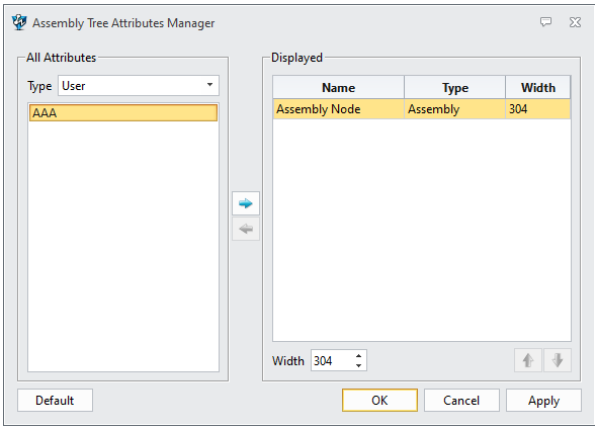


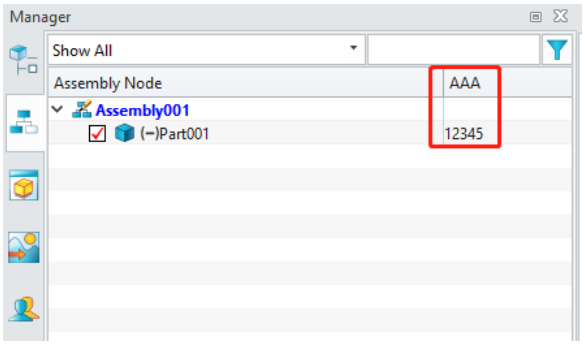
➔ **Where it is**

Assembly Environment >> Inquire >> 3D BOM

3.8.11Assembly Attributes Manager Improvement

The attributes manager supports acquiring the user defined attributes from the part and being displayed on the assembly manager.





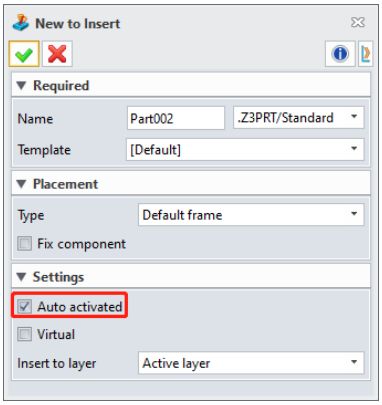
➔ **Where it is**

Assembly Environment >> Manager >> Context Menu >> Attributes Manager

3.8.12New Auto Activated in New to Insert

When inserting new components to assembly, user can set whether automatically activate the new objects or not.

Automatically activate the new components by default. If unchecked the option, the components will not be activated after inserting and still stay in the assembly environment.

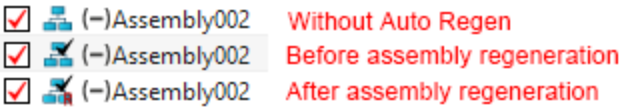


➔ Where it is

Assembly Environment >> Component >> New to Insert

3.8.13Auto Regen Mark Improvement

Distinguish the mark between “Before assembly regeneration” and “After assembly regeneration” in Auto Regen. Use can better distinguish the two auto-regen modes on the assembly nodes. The mark of before regenerating assembly keeps the same as the original while the mark of R will be added at the lower-right after regenerating assembly.

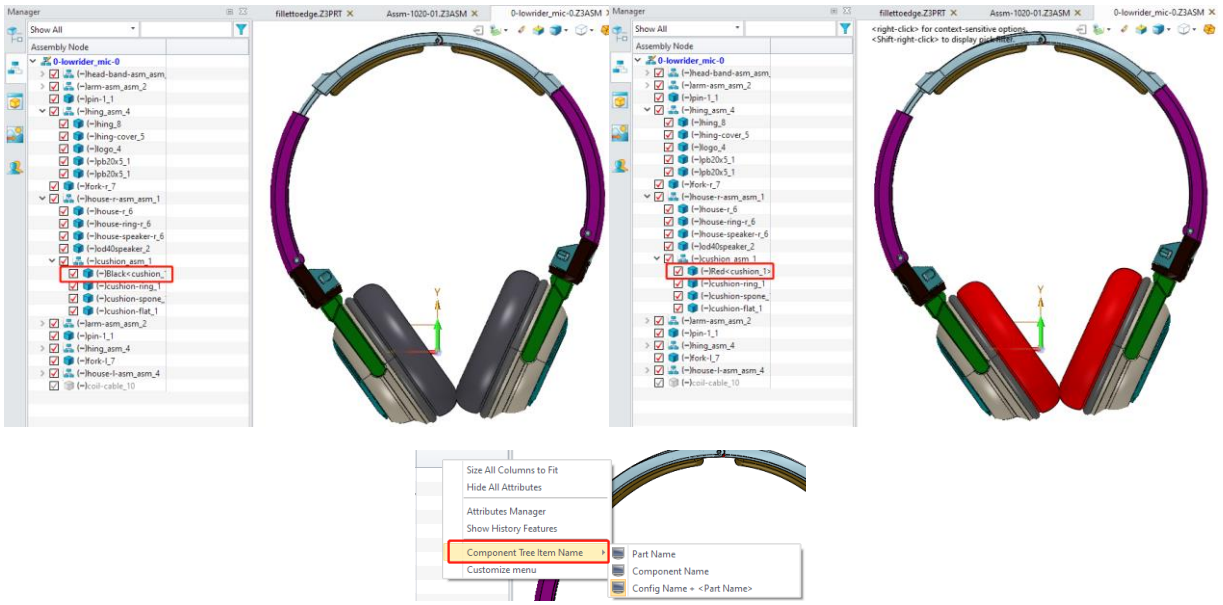


➔ Where it is

Assembly Environment >> Manager >> Assembly Node >> Context Menu >> Auto Regen

3.8.14New Component Tree Item Name

We added the setting of configurating component name to satisfy user’s need to set configuration name as assembly component name when configurating components to assembly.

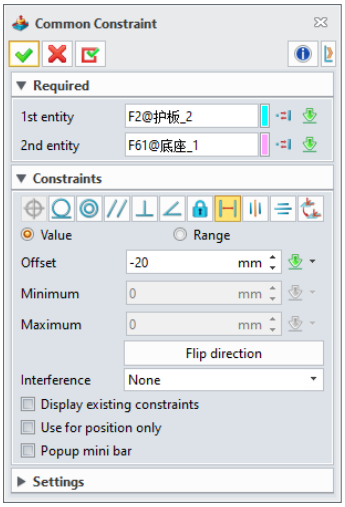


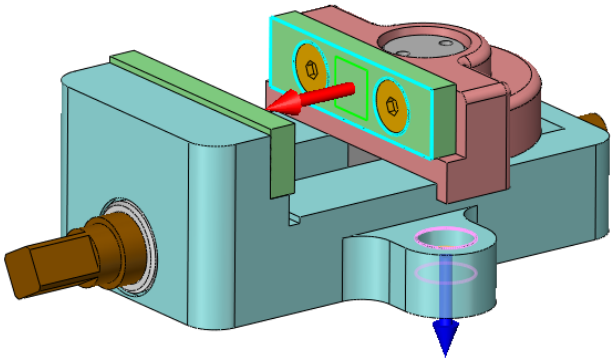
➔ **Where it is**

Assembly Environment >> Assembly Tree Manager >> Context Menu

3.8.15 Distance Constraint Improvement

Distance Constraint supports constraining the distance between the axis of cylinder and the plane to meet more scene application.



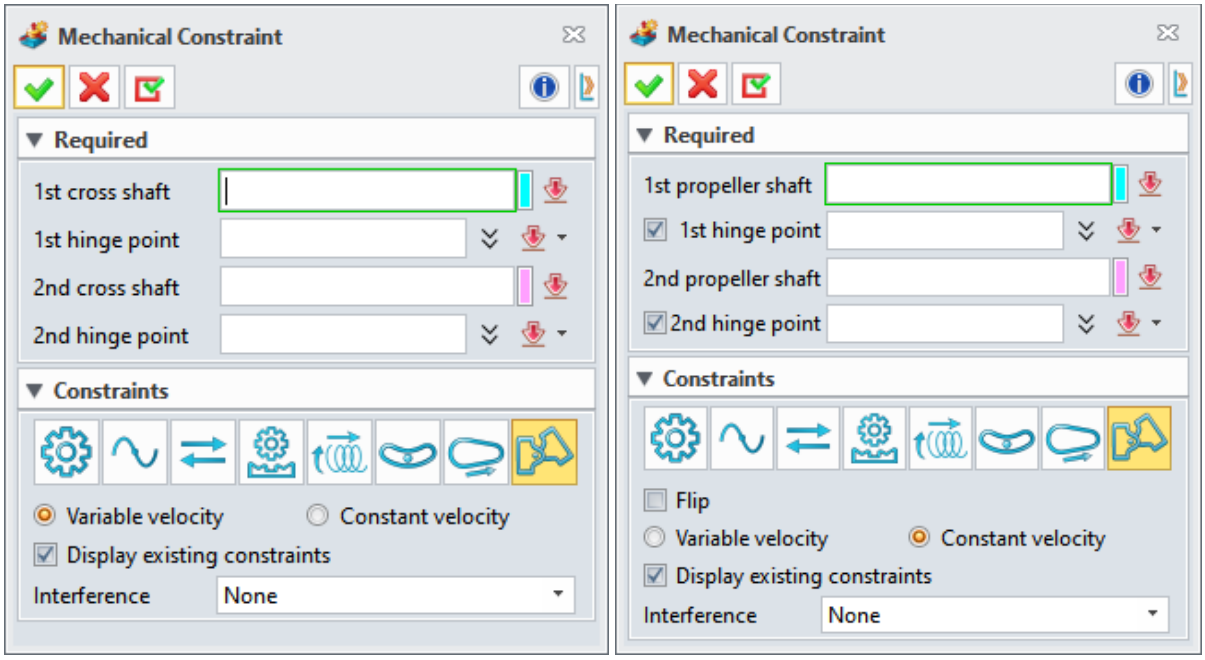


➔ **Where it is**

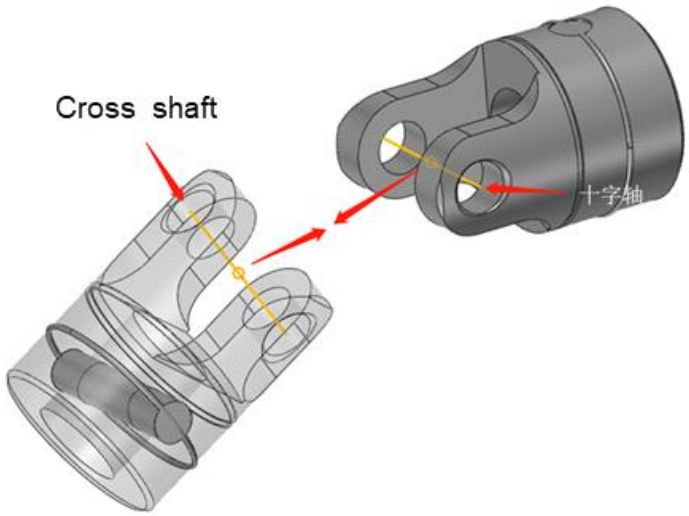
Assembly Environment >> Constraint >> Common Constraint >> Distance

3.8.16 ★New Universal Joint

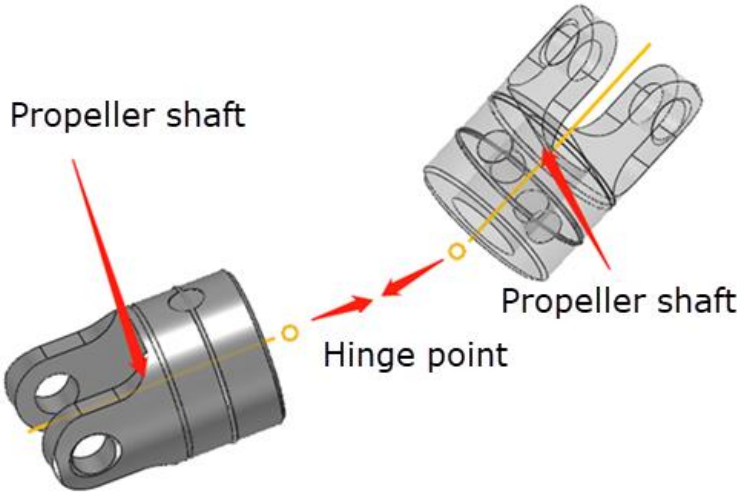
The universal joint constraint can be used to simulate the assembly form with universal joint under variable velocity (cross shaft) or under constant velocity. There are at most four entities to cooperate in the universal joint, two shafts and two points, respectively. The two points are optional inputs under constant velocity. When uncheck the points, user can simulate the retractable constant velocity universal joint. The two points are required inputs under variable velocity and the two shafts can be selected from cylindrical surface, circular arc, and datum axis.



As for variable velocity (cross shaft) universal joint, the selected entity to constrain shall be the interaction of two cross shafts. After selecting, the system will generate universal joint constraint with variable velocity between two parts according to the selected entities.



As for constant velocity universal joint, the selected entities to constrain shall be the hinge joint between the propeller shafts and the universal joint. The hinge points onto the two parts are optional inputs. After selecting, the system will generate constant velocity universal joint between the two parts. If the hinge points are unchecked, there exists retractable freedom of degree on the shaft direction in corresponding parts, which can be used to simulate the retractable universal joint. User can flip the direction of universal joint under constant velocity.

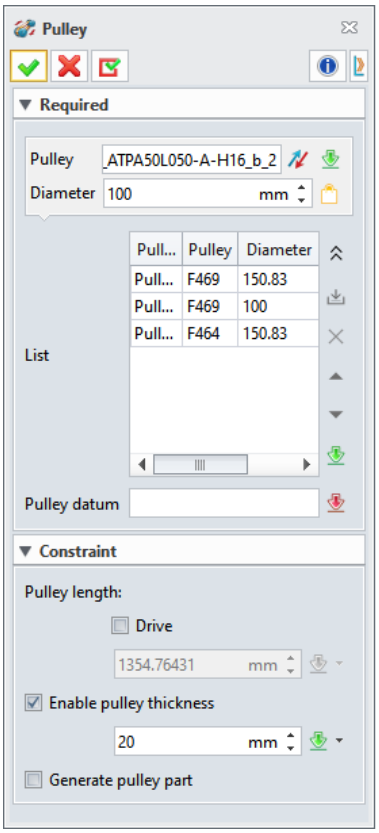


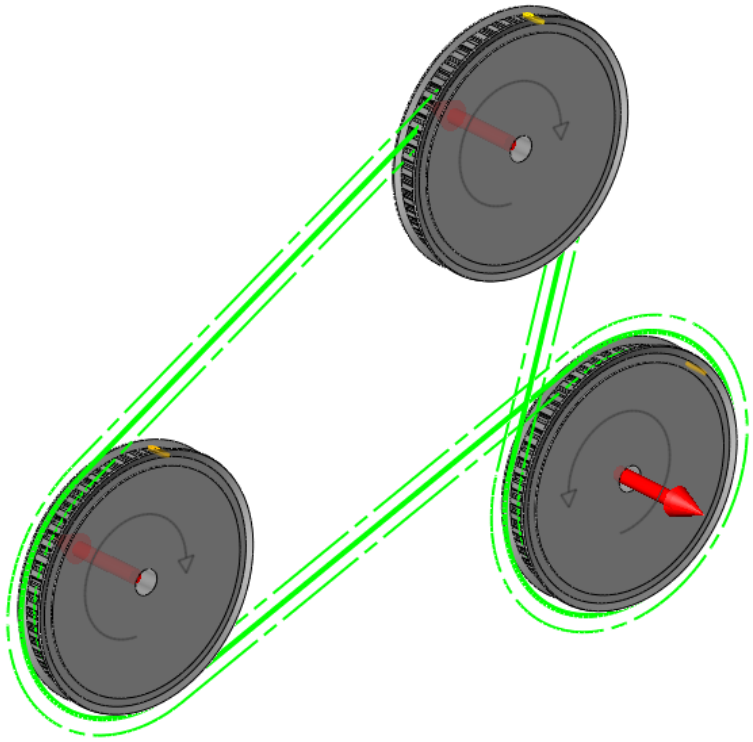
➔ Where it is

Assembly Environment >> Assembly >> Constraint >> Mechanical Constraint >> Universal joint

3.8.17 ★New Pulley Assembly Feature

Use this new command “Pulley” to design pulley components.

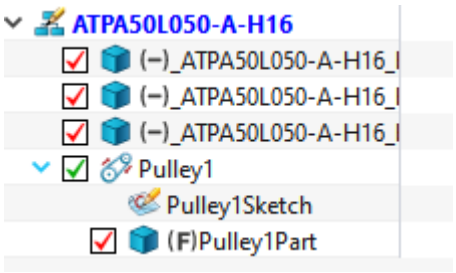




This command can be used to

1) generate a pulley feature.
2) specify a placing plane for pulley datum.
3) manually input pulley diameter
4) adjust the belt order passing through the pulley.
5) change the wrapping direction of pulley.
6) specify the belt length and automatically adjust pulley position.
7) have influence in the position of belt and pulley from the pulley thickness.
8) generate pulley sketch which can be used to calculate the pulley length and pulley modeling.
9) generate the pulley part with pulley sketch.

The pulley sketch and pulley part will be loaded under the pulley feature.



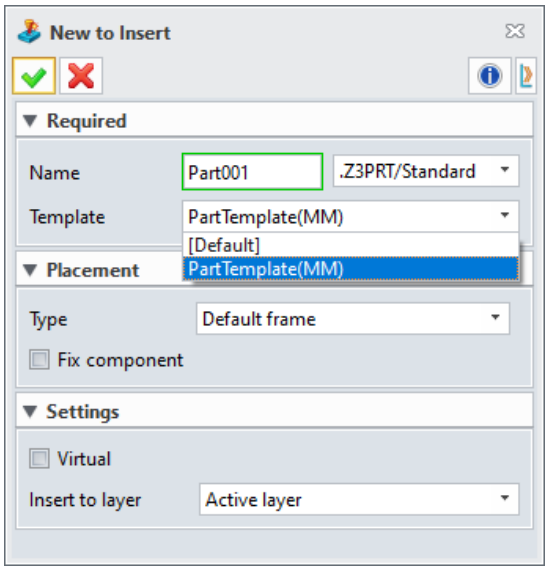
➔ **Where it is**

Assembly Environment >> Assembly >> Constraint >> Pulley

3.8.18 Insert Improvement

3.8.18.1 New to Insert

In the assembly environment, we added “Template” to the command “New to Insert” so that user can select the template type for new component. The data source in the drop-down list is the same as the template of new command.



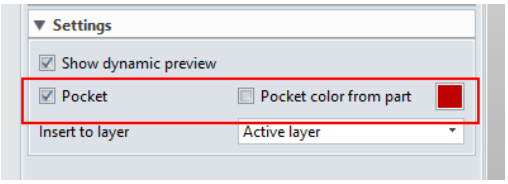
➔ **Where it is**

Assembly Environment >> Assembly >> Component >> New to Insert

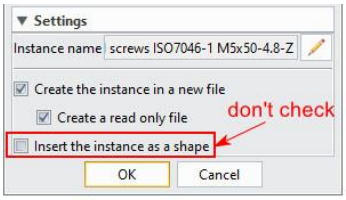
3.8.18.2 Standard Part with Pocket Attribute

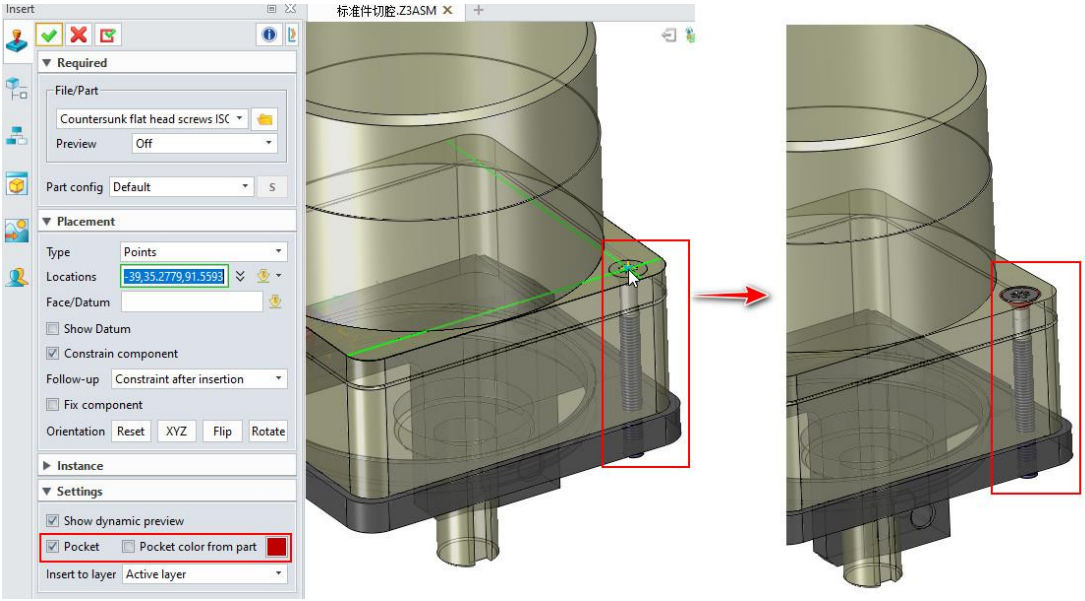
We added “Pocket” to the standard part “Insert” panel. When the standard part’s pocket is defined pocket attribute, after inserting the standard part, the pocket feature will be inserted in the pocketed part

(but no corresponding feature in the assembly tree so unable to be redefined). We also added the “Pocket color from part” option so that user can customize color or use color from part.



See the following figure for details. When the “Pocket” is checked, the corresponding pocket feature will be generated on the pocketed part. Note: only when the standard part inserts as a component can the function of pocket can be used.





There exists several characteristics in standard part and pocket feature:

1)The pocket feature is generated as original model and associates with inserting standard instance.

2)When the associated standard instance is moved, the corresponding pocket feature should be updated.

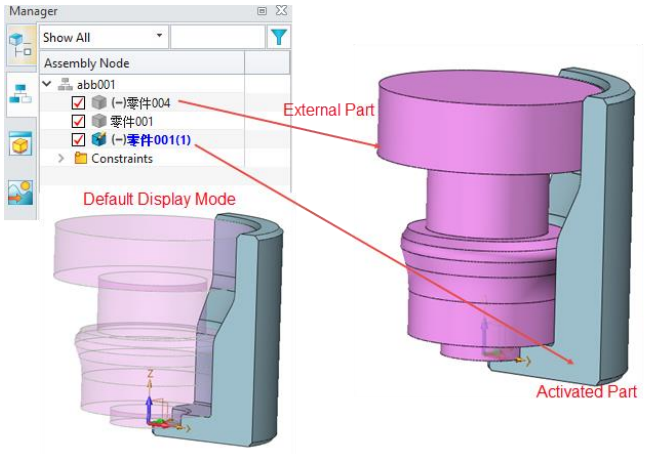
3)When the associated instance is deleted, the corresponding pocket feature should be deleted.

➔ **Where it is**

Assembly Environment >> Assembly >> Component >> Insert

3.8.19 New Shade Mode in Display OutofScope Parts

When Shade mode is selected in “Display OutofScope parts”, the excluding parts of the activated part will be displayed in shade mode during the in-place editing.

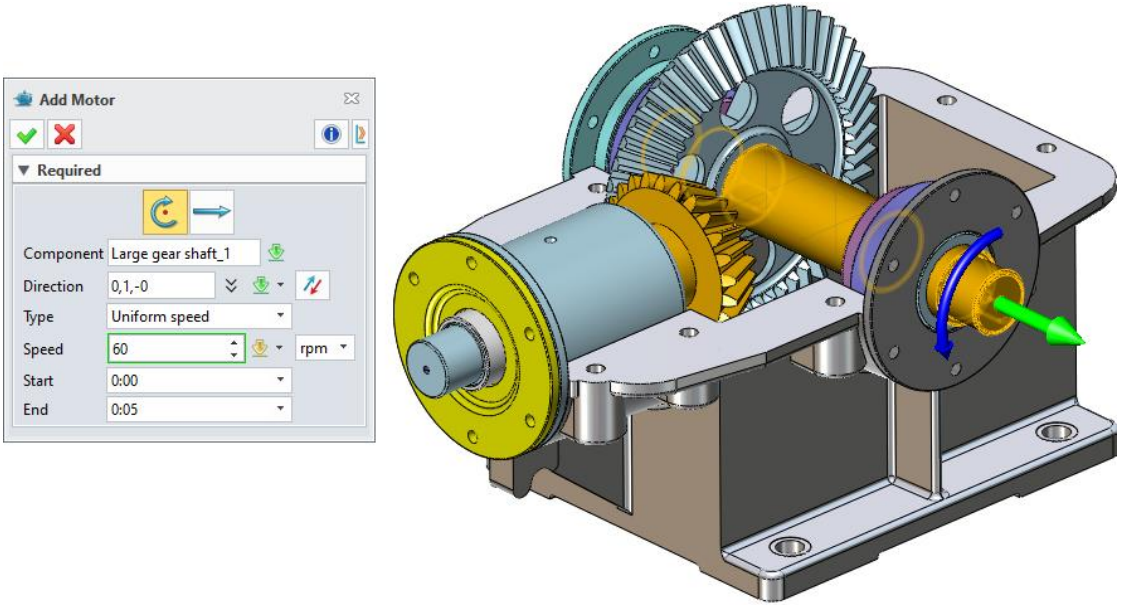


➔ **Where it is**

Configuration >> Display >> Display OutofScope parts >> Shade

3.8.20 ★New Motor in Assembly Animation

We added “**Motor**” to the assembly animation in ZW3D 2023. In movement simulation, the motor provides the source of original powder to the assembly. User can define speed and direction and the whole assembly will simulate the movement under the powder. We supported two motors: **Rotary motor** and **Linear motor**.

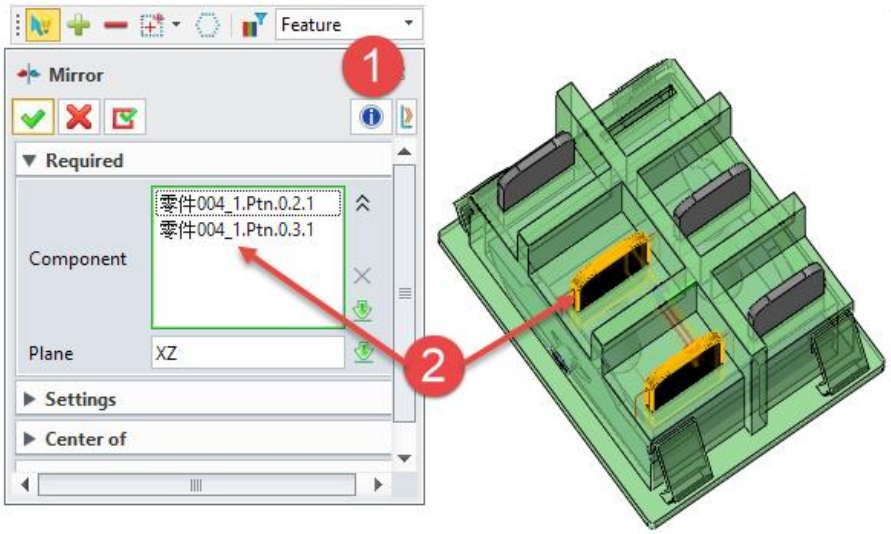


➔ **Where it is**

Assembly Environment >> Animation >> Animation >> Add Motor

3.8.21 Assembly Pattern/Mirror Improvement

Assembly pattern/mirror supports patterning assembly features (such as mirror assembly and pattern assembly) which is faster than picking a component one by one.



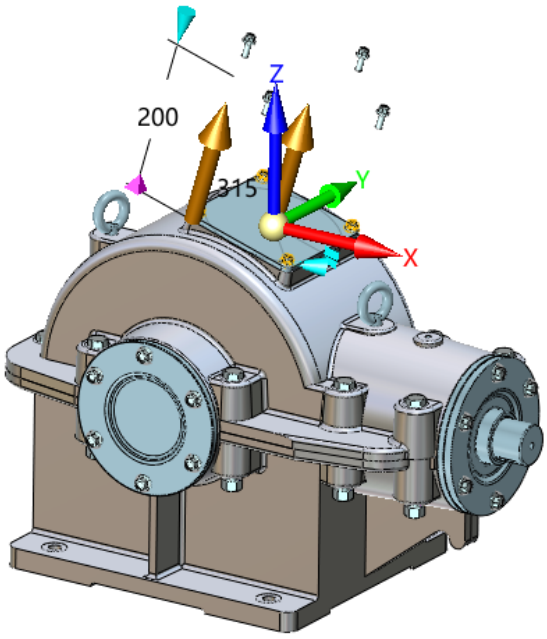
➔ **Where it is**

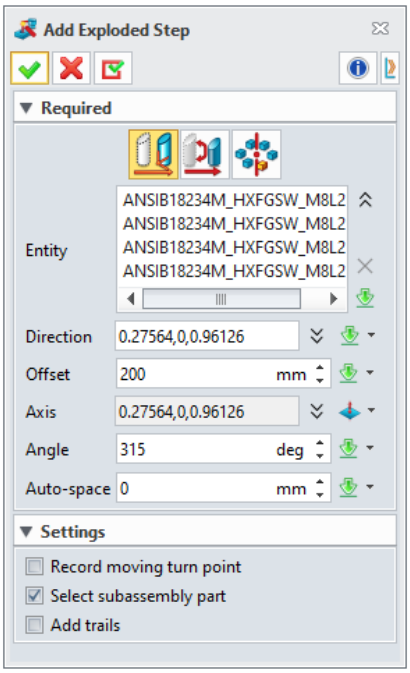
Assembly Environment >> Assembly >> Basic Editing >> Patten/Mirror

3.8.22 Exploded View Improvement

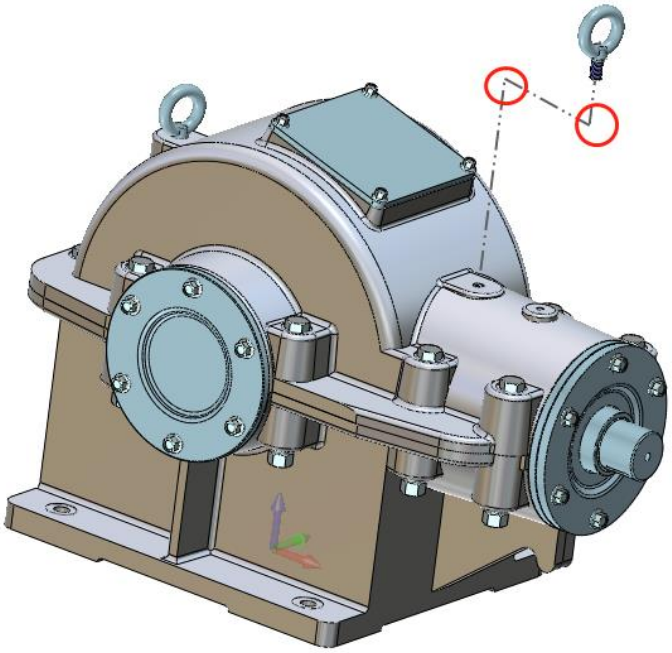
3.8.22.1 Add Exploded Step Improvement

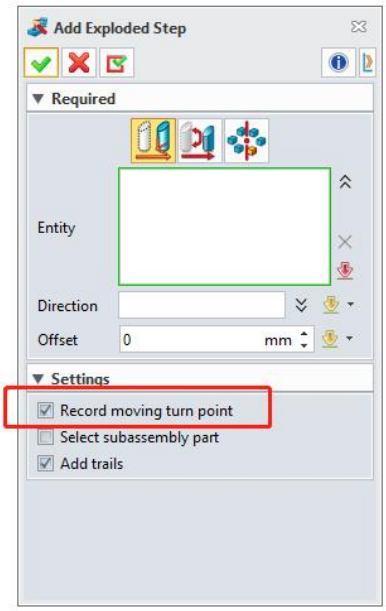
We added “Move exploded” “Rotate exploded” and “Radial exploded” in “Add Exploded Step” command to meet user’s requirement of more complex exploded view including complicated path, display path and entirely exploded of sub-assembly. Meanwhile, we added “Record moving turn point” and “Add trails” to enrich the exploded path. Besides, the option of “Select subassembly part” has achieved different exploded needs.



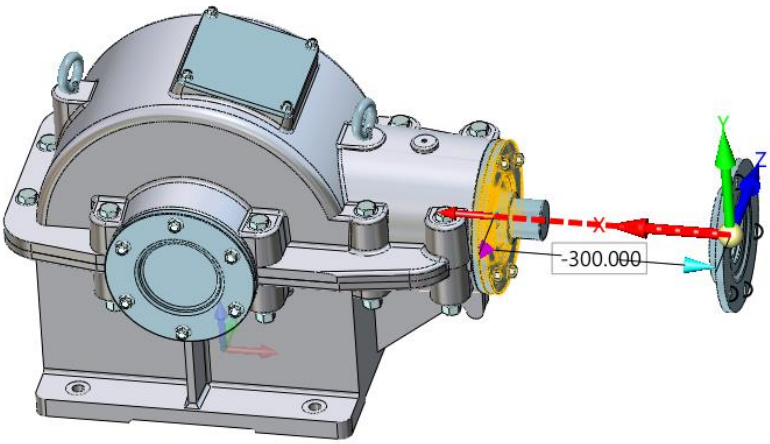


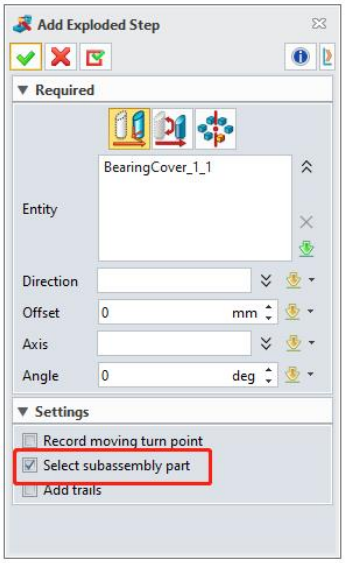
- Check “**Record moving turn point**” to record the current position, and then perform exploded steps in other directions. The final one will be a complex and continuously multiple exploded paths.

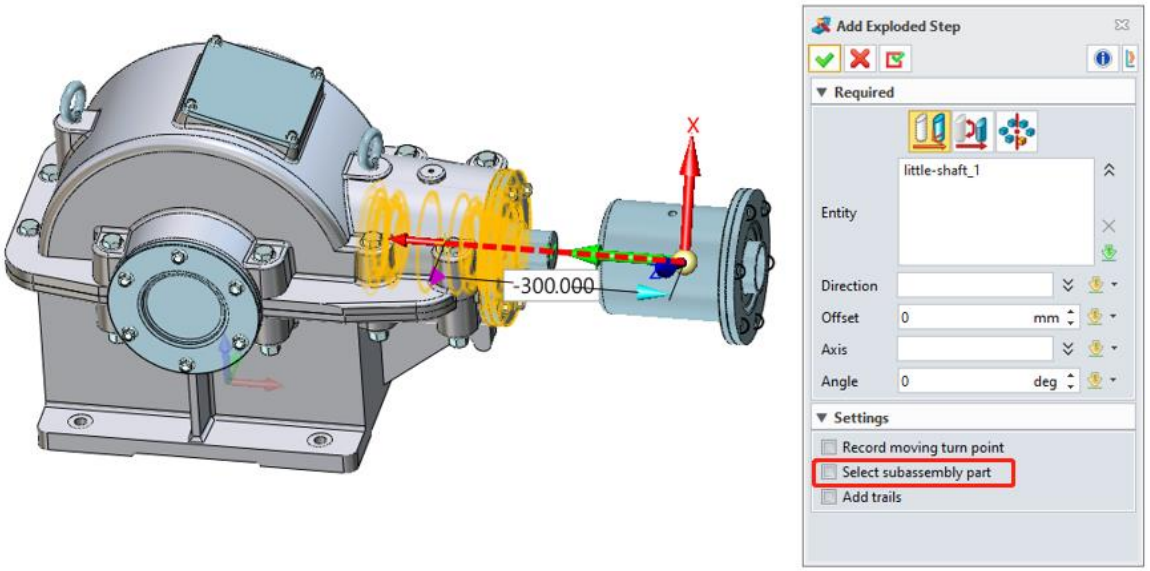




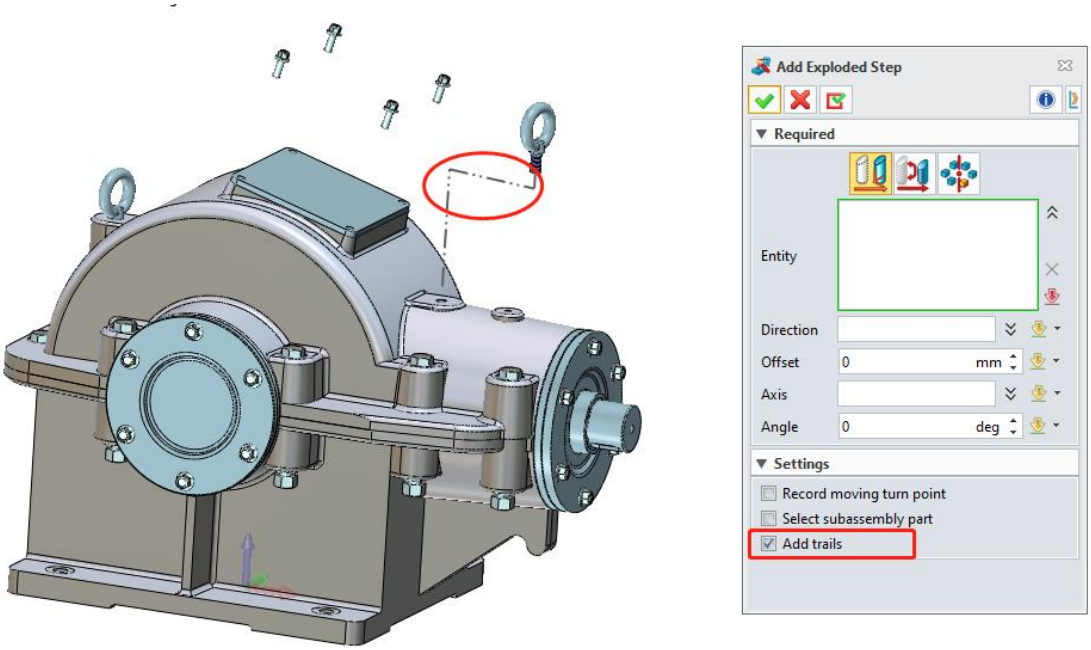
- Check "**Select subassembly part**" to explode the entire subassembly of the currently selected object, which can satisfy the request of the whole assembly explosion and finally get the designed explosion result.







- Check "**Add trails**" option to record exploded trail with double dots line while creating an exploded view.

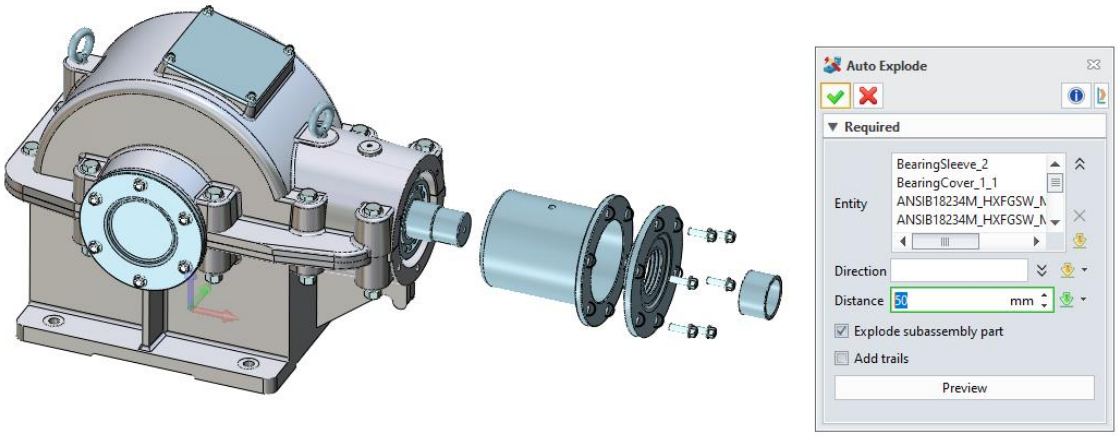


➔ Where it is

Assembly Environment >> Exploded View >> Add Exploded Step

3.8.22.2 New Auto Explode

We added “Auto Explode” to increase the efficiency of creating exploded view, which achieves to create batch component exploded view in uniform. The options “Explode subassembly part” and “Add trails” can achieve different exploded needs.

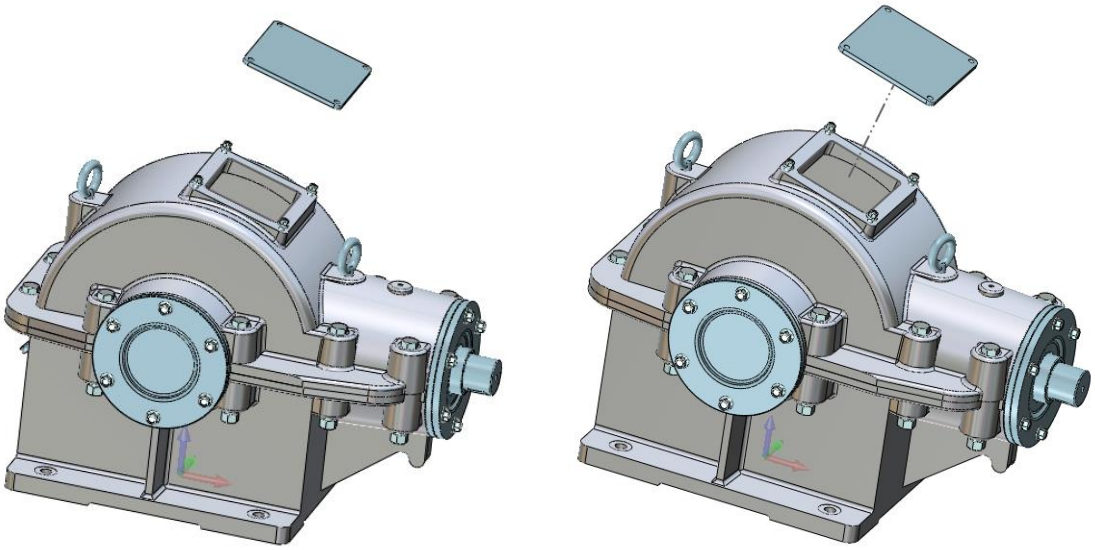


➔ Where it is

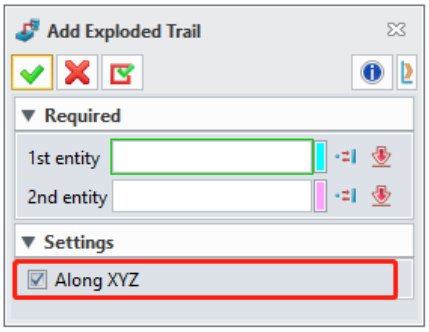
Assembly Environment >> Exploded View >> Auto Explode

3.8.22.3 New Add Exploded Trail

To solve the problem that user did not record exploded trial and continued to add exploded trial in follow-up, ZW3D can manually add exploded trial in exploded view through defining start and end position so to reach exploded trail creation.



- If the option "**Along XYZ**" is checked, the created trail shall be along the axis of the orthogonal.



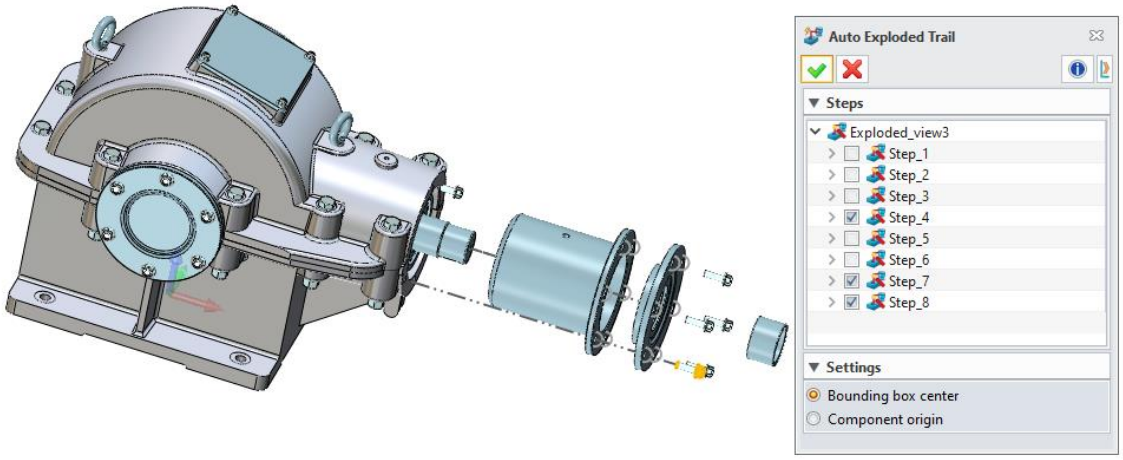
➔ **Where it is**

Assembly Environment >> Exploded View >> Add Exploded Trail

3.8.22.4 New Auto Exploded Trail

We added "Auto Exploded Trail' to improve the efficiency of exploded trail creating. In addition to batch create exploded trails, the function also provides the method to select the start and end of the trial, which achieves the addition to different exploded trails.

- The trail can set the bounding box center.
- The trail can set the component origin.



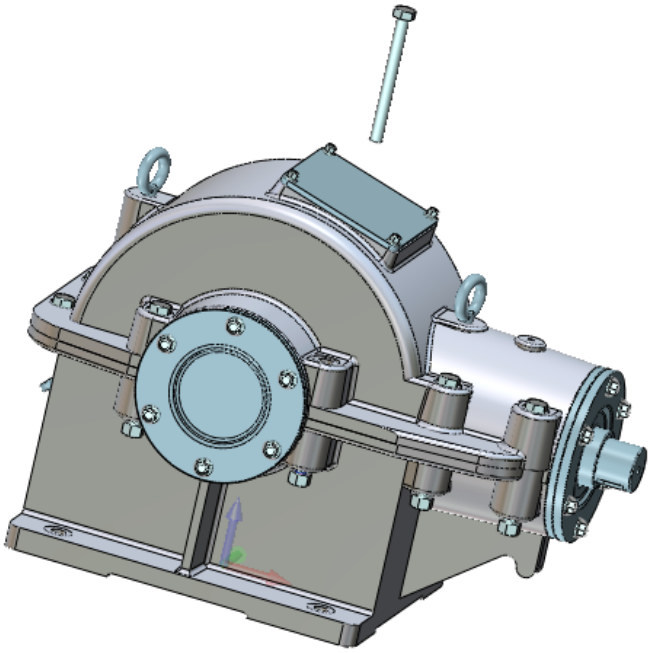
➔ **Where it is**

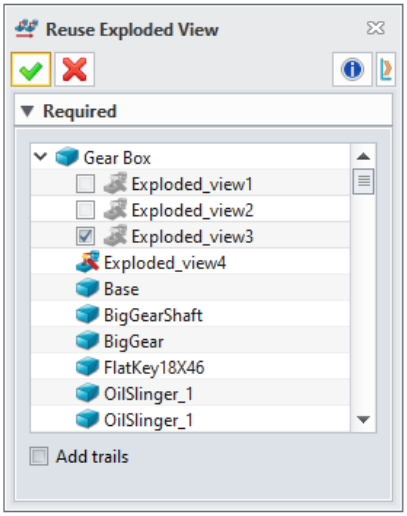
Assembly Environment >> Exploded View >> Auto Exploded Trail

3.8.22.5 New Reuse Exploded View

We added “Reuse Exploded View” function to raise the efficiency of multiple exploded view. The created exploded view should be applied in the current exploded view, which meet the combined application of complex exploded view.

- If the option “Add trails” is checked, the reused exploded view will carry with the exploded trails.



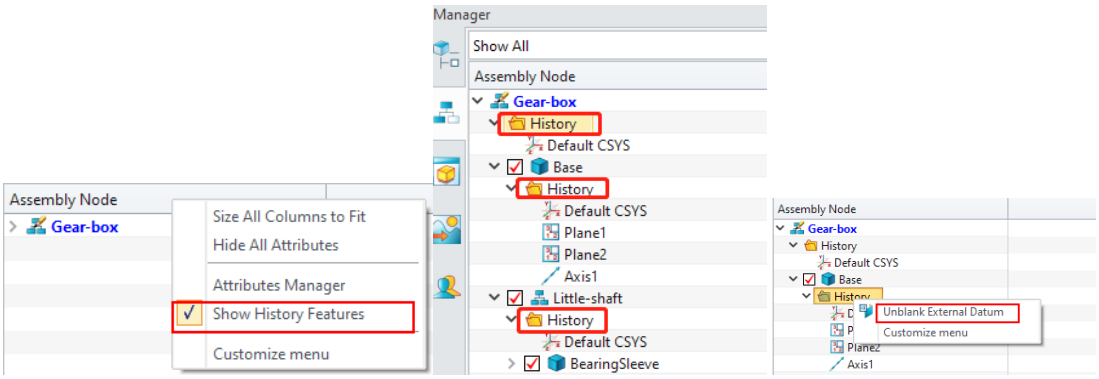


➔ Where it is

Assembly Environment >> Exploded View >> Reuse Exploded View

3.8.23 New Show History Features

We added “Show History Features” in assembly three where displayed the history nodes of assembly and part, to solve the issue that some history node was utilized in complex assembly but difficult to be selected quickly and precisely. The assembly history node will be displayed in the assembly level while the part history node in the part level. The datum objects including datum axes, datum planes and coordinates are currently supported. User can enable “Show History Features” by right-clicking on the assembly tree node and show/hide datum objects in the drawing area by right-clicking history feature node.



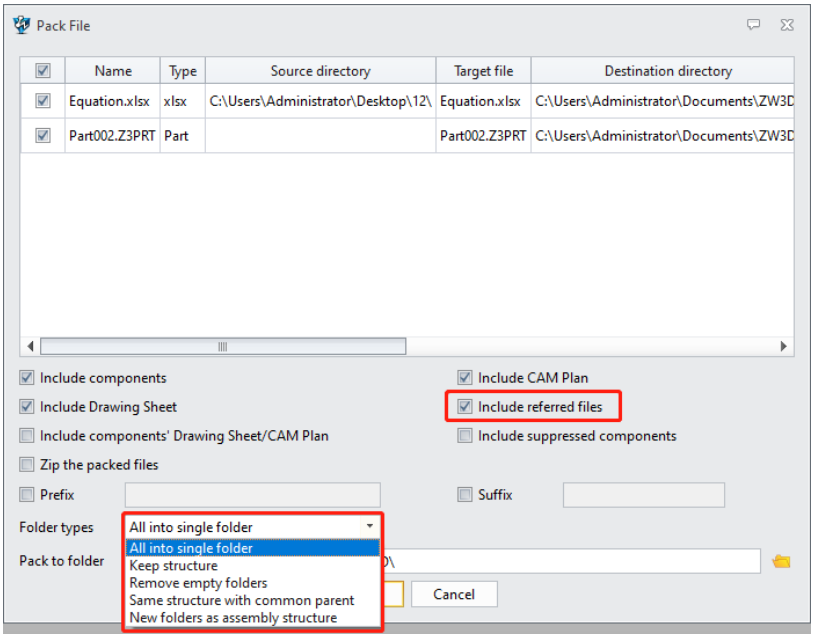
➔ **Where it is**

Assembly Environment >> Manager >> Assembly Node >>Context Menu >> Show History Features

3.8.24 Pack Improvement

We added “New folders as assembly structure” as a folder type in the command “Pack”. The folder type supports both creating folder directory and saving files according to the assembly level relationship.

When the packed file exists link to excel, “Include referred files” is checked, and then the excel files can be found and packed together.



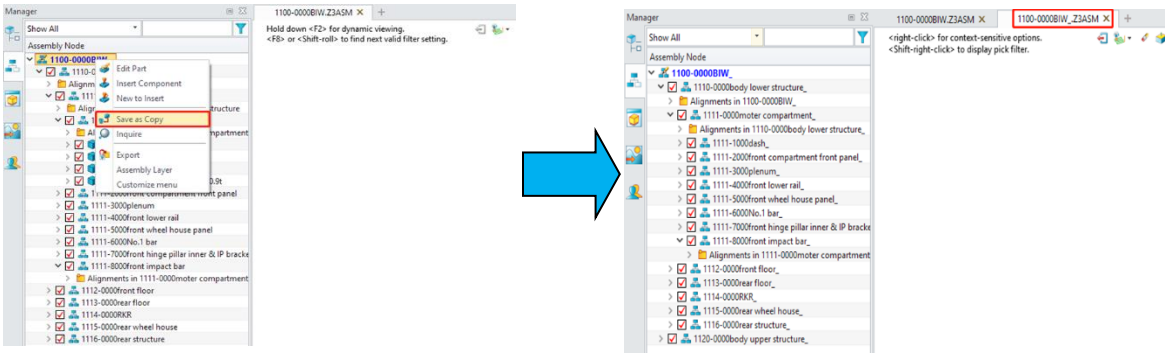
➔ **Where it is**

Assembly Environment >> File >> Pack

3.8.25 New Save as Copy

The original “Clone” function has been upgraded to “Save as Copy” to support the design operation that simply modify newly generated Component B on the base of original Component A when there are many similar structure components in the assembly.

- As the general assembly is upgraded from “Clone” to “Save as Copy”, the original assembly window will not be closed so that user can compare with the two models.



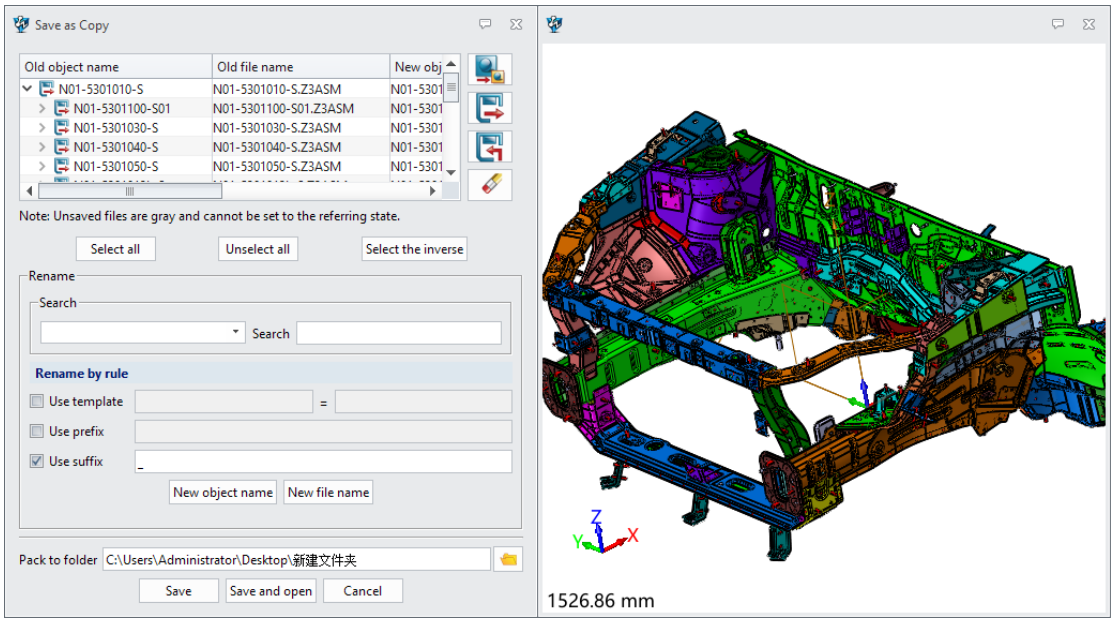
After enabling “Save as Copy”, there will popup windows like the renaming manager, where user can label components as three status: Change to **Save-as-copy state, Change to Referring state**, and **Change to Remove state**.

Change to save-as-copy state : Refer to create a copy file in the new path. Note that the general assembly node can only be “Save-as-copy state”.

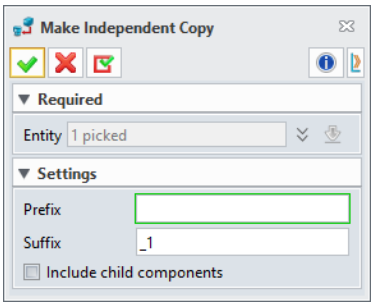
Change to Referring state : Refer to not create a copy file in the new path but the parent assembly of copy calls the file of the sourced file.

Change to Remove state : Refer to not create a copy file in the new path and move the file from the copy assembly.

Use the small window to preview the generated general assembly copay, as the following figure shows:



- As part is upgraded from “Clone” to “Make Independent Copy”, in addition to support “Clone”, “Make Independent Copy” also supports multiple same components generating independent copies at the same time as well as withdrawing operation.



➔ Where it is

File >> Save as Copy

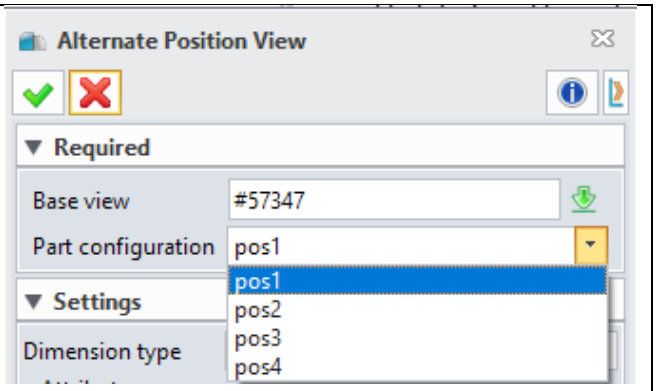
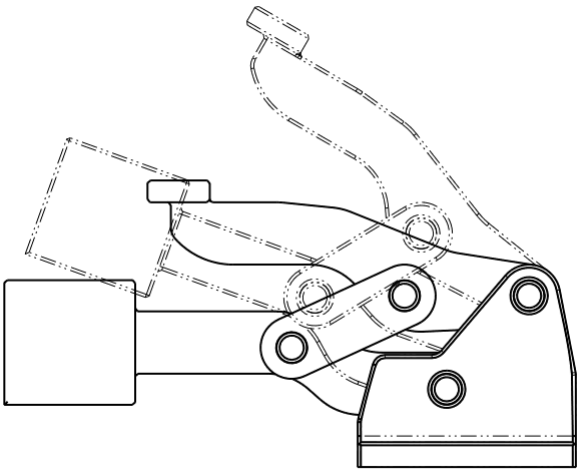
Assembly Environment >> General Assembly Context Menu >> Save as Copy

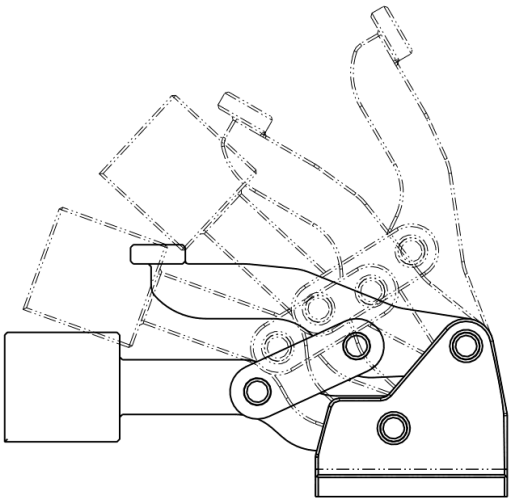
Assembly Environment >> Component Context Menu >> Make Independent Copy

3.9 Drawing Sheet Design

3.9.1 New Alternate Position View

The alternate position views are created based on the main views to present the component status in different configuration in such way. In general, the alternate position views are applied when there are multiple configurations in the assembly. In one main view, one configuration can only create one alternate position view and it cannot be created repeatedly.

Insert alternate position view Select base view and part configuration	
Generate alternate position view	

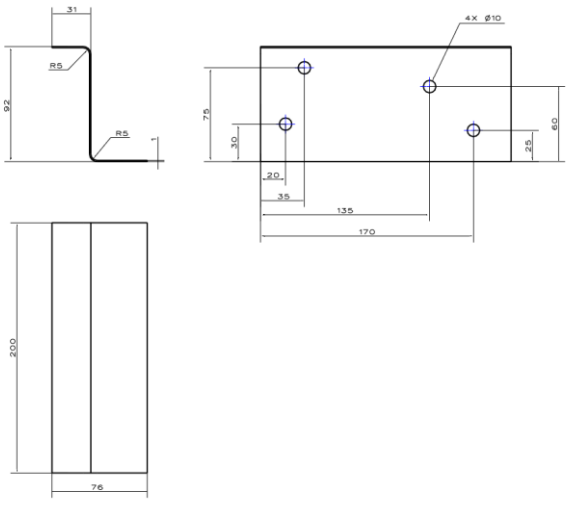
Generate multiple alternate position views	

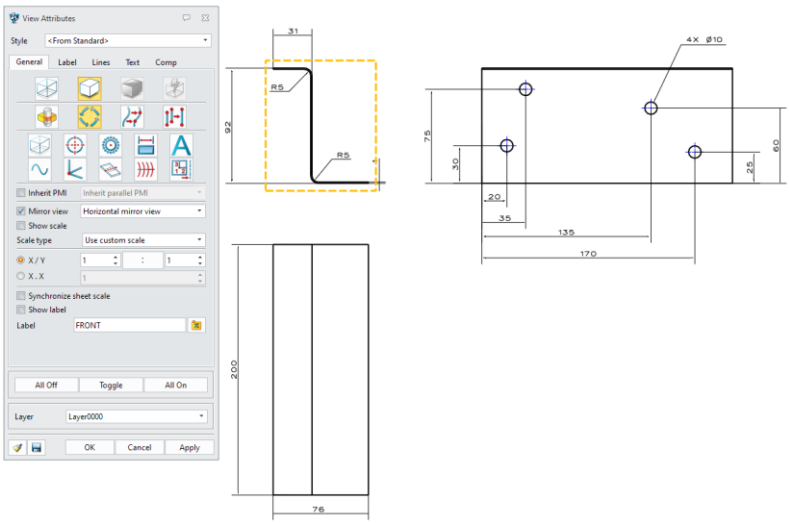
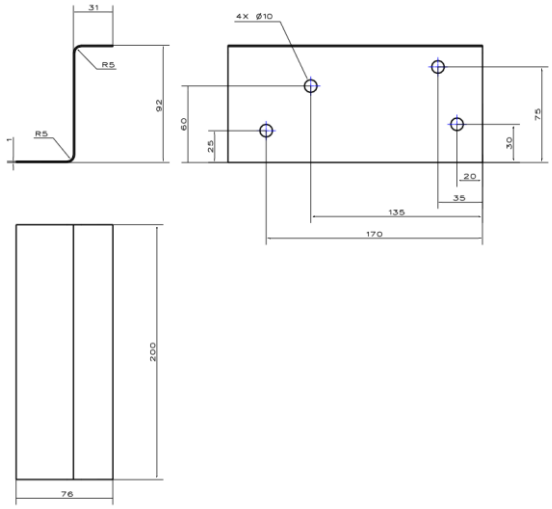
➔ **Where it is**

Drawing Sheet Environment >> Layout >> Alternate Position View

3.9.2 New Mirror View

We added the mirror view so that the user can quickly acquire the mirror design drawing sheet of the current components. In View Attributes panel of standard view, user can check the option “Mirror view” and immediately transfer the original view to the mirror view.

Drawing sheet of original design model	

Select main view and open attributes, Check Mirror view	
Mirror design drawing sheet Dimensions are normally inherited	

➔ **Where it is**

Drawing Sheet Environment >> View Attributes >> Mirror View

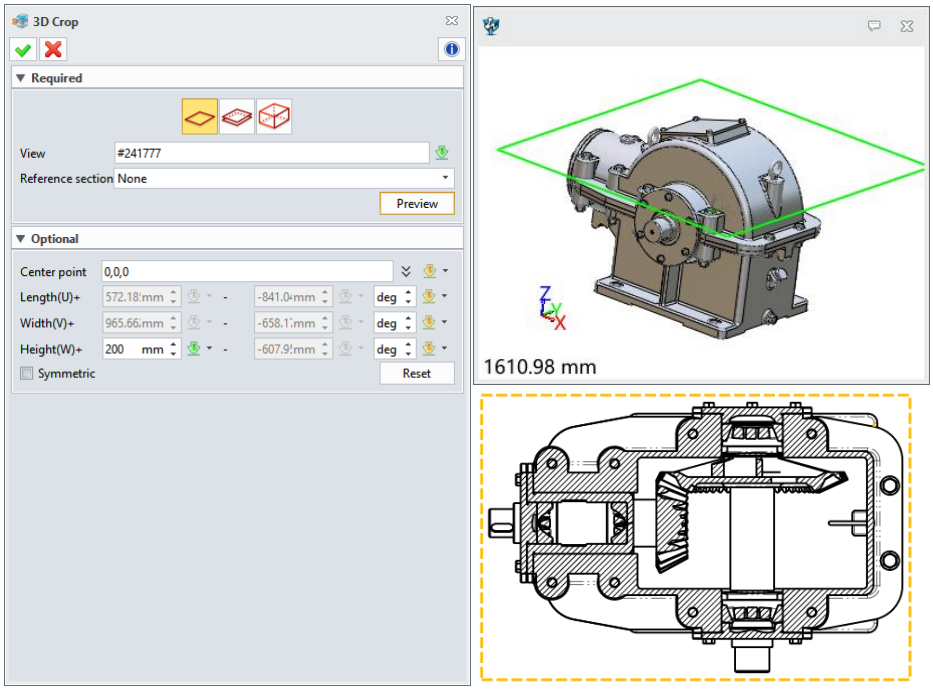
3.9.3 ★New 3D Crop

As for the engineering design of big assembly, it often needs to procure drawings by regions or by division of work or procure detailed layout sheet of partial space when design drawing sheet. Therefore, we added 3D Crop to limit the projection region of the drawing sheet and control. By changing the rectangle cropping position and size, the project control can be realized in any region of the model.

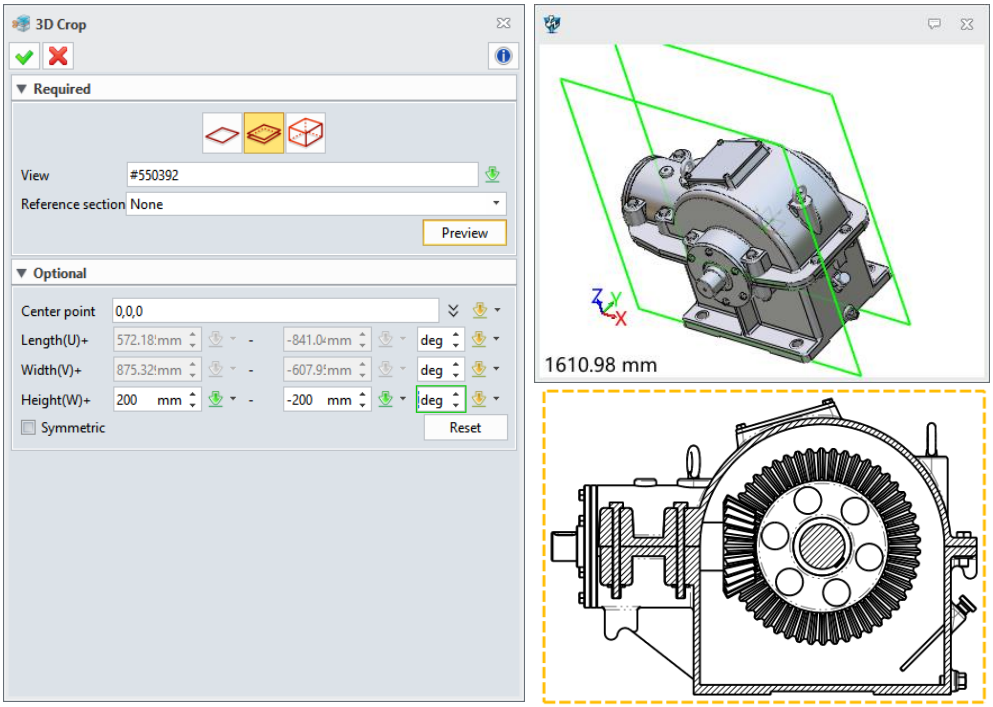
3.9.3.1 3D Crop

We added 3D Crop to the drawing sheet environment, which supported setting 3D crop section based on the generated views and achieved generating drawing sheets of modeling by picking section. There are three ways to 3D crop including Section at plane, Section with slice, and Section with envelope.

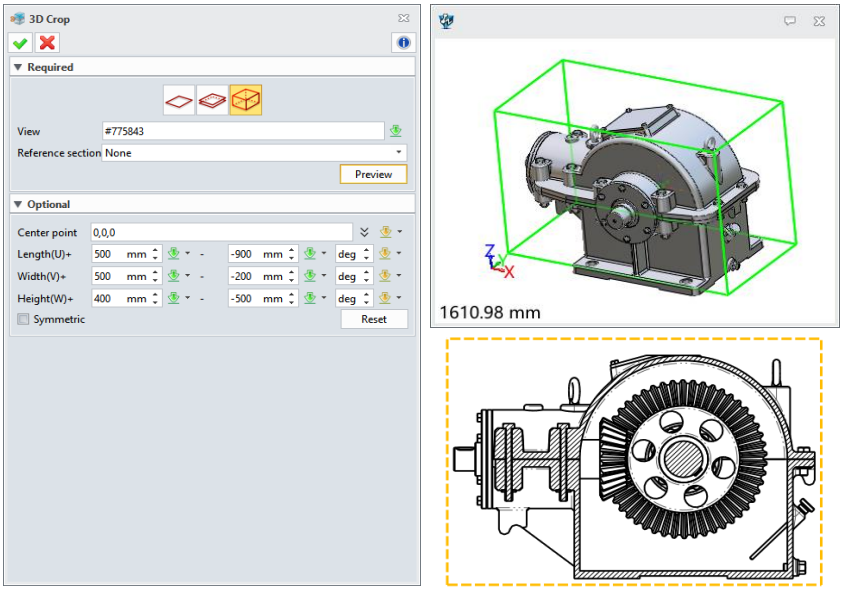
Section at plane: Use a plane to divide a 3D space for projection. Set the positive parameter direction through the current view and select cropping position.



Section with slice: Use two planes to divide a piece of 3D space for projection view. Set positive and negative direction parameters of the current view and select the cropping section.



Section with envelop: Use six planes to layout a piece of 3D region for projection. Set positive and negative direction parameters through X, Y, Z axes.



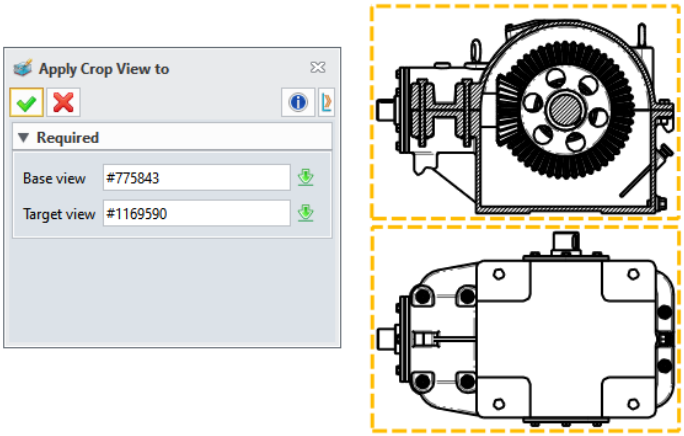
During the cropping region parameters, user can preview the current model and region parameter settings through preview. The operations of mover, rotate, zoom in and out are the same as the system. Besides, user can acquire the region parameter of reference view through selecting reference view.

➔ Where it is

Drawing Sheet Environment >> Layout >> View >> 3D Crop

3.9.3.2 3D Crop Apply Crop View to Specific View

User can quickly set the region parameter of 3D crop to the specific view. After the command is activated, select the base view and target view. If confirmed, the target view will generate new crop view according to the base view of 3D crop region.

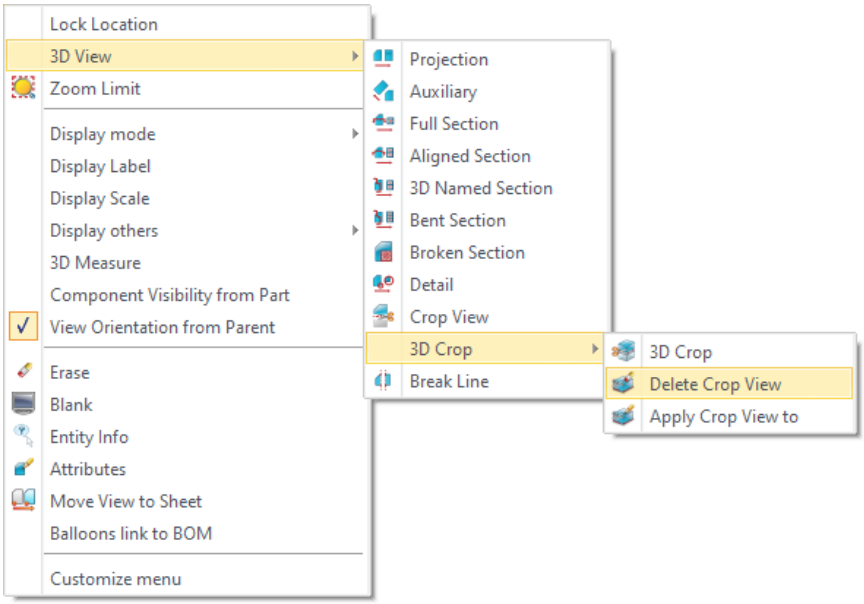


➔ **Where it is**

Drawing Sheet Environment >> Layout >> Edit View >> Apply Crop View to

3.9.3.3 Delete 3D Crop View

After selecting 3D crop view, user can delete the 3D crop view by clicking the context menu. When deleting the parent view of 3D crop, the tip of whether delete the sub view of 3D crop will pop-up. When deleting the sub view of 3D crop, only the current sub view of the 3D crop can be deleted.



➔ **Where it is**

Drawing Sheet Environment >> 3D Crop View Context Menu >> 3D View >> 3D Crop >> Delete Crop View

3.9.4 ★View Projection Efficiency Improvement

We optimized the inside algorithm logics in depth and improved the view creation flow in ZW3D 2023. Thus, the generation speed of projection view increased apparently. The lager file size, the more significant promotion speed.

The below table shows the testing case data. The average speed increased 45%.

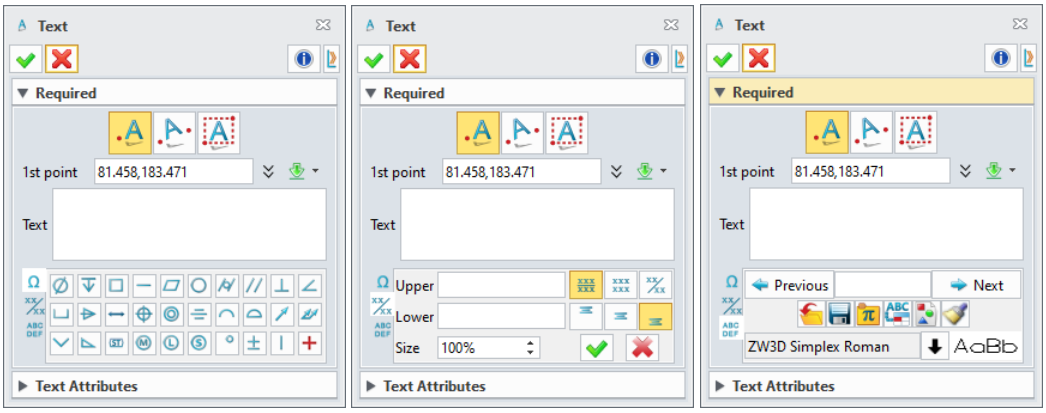
File size	101 MB	214 MB	385 MB	494 MB	850 MB	990 MB	2.11 GB
Speed Increase Rate	16.24%	17.85%	15.43%	20.84%	54.81%	95.52%	94.25%

➔ Where it is

Drawing Sheet Environment >> Layout >> Projection View

3.9.5 Text Editor Improvement

We embedded the text editor was embedding into the panel of each function in ZW3D 2023, where user could use the text editor directly after opening the interface of each function and not need to open the secondary command operation.

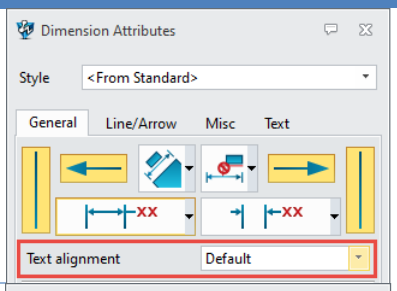
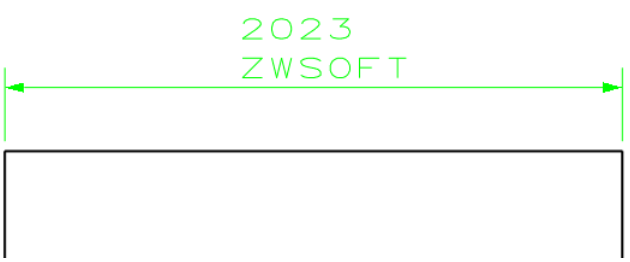
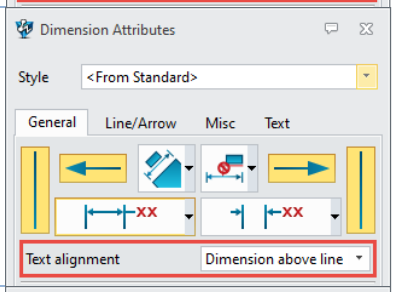
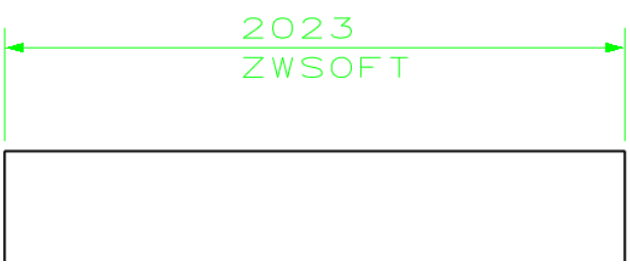
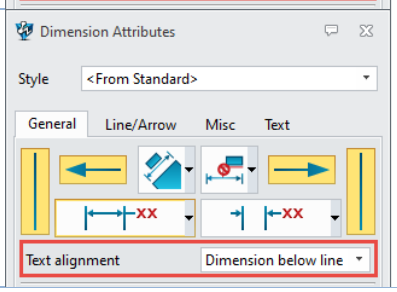
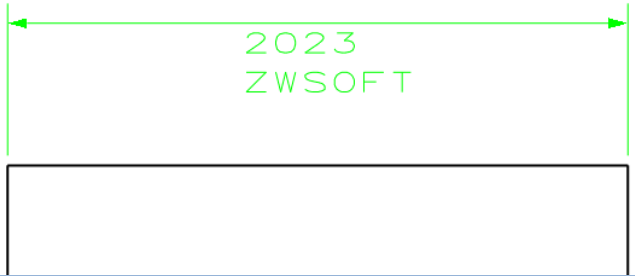


➔ Where it is

Drawing Sheet Environment >> Drawing >> Text

3.9.6 New Text Alignment

We added the drop-drown list of “Text alignment” to the dimension attributes panel, including Default, Dimension above line, and Dimension below line.

Type	Example
Default	 
Dimension above line	 
Dimension below line	 

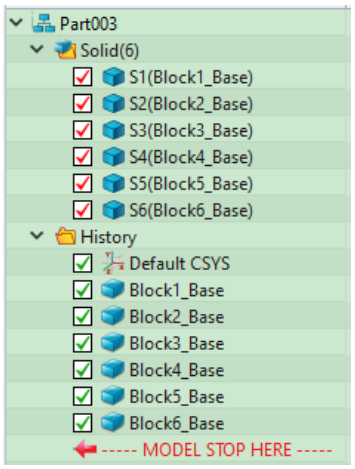
➔ Where it is

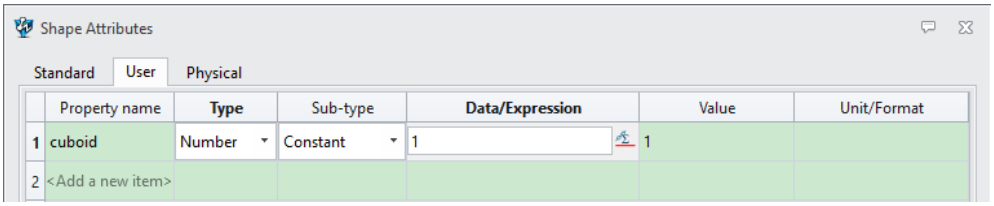
Drawing Sheet Environment >> Dimension Context Menu >> Attributes

3.9.7 2D BOM Support Solid Attribute Filter

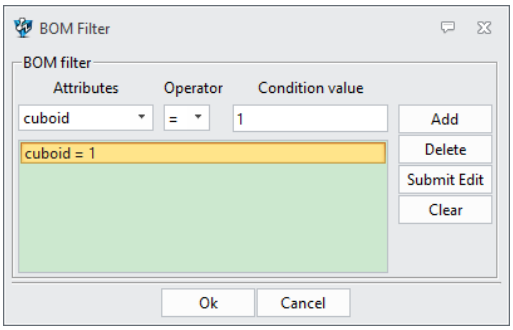
In the environment of single part and multiple entities, the entity objects are grouped by setting attribute markers, and the attributes of the entity are filtered by using BOM filter in engineering drawings, thus generating BOM as a group.

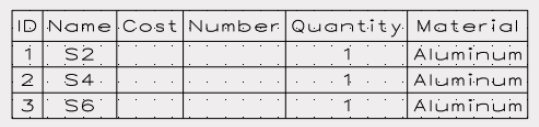
The first following figure set custom attributes for Solid 2, 4, 6: cuboid=1.





When generating the engineering sheets, the option “Show shape” in Model of setting should be checked so that BOM filter could read the custom attributes of solids. As the following figure shown, it used “cuboid=1” attribute to combine Solid 2, 4, 6 to generate BOM.





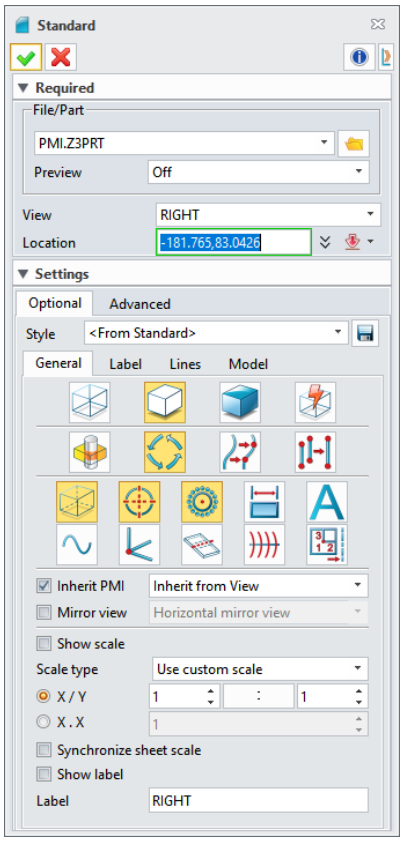
ID	Name	Cost	Number	Quantity	Material
1	S2			1	Aluminum
2	S4			1	Aluminum
3	S6			1	Aluminum

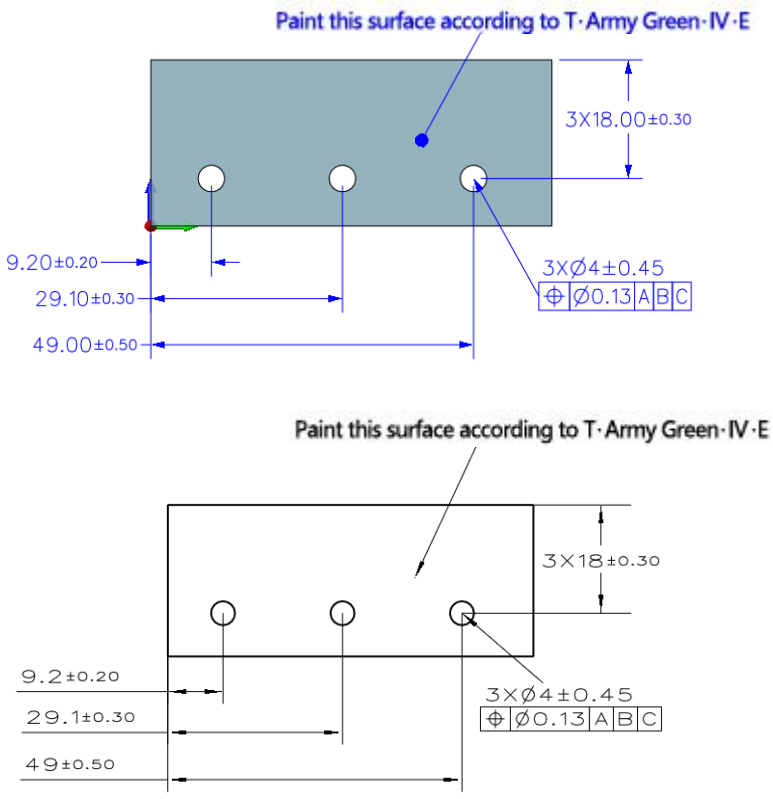
➔ Where it is

Drawing Sheet Environment >> Layout >> Table >>BOM

3.9.8 New Inherit from View

When creating base view and projection view, a new method “Inherit from View” was added to only inherit the PMI dimension information under the view.



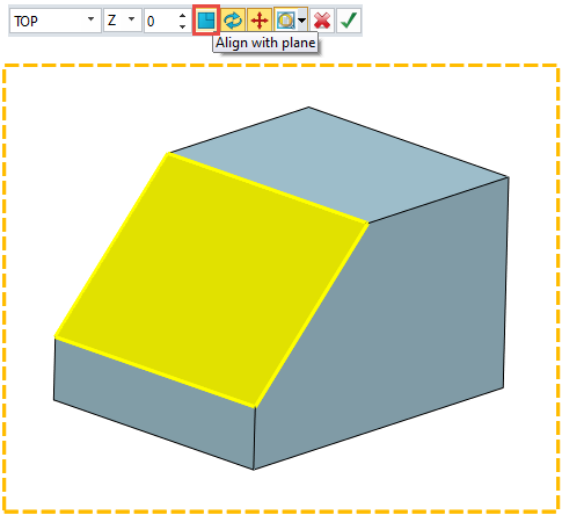


➔ Where it is

Drawing Sheet Environment >> Layout >> View >> Standard

3.9.9 New Align with Plane in Rotate View

We added “Align with plane” to the command “Rotate View” in ZW3D 2023, which can quickly locate view in the engineering drawing and align with a plane to improve drawing efficiency.

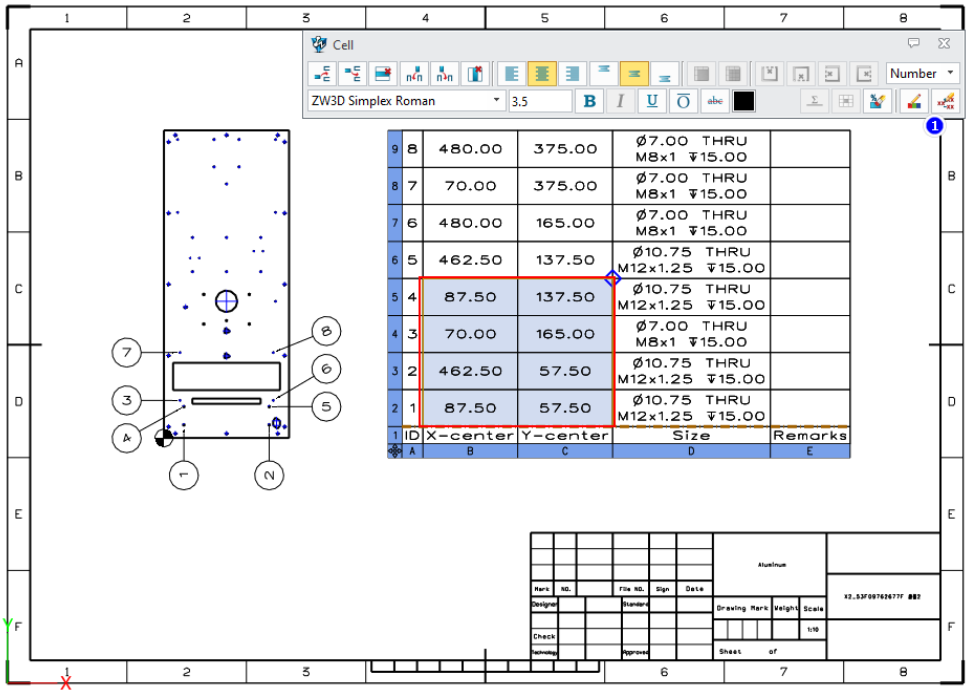


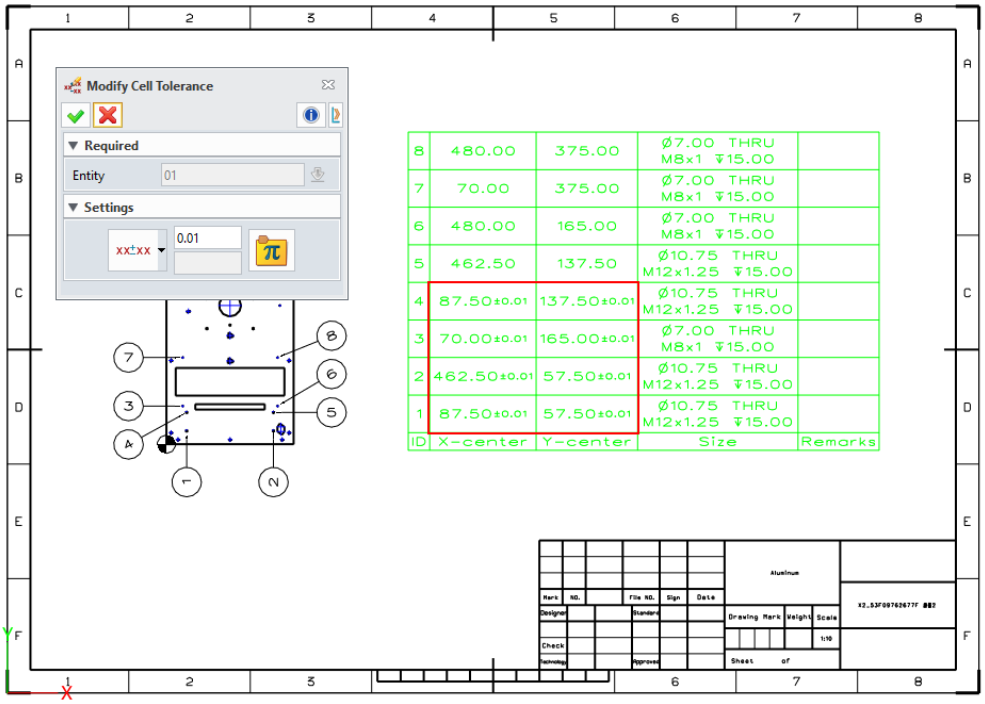
➔ **Where it is**

Drawing Sheet Environment >> Layout >> Edit View >> Rotate View

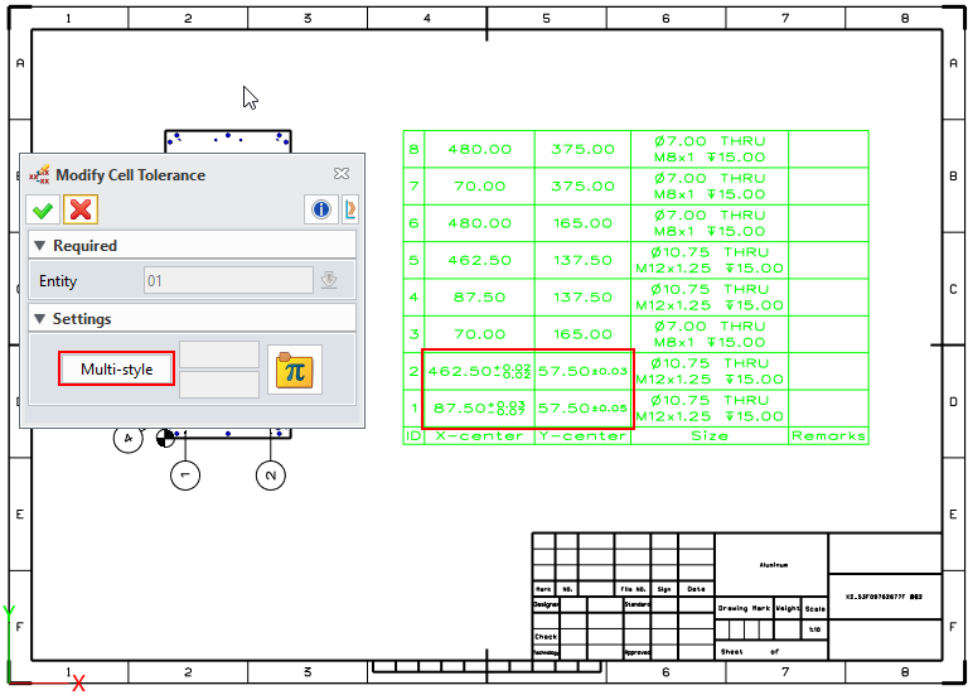
3.9.10Drawing Sheet Table Support Batch Modifying Tolerance

The hole table in the drawing sheet supported the setting of position tolerance in batch. User can box select multiple objects and set tolerance value as the following figure shown:





When the box selected object has multiple tolerance settings, the tolerance setting command displays multiple fields as the following figure shown:

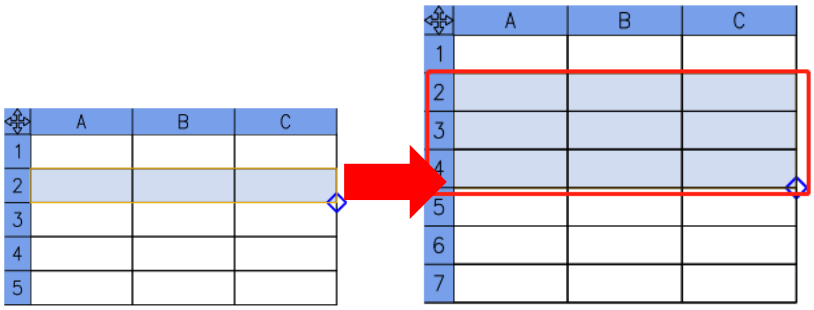


➔ Where it is

Drawing Sheet Environment >> Single Click on Table Cell >> Cell Toolbar >> Modify Cell Tolerance

3.9.11Drawing Sheet Table Support Inserting Row in Batch

In the design industry, tables are often generated in engineering drawings, and customers often need to insert new rows and columns in tables. We added a function that supported inserting multiple rows or columns with a single operation and improved engineers' work efficiency. The way to insert multiple rows are the same as that in excel.

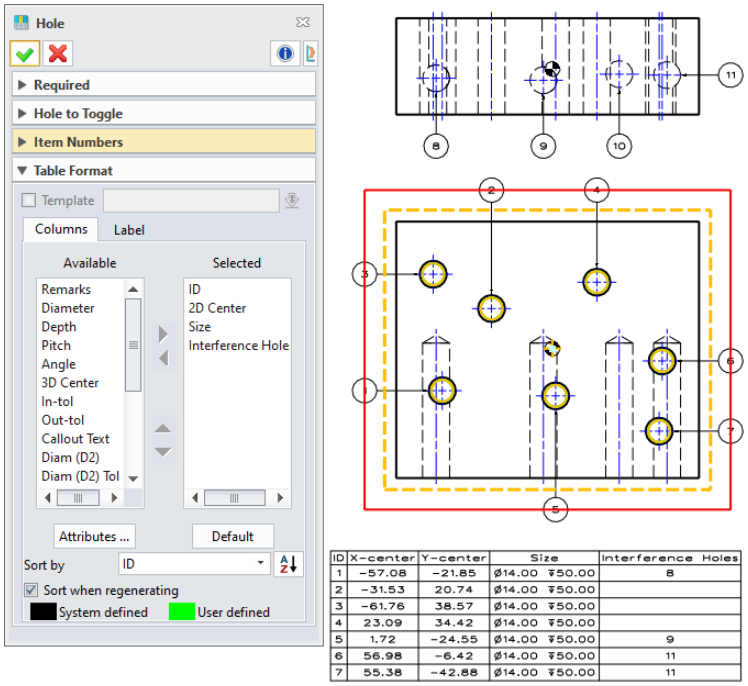


➔ Where it is

Drawing Sheet Environment >> Table Context Menu >> Insert

3.9.12Hole Table Support Displaying Interference Hole

We added "Interference Hole" to Hole table. If checked it, the hole table will display the interference hole under the current model.



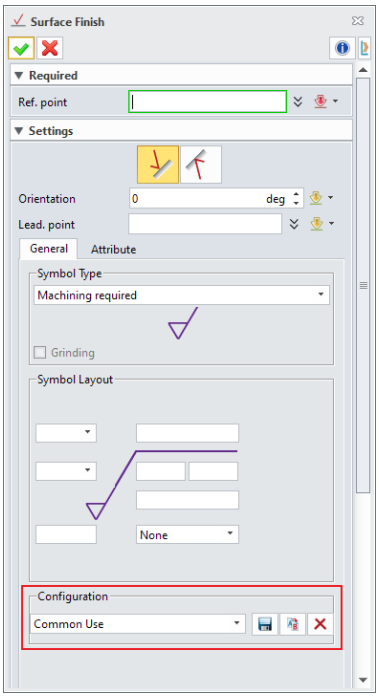
➔ **Where it is**

Drawing Sheet Environment >> Layout >> Table >> Hole

3.9.13Surface Finish Improvement

3.9.13.1 Drawing Sheet Surface Finish Support Configuration

We added the configuration in this version to improve surface finish efficiency and reduce error rate. User can add, rename, and delete configuration on Surface Finish panel and save the commonly used surface finish as configuration. By selecting the configuration, user can get the commonly used surface finish information, which speeds up the dimension efficiency in some way and improve the quality of dimension at the same time.

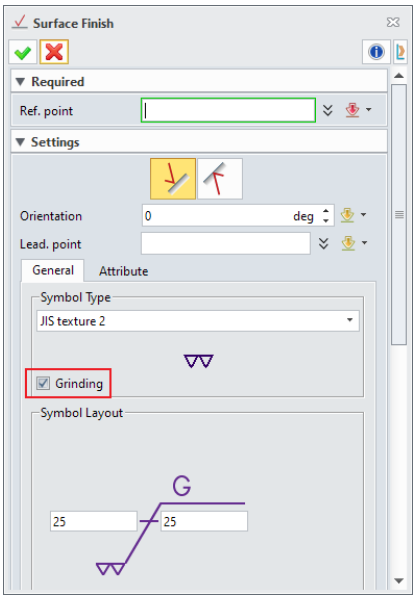


➔ **Where it is**

Drawing Sheet Environment >> Dimension >> Symbol >> Surface Finish >> Configuration

3.9.13.2 Drawing Sheet Surface Finish Remember Grinding Selection

When user is dimensioning “Surface Finish”, the selection setting of “Grinding” will be remembered by the command. When the user dimensions again, the “Grinding” will keep the selected status, which increase dimension efficiency.

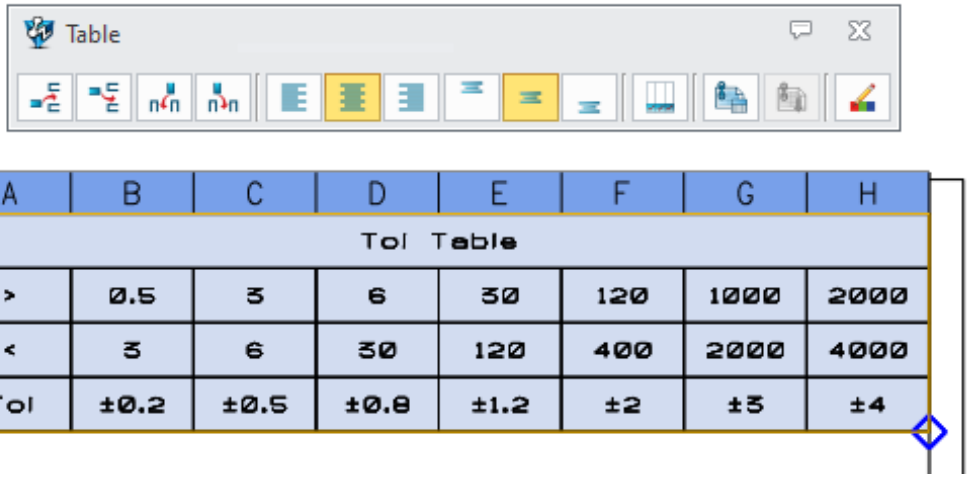


➔ **Where it is**

Drawing Sheet Environment >> Dimension >> Symbol >> Surface Finish >> Grinding

3.9.14Additional List Block and Code List Block Improvement

To solve the problems of difficult to positing and setting text and border, we added using sketch table to make “Additional List Block” and “Code List Block” in ZW3D 2023. User can use table function in the sketch, for example, the text can be left centered or right aligned in the cell, font style can be set, etc.



➔ **Where it is**

Drawing Sheet Environment >> Title Bar Sketch >> User Table

3.9.15 ★New Batch Sheet Format Attributes

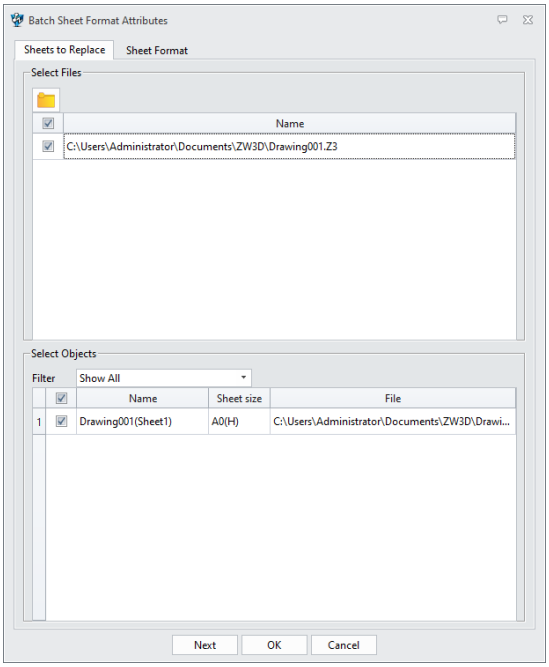
Batch Sheet Format Attributes offers a fast way to batch modify sheet templates. Both multiple-root object files and single-root object files are both supported by the command. The interface of Batch Sheet Format Attributes is divided into two pages **Sheets to Replace** and **Sheet Format**.

3.9.15.1 Sheets to Replace Page

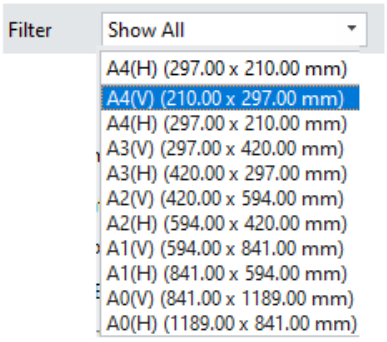
Sheets to Replace page is used to select the sheets of templates that need replacing. The page is divided into two sections. The upper section is the file selection list, in which user selects the target files of template to modify. The file selection can multi object files or single object files. In the currently open files, if they contain drawing sheets object, the files will be automatically added to the file list and check the checkbox.

User can add other local files that are not open by the file selection bottom. The checkbox at the head of table supports all checking/all unchecking files in the list. The context menu of file list should include two items Delete and Remove unchecked items.

At the bottom of **Sheets to Replace** page is the sheet list where all drawing sheets of the currently selected file will be displayed and listed out with the name of “object name (sheet name)”. If a drawing sheet object contains more than one sheets, all the drawing sheets will be separately listed in the list. There are three columns **Name, Sheet size** and **File** in the column.



The sheet list filter supports filtering by the drawing size and lists out all sizes containing in the current sheets.



3.9.15.2 Sheet Format Page

Sheet Format setting supports two sources Custom and Template files. User can decide which attributes of sheets format to be replaced by checking corresponding formats manually.

Sheet Format Page covers all basic items of setting template including **Source format, Title block, Code list, Additional list, Sheet size, Use border, Configuration,** and **Preview.**

- **Source format**: Define which file as sheet format standard. The drop-down list is the currently open Z3DRW files by default. Open the specified Z3DRW file by the file icon. The option “None” is provided, which means no using any standard format file but self-customizing file.
- **Sheet size**: Define the sheet size and select through the drop-down menu.
- **Title block**: Define which title block is applied to the title. The drop-down list will read the title of Z3DEW’s first sheet if a sheet template is selected in Source format, while it will automatically read default template if None is selected in Source format.

  The file path that read by tittle block is \languages\en_US\resource\Templates_Title.Z3

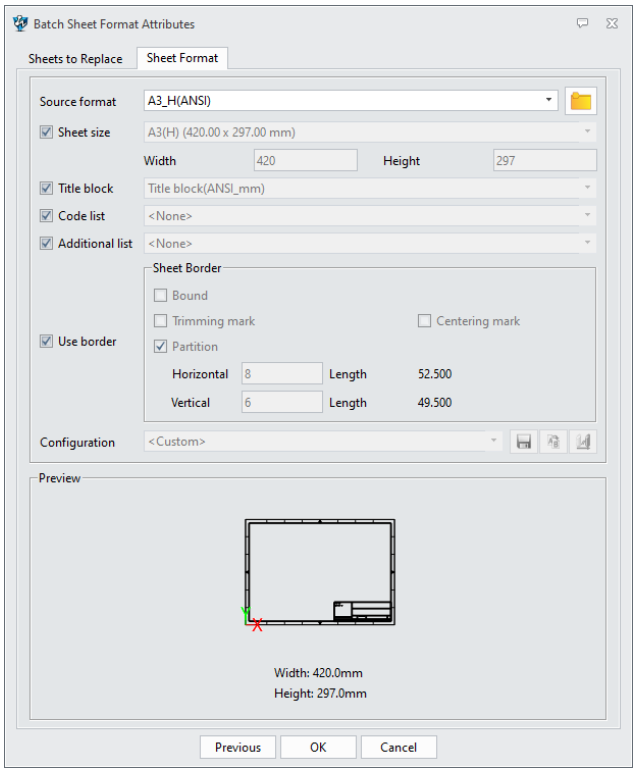
  User can define whether replace the title block be checking/unchecking the checkbox.
- **Code list**: The behavior is the same as title block.

  The file path that read by code list is \languages\en_US\resource\Templates_Code_List.Z3
- **Additional list**: The behavior is the same as title block.

  The file path that ready by additional list is:

  \languages\en_US\resource\Templates_Additional_List.Z3
- **Use border**: The behavior is the same as title block. When “None’ is selected in source format, the sub-checkbox of Use border can read the content of Z3DRW.
- **Configuration:** Save the current configuration and the saved configuration can be re-read.

- **Preview**: Preview the result of the above-mentioned selection.

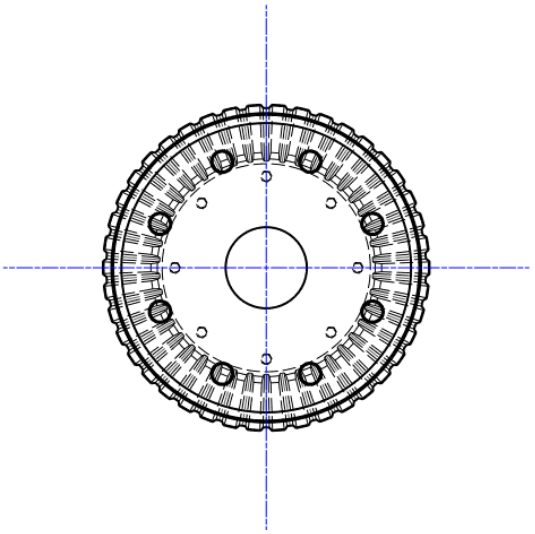


➔ Where it is

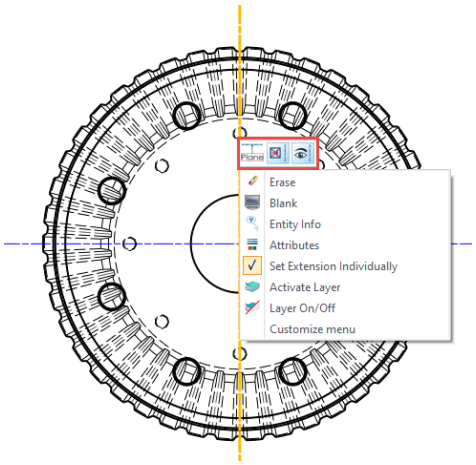
Drawing Sheet Environment >> Layout >> Sheet >> Batch Sheet Format Attributes

3.9.16 New Show Projected Datum

We added Show Projected Datum in the drawing sheet project, which can project the datum planes and datum axes of 3D environment to the drawing sheets. A line from datum projection is presented by a dotted line, whose attributes are the same as center line. User can modify the length of projected line through dragging.



After datum projection, it can be marked by the context datum of datum. If a datum in a view is marked, the projection of the datum in other views can be automatically marked. Support quickly hide/show an unmarked datum.

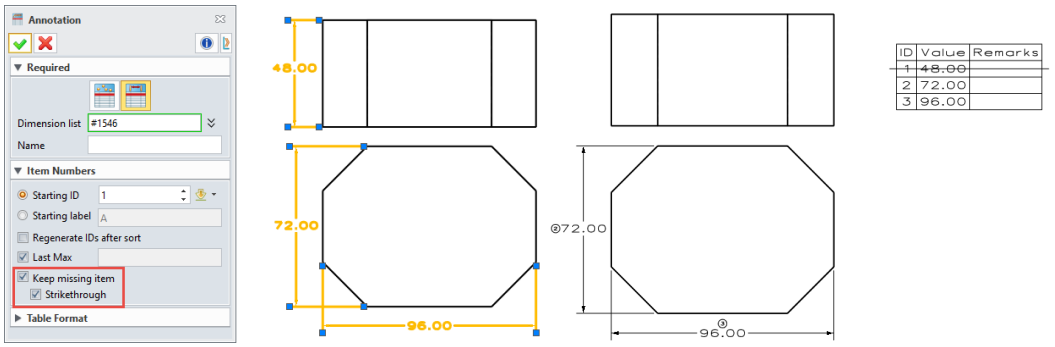


➔ Where it is

Drawing Sheet Environment >> View Attributes >> General >> Show Projected Datum

3.9.17 New Keep Missing Item in Annotation

We added **Keep missing item** in Annotation command in the drawing sheet. As the below figure shown, when dimensions are missing, they will be identified with line-through in the annotation table.

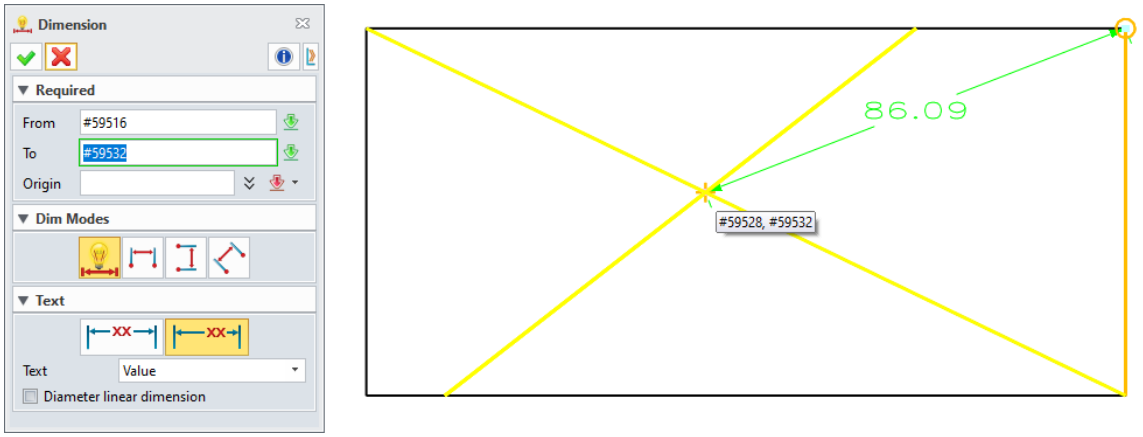


➔ **Where it is**

Drawing Sheet Environment >> Layout >> Table >> Annotation

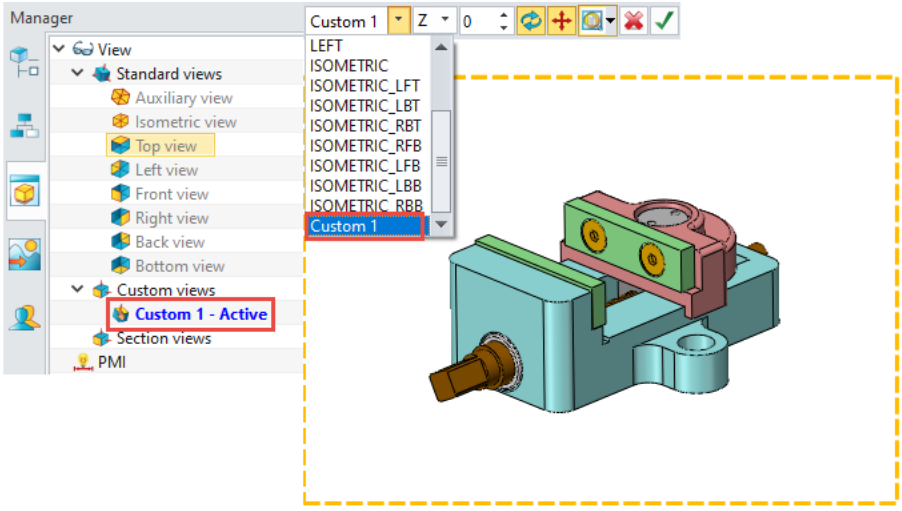
3.9.18 Dimension Capture Improvement

In the drawing sheet environment, the dimension supports capturing an interaction without manually using the filtering tool.



3.9.19 Support Custom View in Rotate View

In the drawing sheet environment, the rotate view supports selecting the pre-defined custom view.

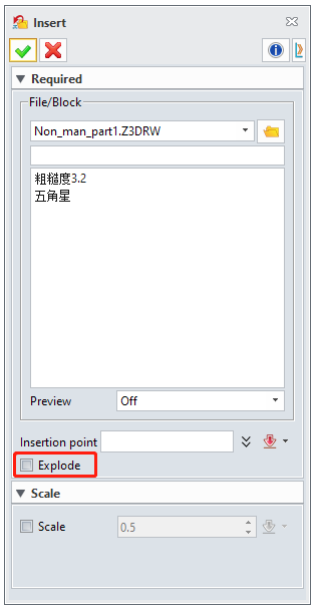


➔ **Where it is**

Drawing Sheet Environment >> Layout >> Edit View >> Rotate View

3.9.20 Support Explode While Inserting Block

In the drawing sheet environment, user can check the Explode while inserting a block to better edit the block.

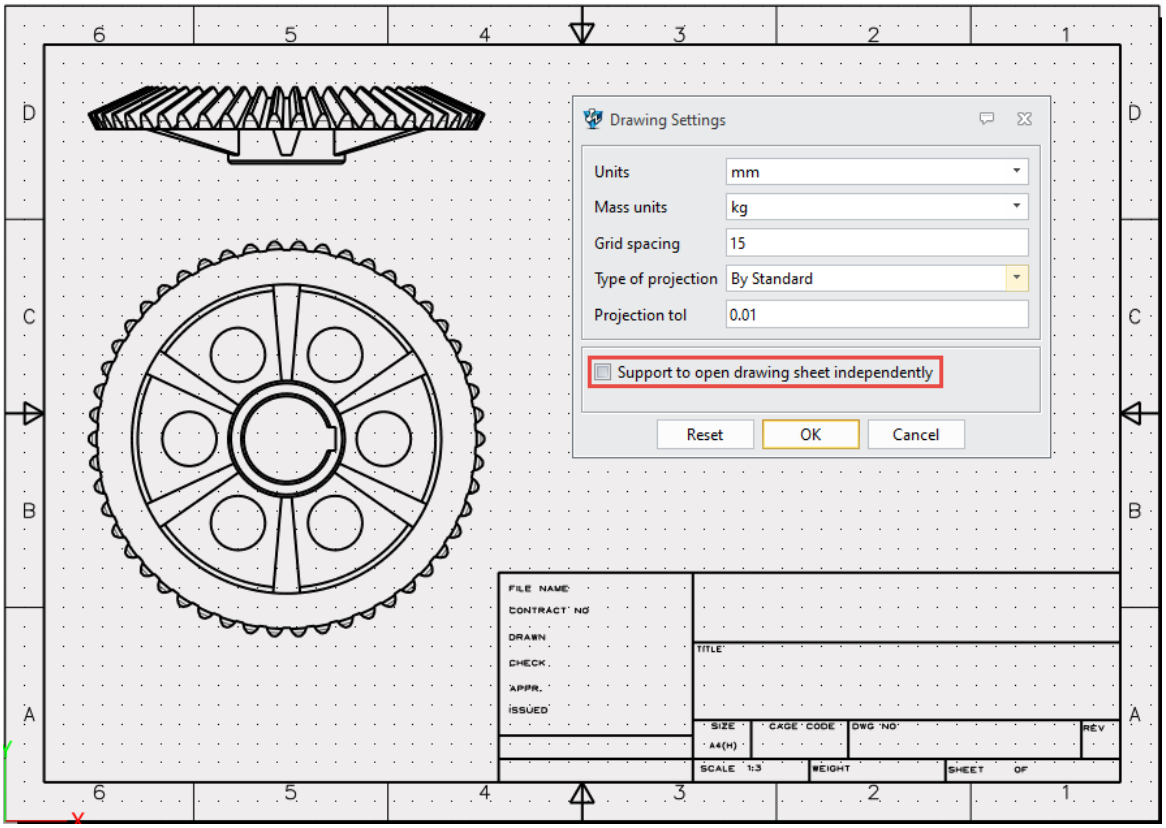


➔ **Where it is**

Drawing Sheet Environment >> Dimension >> Symbol >> Insert

3.9.21 New Support to Open Drawing Sheet Independently

We added “Support to open drawing sheet independently” to acquire dimensions and texts from 3D modeling. After user enabling the function in Drawing Settings, the drawing sheet can be opened under the situation of no corresponding 3D modeling and display all information.

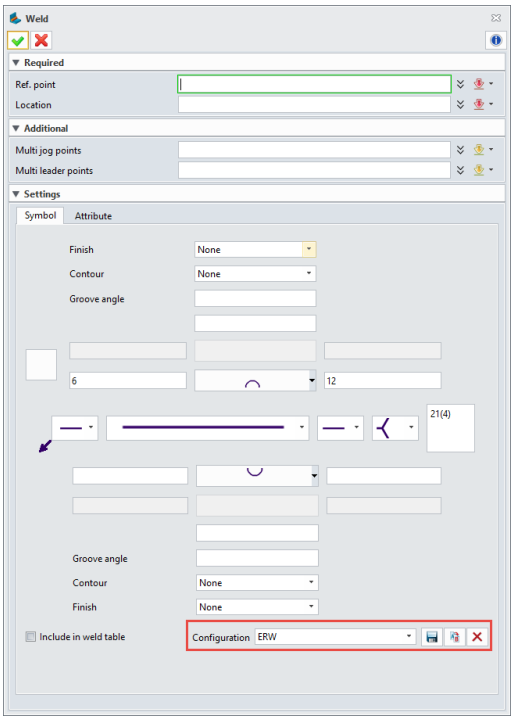


➔ Where it is

Drawing Sheet Environment >> Tools >> Settings >> Drawing Settings >> Support to open drawing independently

3.9.22 Weld in Drawing Sheet Supports Configuration

we added “Configuration” to increase the weld dimension efficiency and reduce error rate. User can create, edit, rename, and delete the weld configuration in the drawing sheet and can save the commonly used weld configuration. To better manage piping weld configuration, the created configuration supports being edited, renamed, and deleted. Weld dimensions can be configured by selecting configuration, which increases the efficiency of dimensions and improves the quality of dimension to a certain extent.

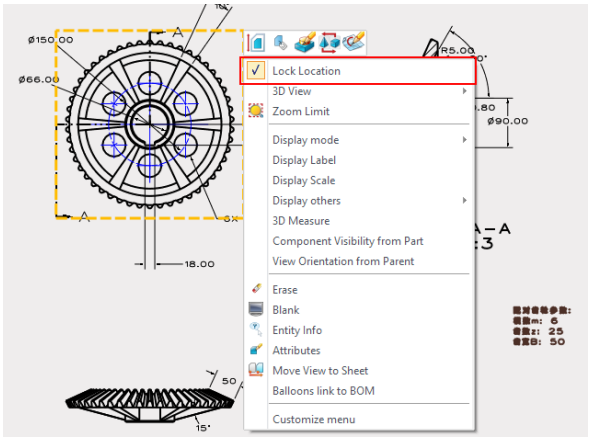


➔ **Where it is**

Drawing Sheet Environment >> Dimension >> Symbol >> Weld >> Configuration

3.9.23 New Lock Location in Drawing Sheet

We added “Lock Location” to lock the view in the drawing sheet. When the option is checked, the locked view cannot be moved.



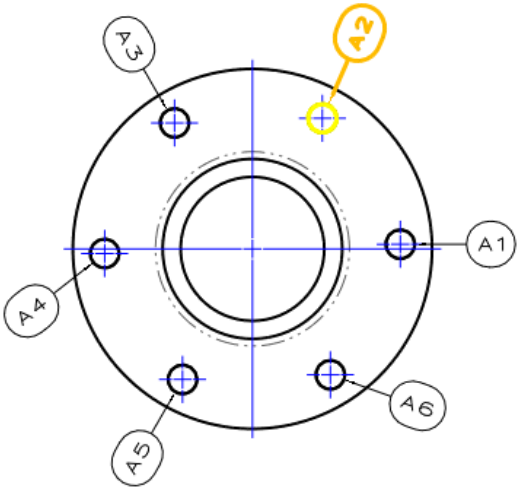
➔ **Where it is**

Drawing Sheet Environment >> View Context Menu>> Lock Location

3.9.24 Hole Table Improvement

3.9.24.1 New Highlight Hole Object

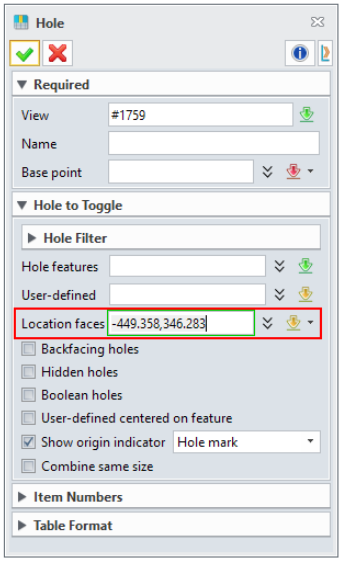
In the drawing sheet environment, user often needs to dimension holes among many hole parts and match up with hole table. The view now can highlight the hole while the hole dimension is selected so that user can locate hole position more clearly.

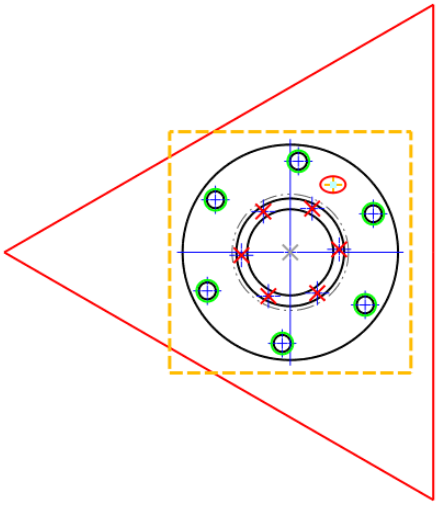


ID	X-center	Y-center	Size
A1	41.15	101.22	Ø8.00 THRU
A2	19.46	136.22	Ø8.00 THRU
A3	-21.69	134.94	Ø8.00 THRU
A4	-41.15	98.66	Ø8.00 THRU
A5	-19.46	63.66	Ø8.00 THRU
A6	21.69	64.94	Ø8.00 THRU

3.9.24.2 New Location Faces in Hole

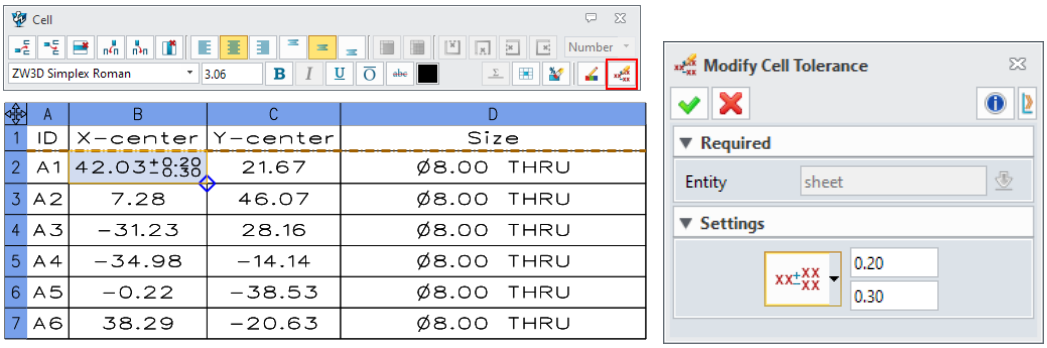
In the drawing sheet environment, we added “Location faces” to assist user specifying the holes on faces to generate the hole table.





3.9.24.3 New Cell Tolerance

In the drawing sheet environment, we added “Modify cell tolerance”, which supports user modifying and defining the tolerance value of hole position.



	A	B	C	D
1	ID	X-center	Y-center	Size
2	A1	42.03 +0.30 −0.30	21.67	Ø8.00 THRU
3	A2	7.28	46.07	Ø8.00 THRU
4	A3	−31.23	28.16	Ø8.00 THRU
5	A4	−34.98	−14.14	Ø8.00 THRU
6	A5	−0.22	−38.53	Ø8.00 THRU
7	A6	38.29	−20.63	Ø8.00 THRU

➔ Where it is

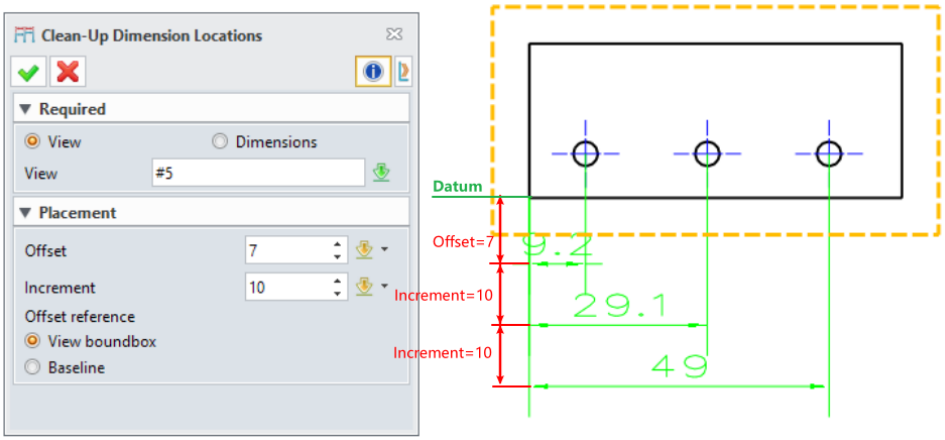
Drawing Sheet Environment >> Layout >> Table >> Hole

3.9.25 Clean-Up Dimension Locations Improvement

In the drawing sheet environment, with the improvement of clean-up dimension, there are two methods: View and Dimension to select dimension object. User can control the distance between dimension object and the baseline through setting the offset value and control the spacing between each dimension through setting increment value.

In the view mode, user can clean up the position of horizontal, vertical, and aligned annotations on the same axis in the layout view. In the dimension mode, user can manually pick the dimension object.

Meanwhile, when the offset reference is regarded as the view bound box by default, the offset baseline is regarded as the view bound box by default. When the offset reference is set as baseline, user can select the reference baseline of dimension object manually.



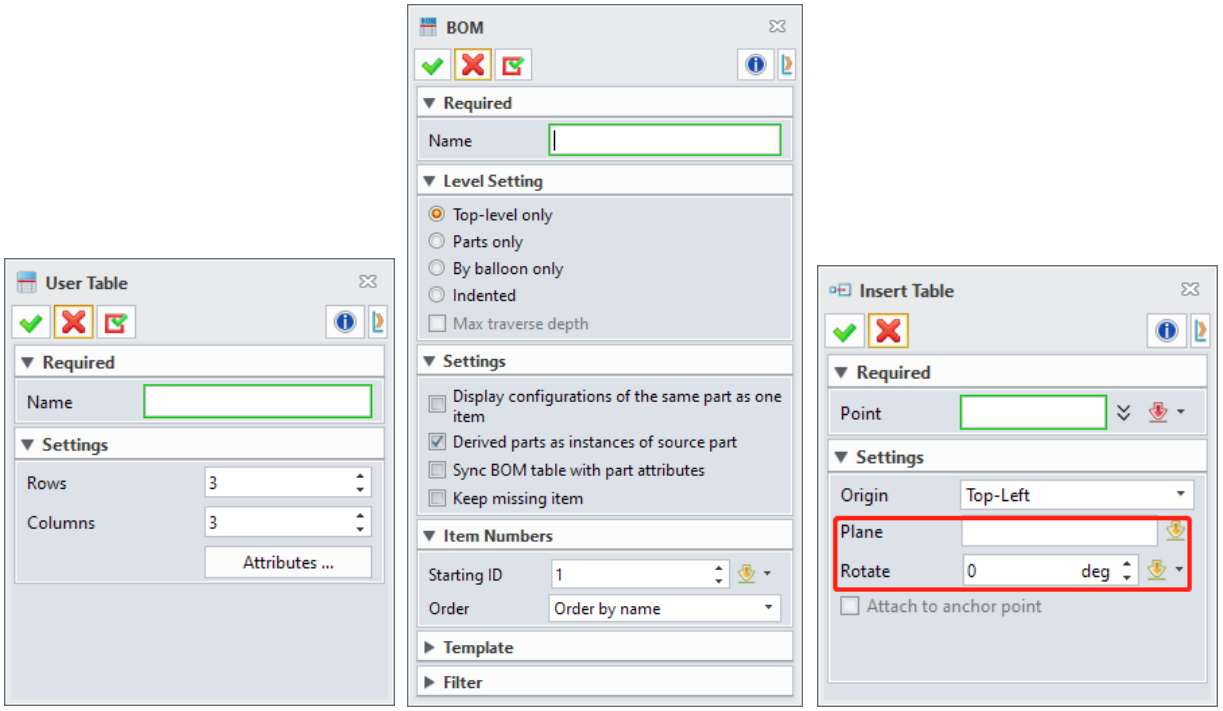
➔ **Where it is**

Drawing Sheet Environment >> Dimension >> Edit Dimension >> Clean-Up Dimension Locations

3.10 PMI

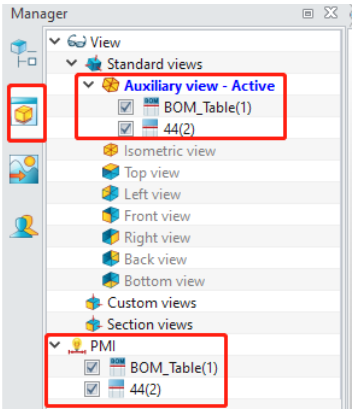
3.10.1 Support Inserting Table in PMI

The function is basically the same as User Table and BOM functions. We added Plane and Rotate functions.



1) User specifies the table name and sets table style. After clicking OK, an interface of Insert Table will prompt up, in which user can specify the place to insert table as well as set the plane to place table. The priority to place plane goes as: the plane specified in Plane widget >> PMI plane specified in View >> activated View Plane.

2) Insert Table is placed under the current view and PMI of Visual Manager, and supports the functions of editing, redefining, deletion, and placing adjustment.

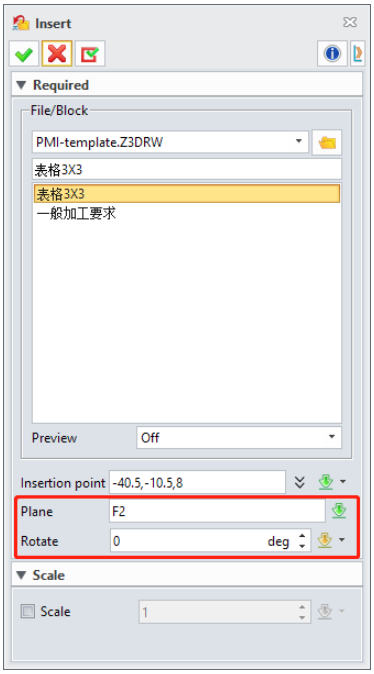


➔ Where it is

Part Environment >> PMI >> Annotation >> User Table/BOM

3.10.2 Insert Block in PMI

The function is the same as Block in the drawing sheet. We added functions of Plane and Rotate in Insert.

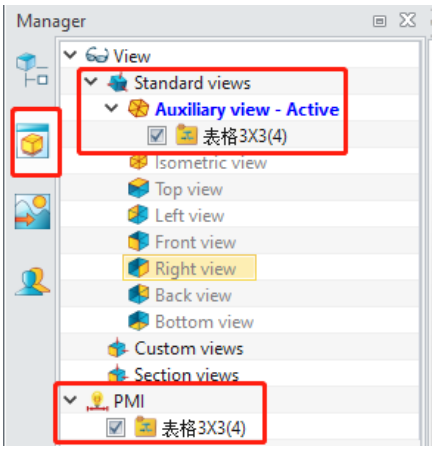


1) In the drawing sheet environment, the objects of tables, annotation, and symbols all can be selected to generate symbol blocks by block's command "Add".

2) The block can only be in the drawing sheet environment, and the new block is saved in the current drawing sheet.

3) When using inserting block in the modelling environment, user can select the corresponding drawing sheet filles and specify the symbol block to insert. User can complete the block inserting to the modeling environment by clicking Insert after setting placing point, placing plane and rotating angle.

4) The inserting Block is placed in the current view and PMI of the visual manager and supports the functions of editing, redefining, deletion, and placing adjustment.



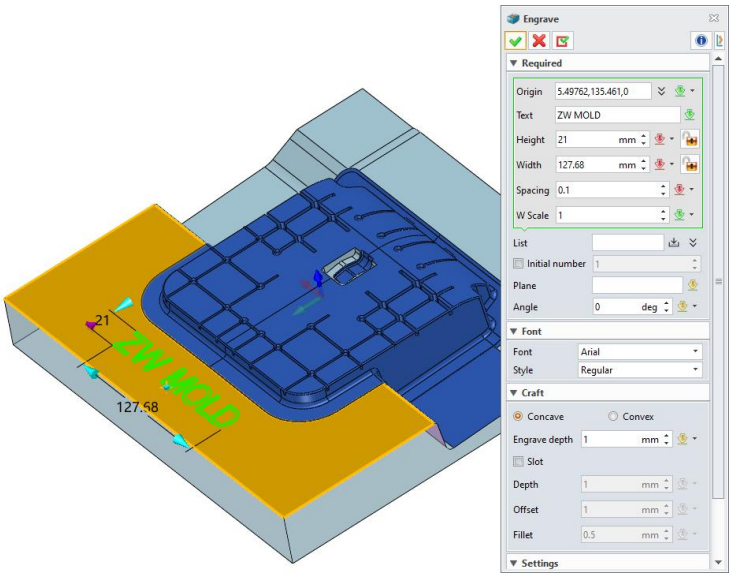
➔ Where it is

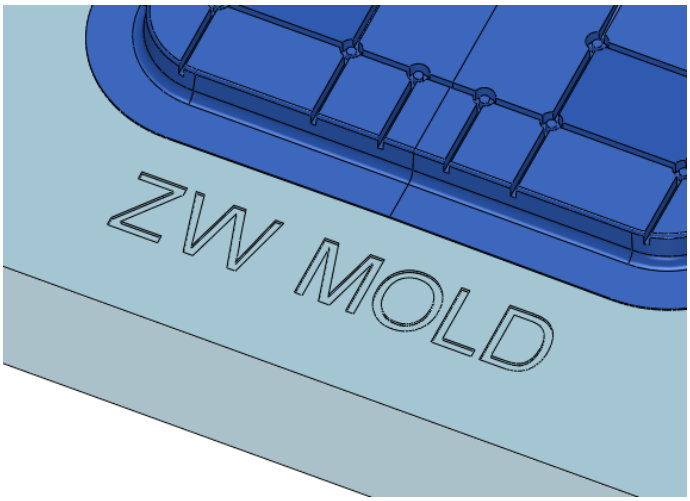
Part Environment >> PMI >> Annotation >> Insert

3.11Mold Design

3.11.1★New Engrave

Engraving is a common feature in mold design. We added "Engrave" to support text setting, placement setting, font setting and engraving setting to help quickly create lettering.



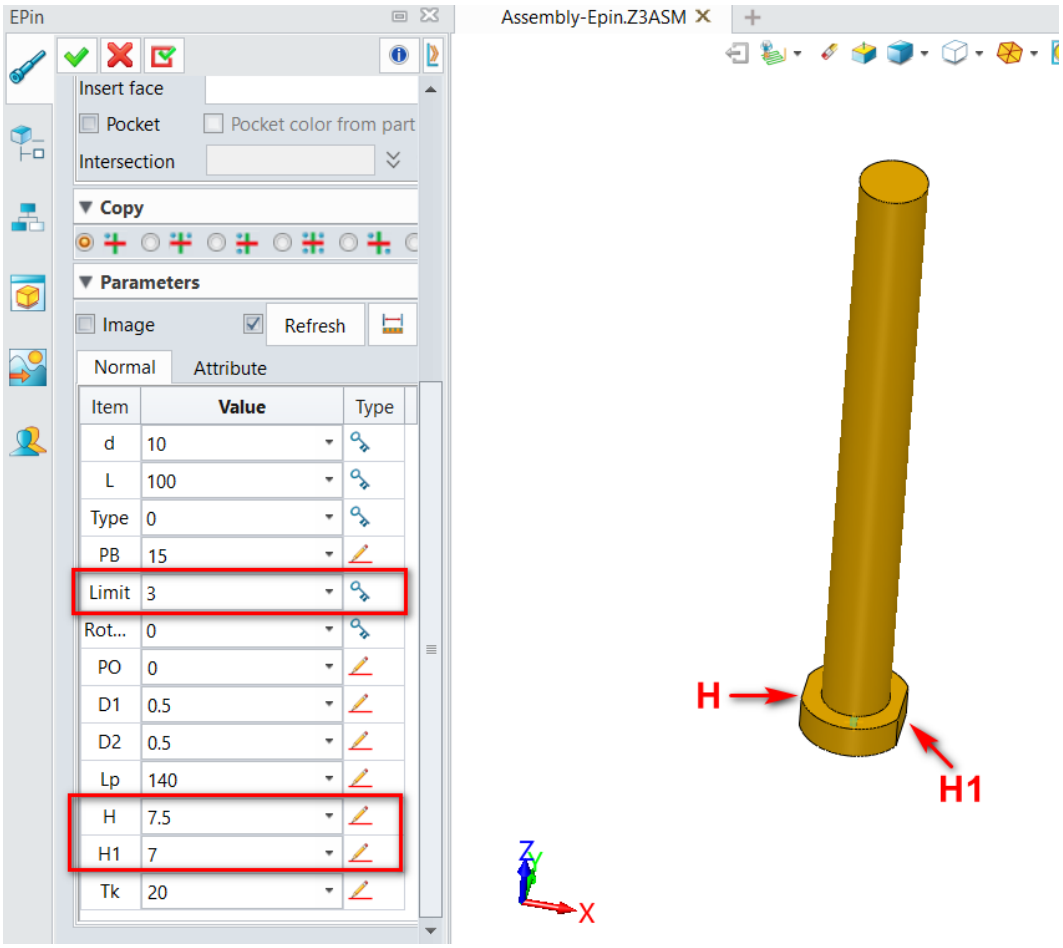


➔ Where it is

Part/Assembly Environment >> Mold >> Detail Design >> Engrave

3.11.2EPin Improvement

To meet the requirements of different scenarios, we added limit way of double cutting edge to Epin and added “Limit3” cutting edge type on the interface as well as two parameters H and H1 to control cutting edge depth.

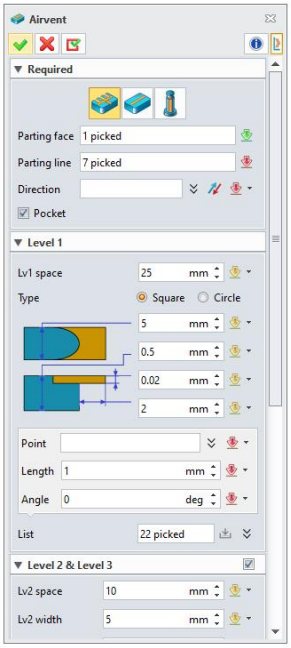


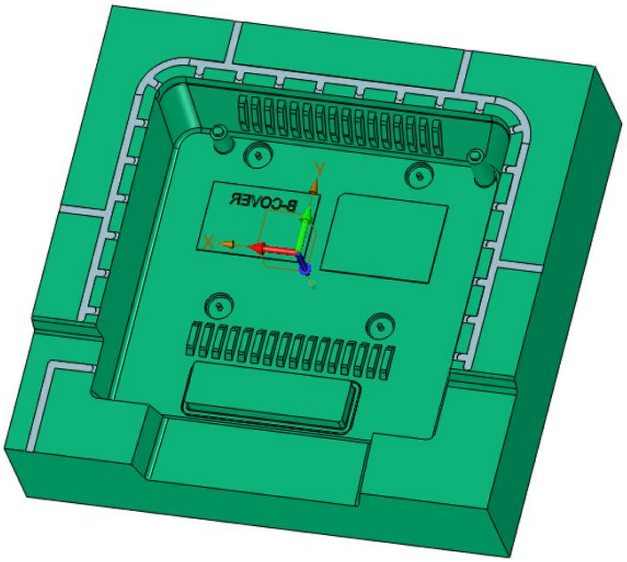
➔ **Where it is**

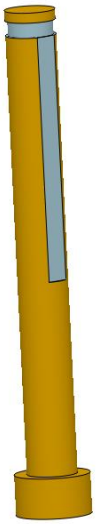
Mold >> Library >> Flat Epin

3.11.3New Airvent

Airvent is one of the common features in mold manufacturing. The command is aimed to provide smart, efficient, and convenient exhaust function. The current airvent command provides three types of exhaust: Face, Single, and Pin.





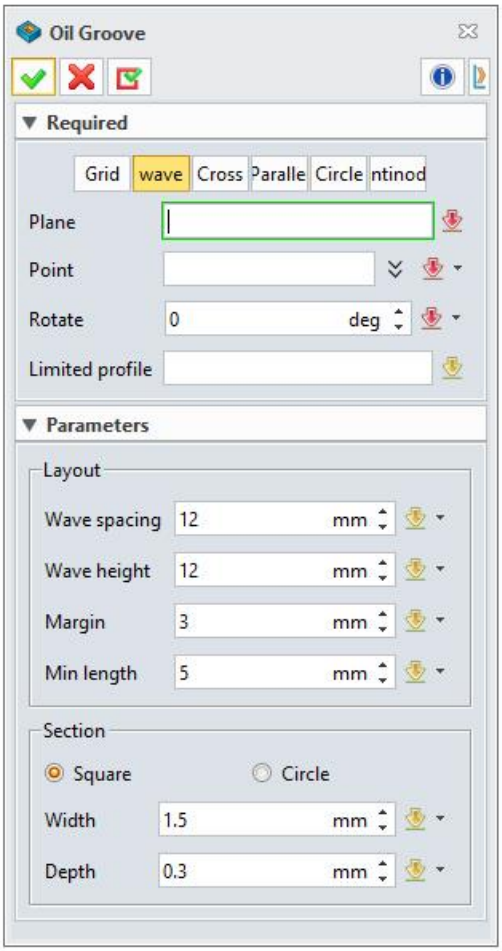


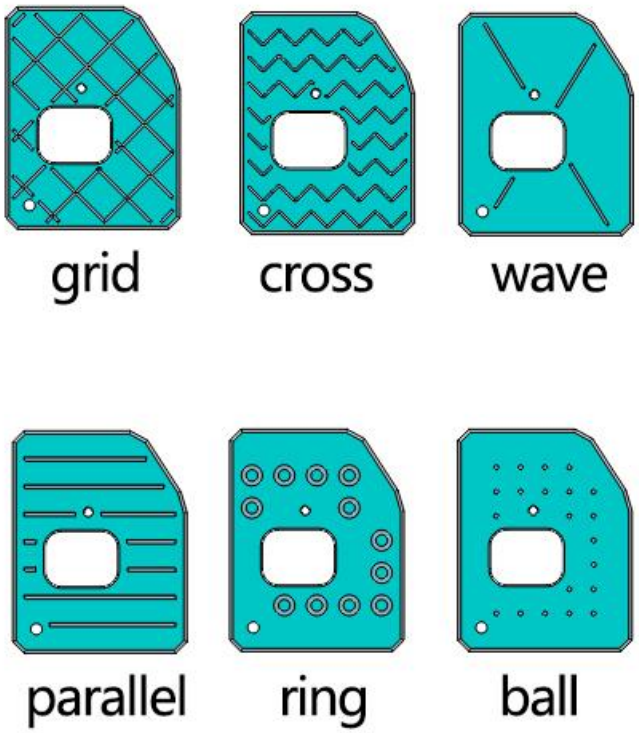
➔ **Where it is**

Mold >> Detail Design >> Airvent

3.11.4★New Oil Groove

Oil groove has a wide range of application in the mold. The oil groove is mainly opened on the friction surface with relative movement, but there are many steps in manual oil groove design. Thus, the command is added to simply user's design step and improve design efficiency. There are six types of oil groove: grid, cross, wave, parallel, ring, and ball.



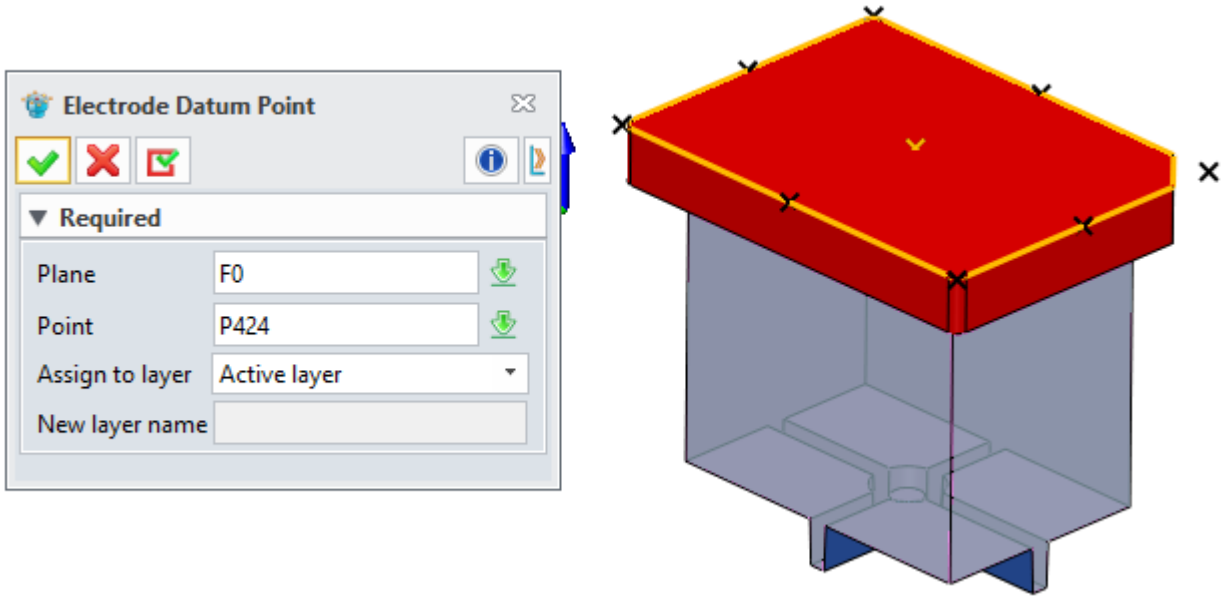


➔ **Where it is**

Mold >> Detail Design >> Oil Groove

3.11.5Electrode Datum Point Improvement

Electrode Datum Point support setting datum point to create the locate layer.



➔ **Where it is**

Part/Assembly Environment >> Electrode >> Electrode Tools >> Electrode Datum Point

3.11.6 Mold Workflow Improvement

We adjusted the mold design workflow by putting Load Model before Layout so to layout the mold by referring to the realistic modeling situation.

➔ **Where it is**

Part Environment >> Mold >> Product

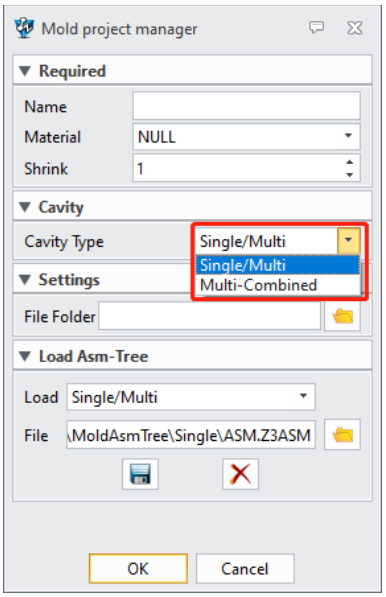
3.11.7 Mold Assembly Optimization

3.11.7.1 Mold Project Mode Adjustment

The mold is designed as two types “Single/Multi” and “Multi-Combined”.

The “Single/Multi” mode supports the function of “Single-Cavity” and “Multi-Cavity (combine cavities is unchecked)”.

The “Multi-Combined” mode supports the function of “Multi-Cavity(combine cavities is checked)”.



➔ Where it is

Part Environment >> Mold >> Product >> Mold Project

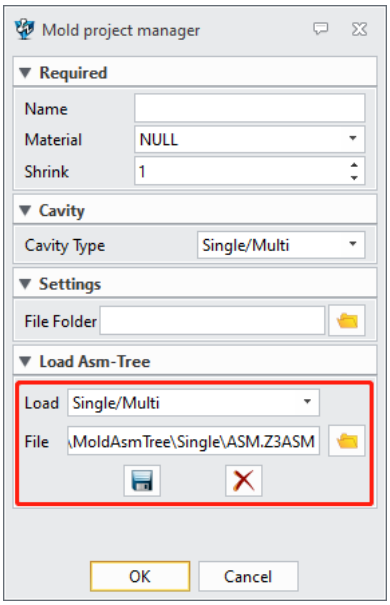
3.11.7.2 ★New Load Asm-Tree in Mold

User wants to customize a suitable mold assembly tree which can be called in creating a mold project.

In Mold Project command, we added sub-function of mold assembly.

The mold assembly configuration template supports being read, selected, and saved. Select the mold assembly template through path (select Z3 files for the multi-root template and select mold project top assembly single-root files for the single-root template). Click to save the selected template or click to delete the selected and saved template. Select the template from the drop-down list in “Load”, which can be used to create mold project.

The template is supported to custom modify in the directory at Program Files\ZWSOFT\R2630\ZWMold\MoldAsmTree.

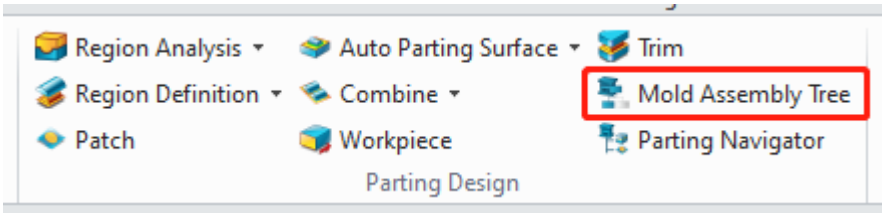


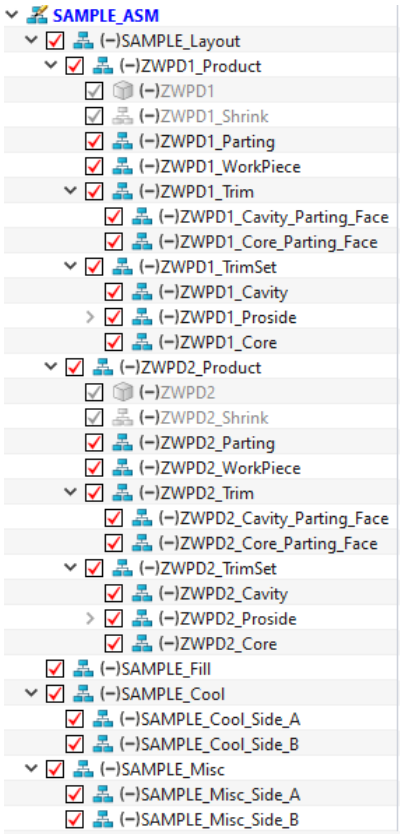
➔ **Where it is**

Part/Assembly Environment >> Mold >> Product >> Mold Project

3.11.7.3 New Mold Assembly Tree

In mold project, the selected assembly template can be loaded in creating mold project with the use of this command.





➔ Where it is

Assembly Environment >> Mold >> Parting Design >> Mold Assembly Tree

3.11.7.4 New Trim Node

Trim node is added in the product of assembly tree.

After using Trim command, the system will automatically add two nodes of core parting face and cavity parting face in the product node of assembly tree. The moving mold parting surface and the fixed mold parting surface are respectively connected to the node in the form of Geom to part.

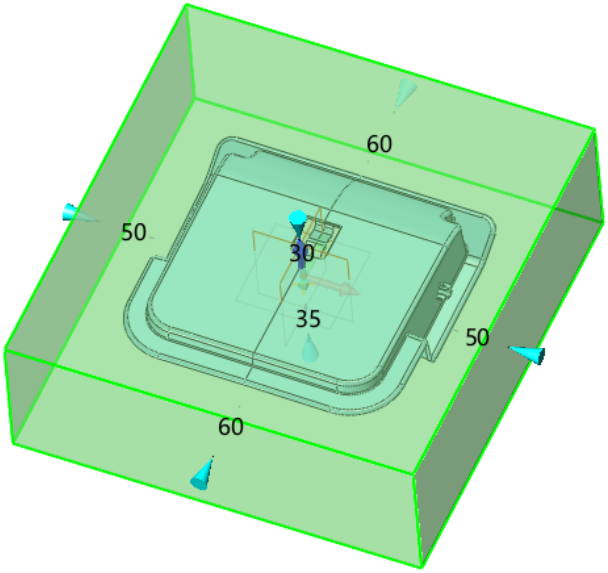
➔ Where it is

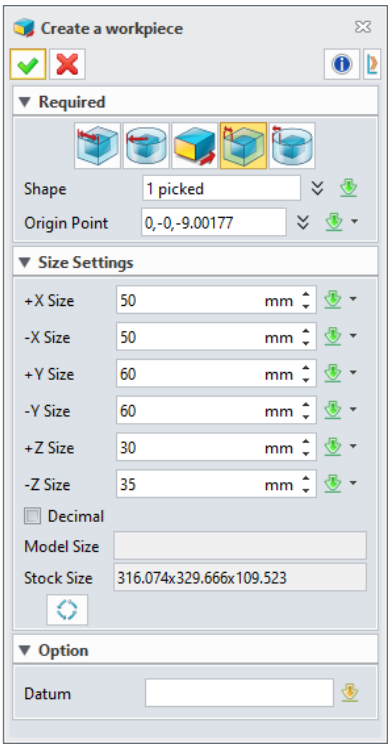
Assembly Environment >> Mold >> Parting Design >> Trim

3.11.8 ★ Workpiece Improvement

The workpiece command supports the creation method by defining the distance from workpiece boundary to product: Block by side and Cylinder by side.

We updated the button of workpiece command. The workpiece will identity product size with the method of Block by side or Cylinder by side. Use “Rebuild” to re-identify product once product is changed.





➔ Where it is

Part/Assembly Environment >> Mold >> Parting Design >> Workpiece

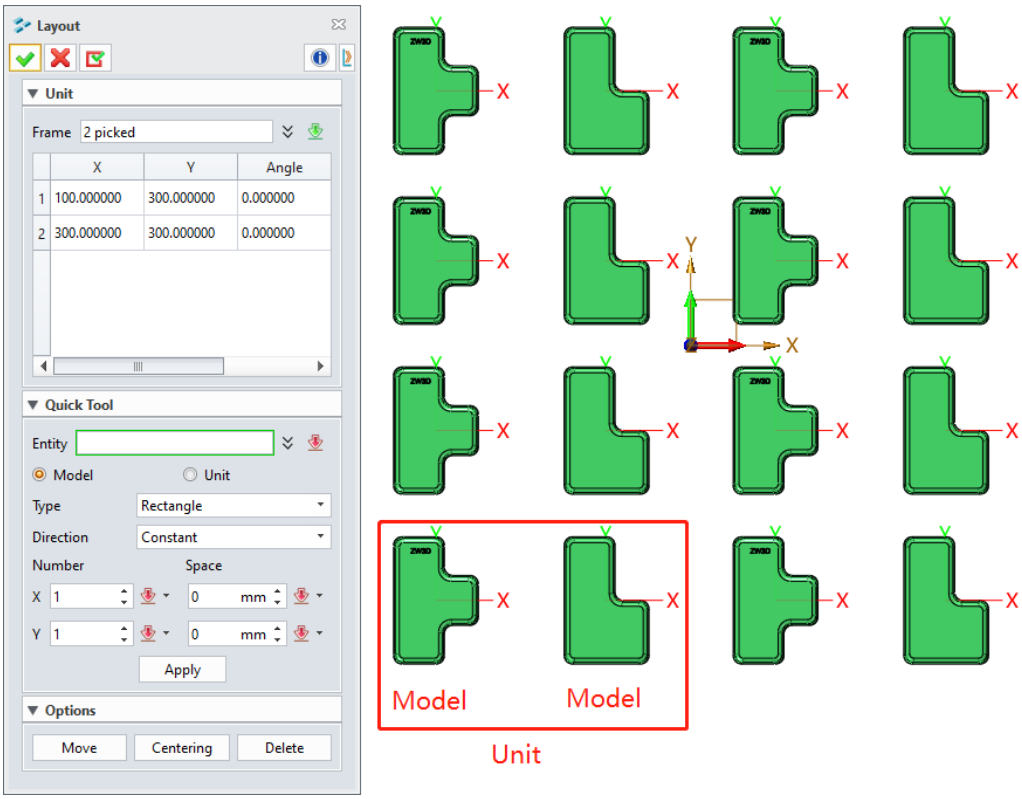
3.11.9 ★Layout Improvement

1) Unit Layout/Overall Layout

The new layout brings more convenient functions.

Unit Layout supports setting layout for a single product. User can adjust layout for product molding freely.

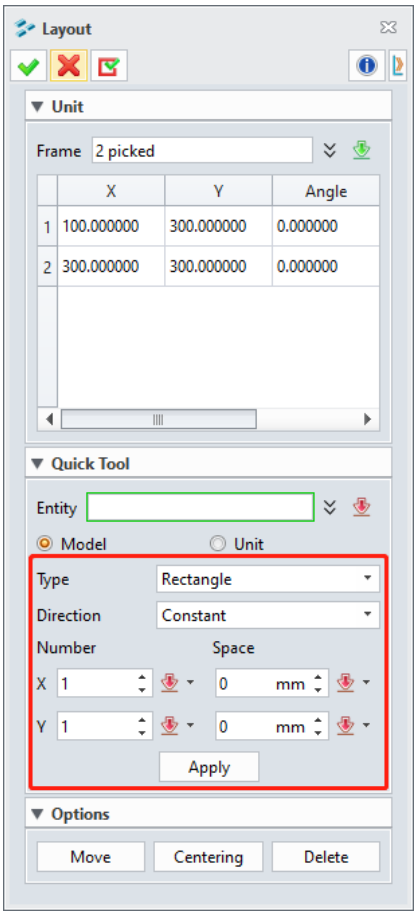
Model Layout supports setting layout for the entire units through one group product layout to quickly complete numbers of layout.



2) Layout Tools

The previous layout functions became the sub-function in the current version as tools, so that user can quickly layout objects.

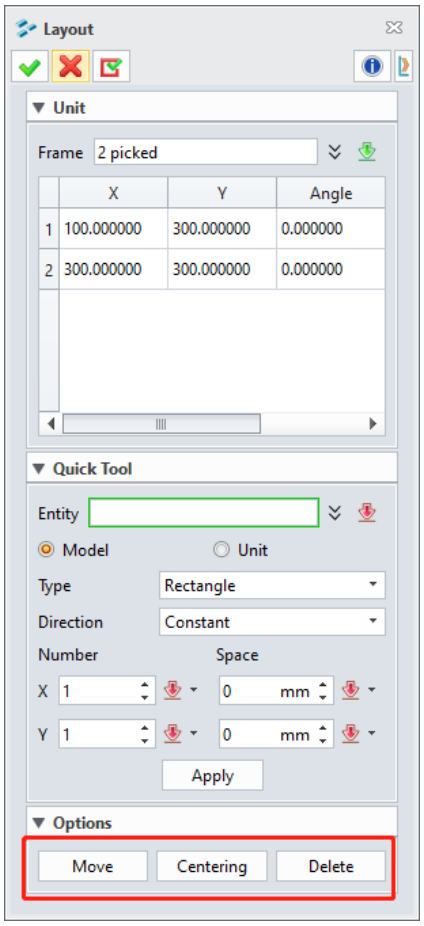
User can create layout quickly in the method of number and spacing.



3) Move, delete, centering

We added the functions of “Move” “Delete” and “Centering” in the current version so that user can edit the layout method in point-to-point.

User can move and delete product. When the layout offsets from the coordinate center, user can use “Centering” to quickly move the layout product to the coordinate.



➔ **Where it is**

Part/Assembly Environment >> Mold >> Product >> Layout

3.11.10 ★Standard Part Mechanism Improvement

Place type

We added place type in the command “General” and default placing coordinate. When the placement mode is set to the default coordinate system, the inserted standard part coordinate system will coincide with the system coordinate system.

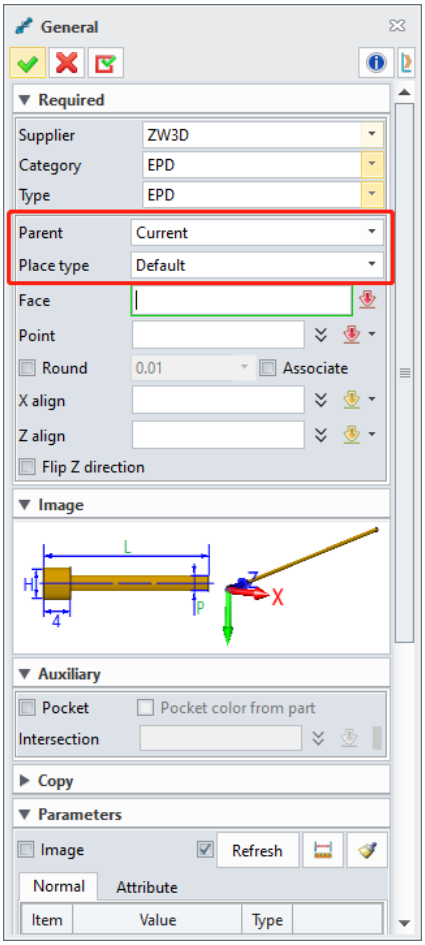
Parent

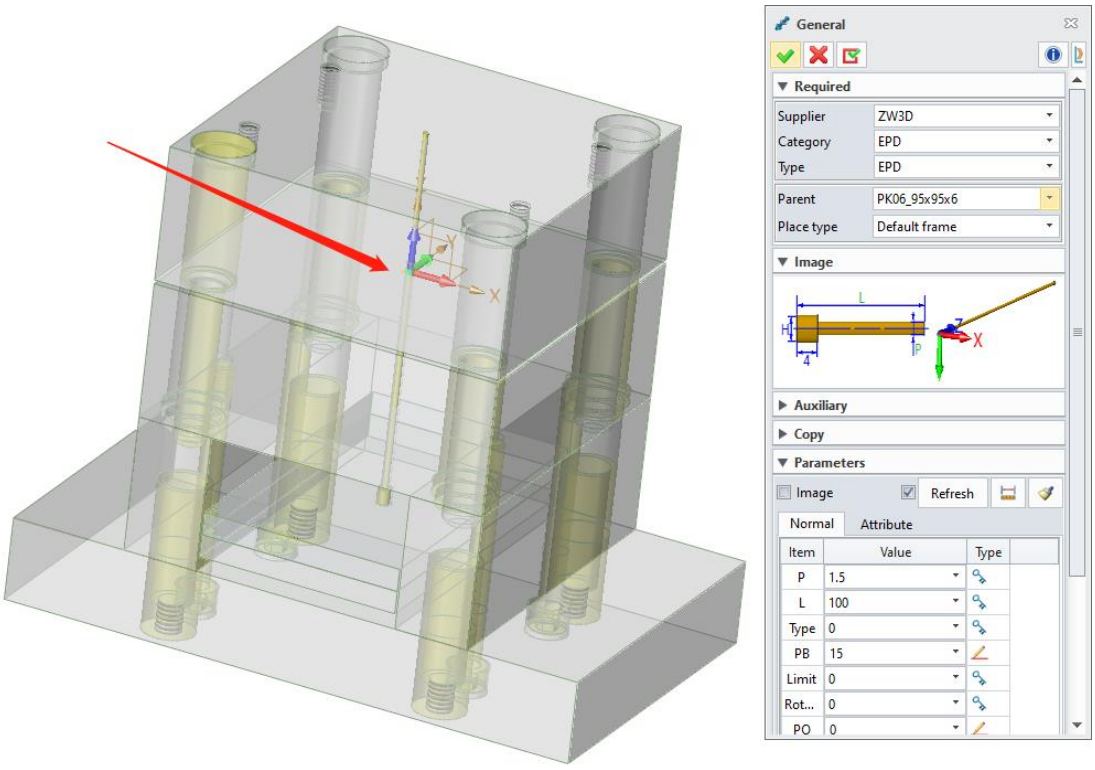
User can add row ##PARENT in standard part excel configuration table and input node name in the table. when there exists ##PARENT in the excel configuration table, the general command will appear parent drop-down list option. When the standard part is called, support inserting nodes in parent item. The inserted standard part will generate subassembly/sub-part in the selected node.

External Reference Read

In the standard part excel configuration table, use $W=&Aplate: H format, calling other variable of other files. For instance, the above-mentioned case, $W is the variable as standard part, &Aplave: H means H variable of Aplate part.

13	**##EXPRESSIONS**
14	$ZPlace=-[&MB_ZWM:BP0H]-[&MB_ZWM:CP0H]+[&MB_ZWM:GAP0FP]+[&MB_ZWM:FP0H]-0.5
15	
16	**##PARENT**
17	**PK06_95x95x6**



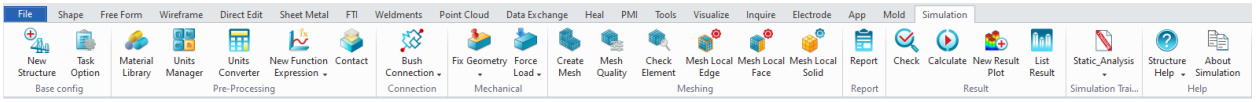


➔ **Where it is**

Part/Assembly Environment >> Mold >> Library >> General

3.12ZWStructure

The module of ZWStructure simulation is integrated inZW3D 2023. Thus, the functionality of Sim is supported inZW3D 2023.



➔ **Where it is**

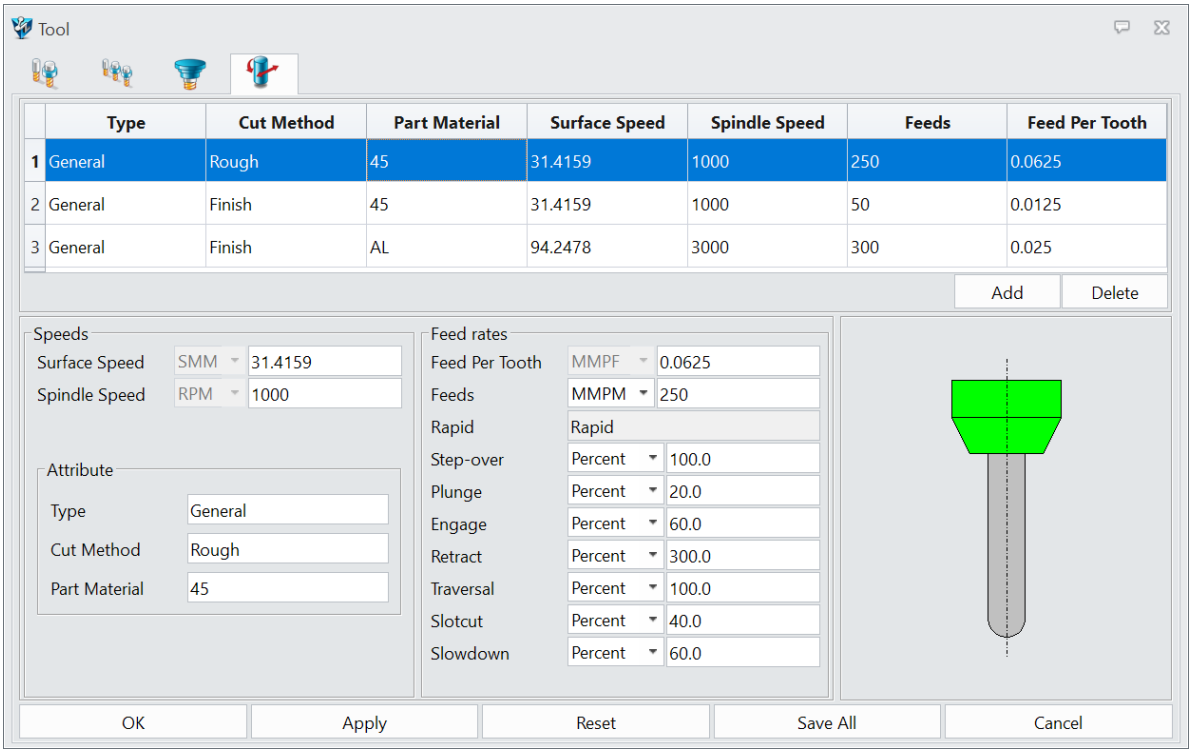
Part/Assembly Environment >> Simulation

4 CAM

4.1 ★Tool Speed and Feed Overall Upgrade

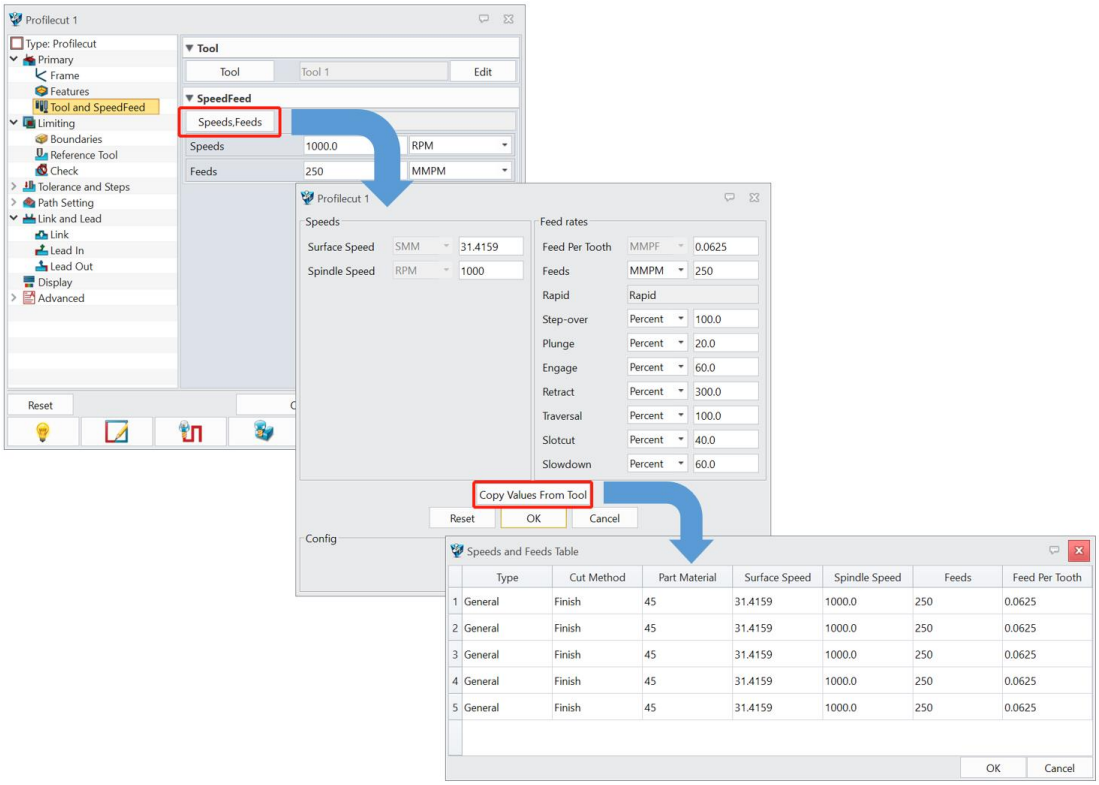
We had a comprehensive innovation to speed and feed function in ZW3D 2023 including:

- Re-design the function interface and remove redundant speed setting parameters.
- Newly support a tool loading multiple groups of speed and feed parameters to make it adjust to different machining scenarios.
- Newly support the real time conversion computing between the tool surface speed and spindle speed.
- Newly support the real time conversion computing between the tool feed amount and feed per tooth amount.



- Improve the workflow of operation loading tool speed and feed.

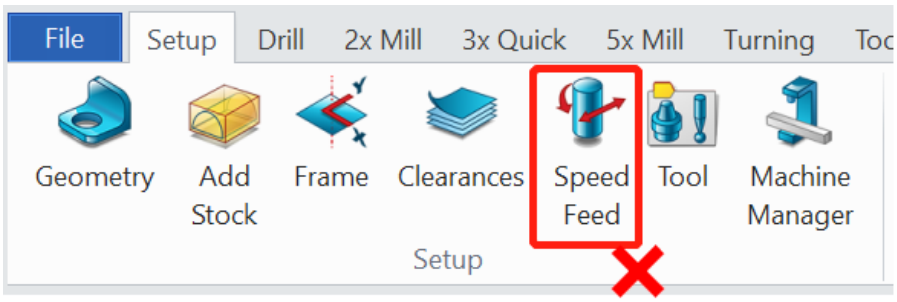
  1. On CAM tree, when dragging tool to operation Tool node, operation will automatically load the first group of speed and feed of tool.
  2. In the operation, only the tool's speed and feed applied to the current operation can be loaded.



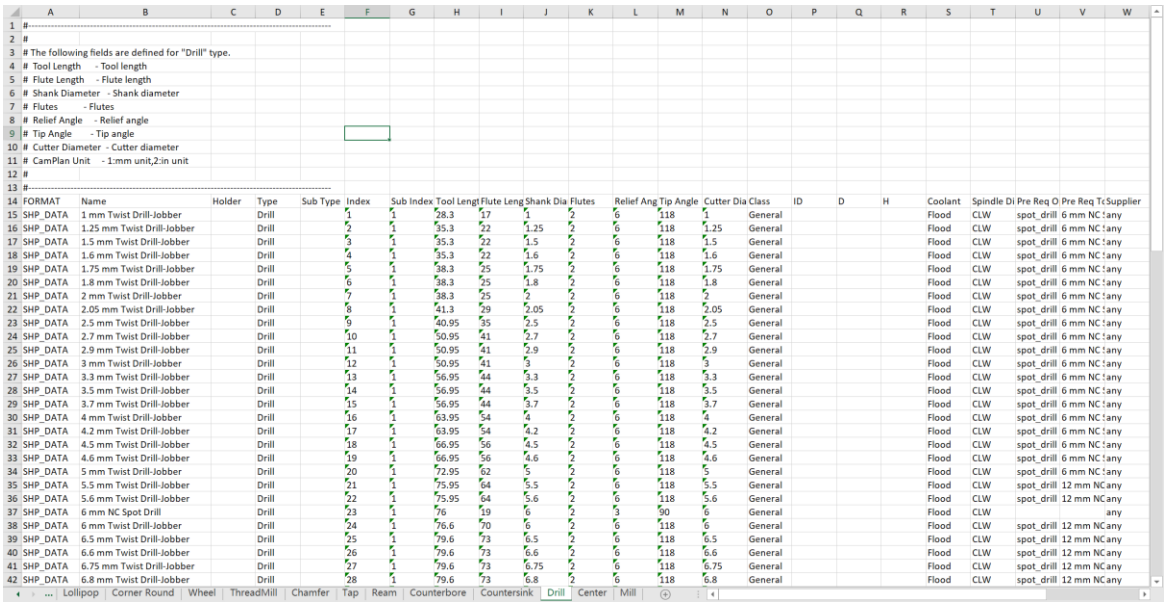
4.2 ★Tool Library Overall Upgrade

We had made comprehensive innovations in tool library in ZW3D 2023, including:

- Remove Speed Feed from Ribbon toolbar and tool's speed and feed do not exist as an independent object.



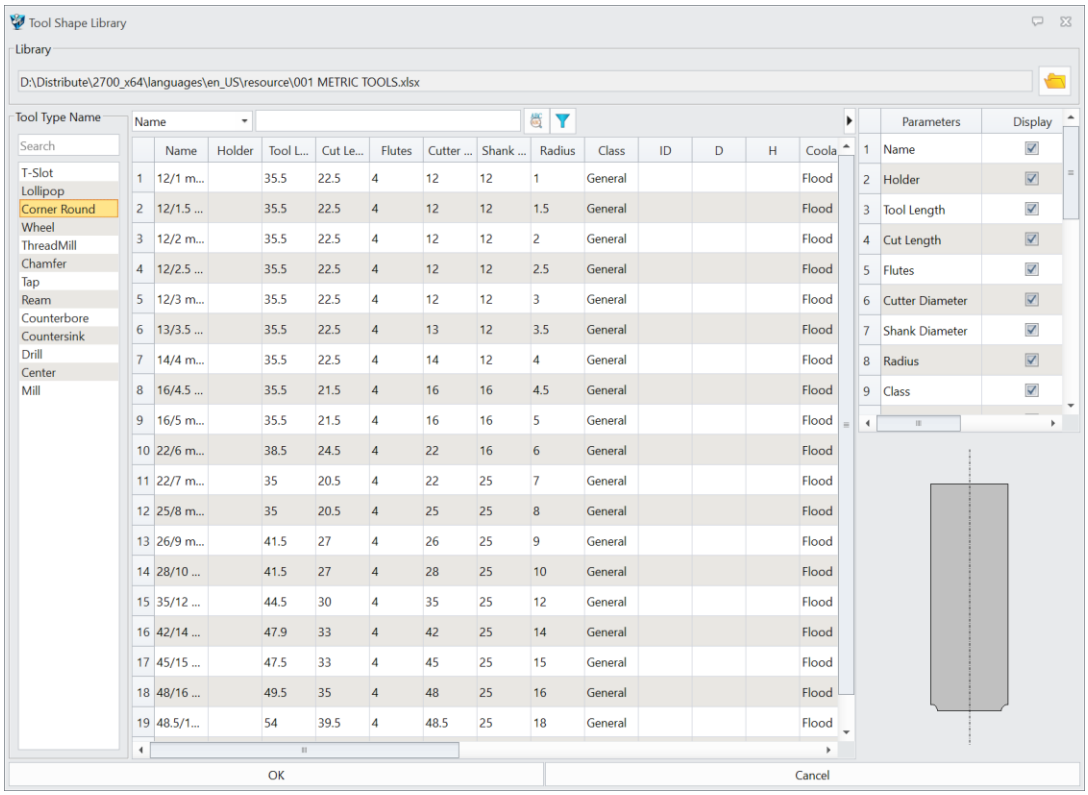
- The tool is no longer stored in Z3 format but in text format (Excel file) and make tool parameters more intuitive.



	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V	W
1	#----------																						
2	#																						
3	# The following fields are defined for "Drill" type.																						
4	# Tool Length - Tool length																						
5	# Flute Length - Flute length																						
6	# Shank Diameter - Shank diameter																						
7	# Flutes - Flutes																						
8	# Relief Angle - Relief angle																						
9	# Tip Angle - Tip angle																						
10	# Cutter Diameter - Cutter diameter																						
11	# CamPlan Unit - 1:mm unit,2:in unit																						
12	#																						
13	#----------																						
14	FORMAT	Name	Holder	Type	Sub Type	Index	Sub Index	Tool Lengt	Flute Leng	Shank Dia	Flutes	Relief Ang	Tip Angle	Cutter Dia	Class	ID	D	H	Coolant	Spindle Di	Pre Req O	Pre Req Tc	Supplier
15	SHP_DATA	1 mm Twist Drill-Jobber		Drill		1	1	28.3	17	1	2	6	118	1	General				Flood	CLW	spot_drill	6 mm NC	any
16	SHP_DATA	1.25 mm Twist Drill-Jobber		Drill		2	1	35.3	22	1.25	2	6	118	1.25	General				Flood	CLW	spot_drill	6 mm NC	any
17	SHP_DATA	1.5 mm Twist Drill-Jobber		Drill		3	1	35.3	22	1.5	2	6	118	1.5	General				Flood	CLW	spot_drill	6 mm NC	any
18	SHP_DATA	1.6 mm Twist Drill-Jobber		Drill		4	1	35.3	22	1.6	2	6	118	1.6	General				Flood	CLW	spot_drill	6 mm NC	any
19	SHP_DATA	1.75 mm Twist Drill-Jobber		Drill		5	1	38.3	25	1.75	2	6	118	1.75	General				Flood	CLW	spot_drill	6 mm NC	any
20	SHP_DATA	1.8 mm Twist Drill-Jobber		Drill		6	1	38.3	25	1.8	2	6	118	1.8	General				Flood	CLW	spot_drill	6 mm NC	any
21	SHP_DATA	2 mm Twist Drill-Jobber		Drill		7	1	38.3	25	2	2	6	118	2	General				Flood	CLW	spot_drill	6 mm NC	any
22	SHP_DATA	2.05 mm Twist Drill-Jobber		Drill		8	1	41.3	29	2.05	2	6	118	2.05	General				Flood	CLW	spot_drill	6 mm NC	any
23	SHP_DATA	2.5 mm Twist Drill-Jobber		Drill		9	1	40.95	35	2.5	2	6	118	2.5	General				Flood	CLW	spot_drill	6 mm NC	any
24	SHP_DATA	2.7 mm Twist Drill-Jobber		Drill		10	1	50.95	41	2.7	2	6	118	2.7	General				Flood	CLW	spot_drill	6 mm NC	any
25	SHP_DATA	2.9 mm Twist Drill-Jobber		Drill		11	1	50.95	41	2.9	2	6	118	2.9	General				Flood	CLW	spot_drill	6 mm NC	any
26	SHP_DATA	3 mm Twist Drill-Jobber		Drill		12	1	50.95	41	3	2	6	118	3	General				Flood	CLW	spot_drill	6 mm NC	any
27	SHP_DATA	3.3 mm Twist Drill-Jobber		Drill		13	1	56.95	44	3.3	2	6	118	3.3	General				Flood	CLW	spot_drill	6 mm NC	any
28	SHP_DATA	3.5 mm Twist Drill-Jobber		Drill		14	1	56.95	44	3.5	2	6	118	3.5	General				Flood	CLW	spot_drill	6 mm NC	any
29	SHP_DATA	3.7 mm Twist Drill-Jobber		Drill		15	1	56.95	44	3.7	2	6	118	3.7	General				Flood	CLW	spot_drill	6 mm NC	any
30	SHP_DATA	4 mm Twist Drill-Jobber		Drill		16	1	63.95	54	4	2	6	118	4	General				Flood	CLW	spot_drill	6 mm NC	any
31	SHP_DATA	4.2 mm Twist Drill-Jobber		Drill		17	1	63.95	54	4.2	2	6	118	4.2	General				Flood	CLW	spot_drill	6 mm NC	any
32	SHP_DATA	4.5 mm Twist Drill-Jobber		Drill		18	1	66.95	56	4.5	2	6	118	4.5	General				Flood	CLW	spot_drill	6 mm NC	any
33	SHP_DATA	4.6 mm Twist Drill-Jobber		Drill		19	1	66.95	56	4.6	2	6	118	4.6	General				Flood	CLW	spot_drill	6 mm NC	any
34	SHP_DATA	5 mm Twist Drill-Jobber		Drill		20	1	72.95	62	5	2	6	118	5	General				Flood	CLW	spot_drill	6 mm NC	any
35	SHP_DATA	5.5 mm Twist Drill-Jobber		Drill		21	1	75.95	64	5.5	2	6	118	5.5	General				Flood	CLW	spot_drill	12 mm NC	any
36	SHP_DATA	5.6 mm Twist Drill-Jobber		Drill		22	1	75.95	64	5.6	2	6	118	5.6	General				Flood	CLW	spot_drill	12 mm NC	any
37	SHP_DATA	6 mm NC Spot Drill		Drill		23	1	76	19	6	2	3	90	6	General				Flood	CLW			any
38	SHP_DATA	6 mm Twist Drill-Jobber		Drill		24	1	76.6	70	6	2	6	118	6	General				Flood	CLW	spot_drill	12 mm NC	any
39	SHP_DATA	6.5 mm Twist Drill-Jobber		Drill		25	1	79.6	73	6.5	2	6	118	6.5	General				Flood	CLW	spot_drill	12 mm NC	any
40	SHP_DATA	6.6 mm Twist Drill-Jobber		Drill		26	1	79.6	73	6.6	2	6	118	6.6	General				Flood	CLW	spot_drill	12 mm NC	any
41	SHP_DATA	6.75 mm Twist Drill-Jobber		Drill		27	1	79.6	73	6.75	2	6	118	6.75	General				Flood	CLW	spot_drill	12 mm NC	any
42	SHP_DATA	6.8 mm Twist Drill-Jobber		Drill		28	1	79.6	73	6.8	2	6	118	6.8	General				Flood	CLW	spot_drill	12 mm NC	any

Lollipop | Corner Round | Wheel | ThreadMill | Chamfer | Tap | Ream | Counterbore | Countersink | Drill | Center | Mill

- In Excel file, the tools are stored in tables according to tool type so that user can quickly search tool.
- User can directly open tool library excel to edit. It is convenient for user to create and modify tool in batch.
- Redesign the function interface to read tool library in ZW3D. Restart new tool library interface from Load All and Load Tool Shape, separately.

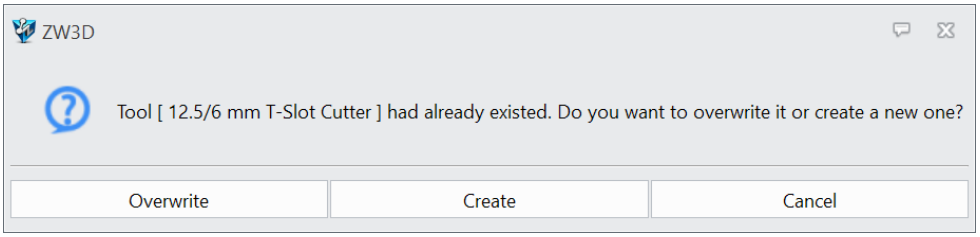


- New duplication of name check mechanism

  When saving a tool to the tool library from CAM plan, if the system detects that a tool with the same name already exists in the CAM plan, a prompt will pop up.

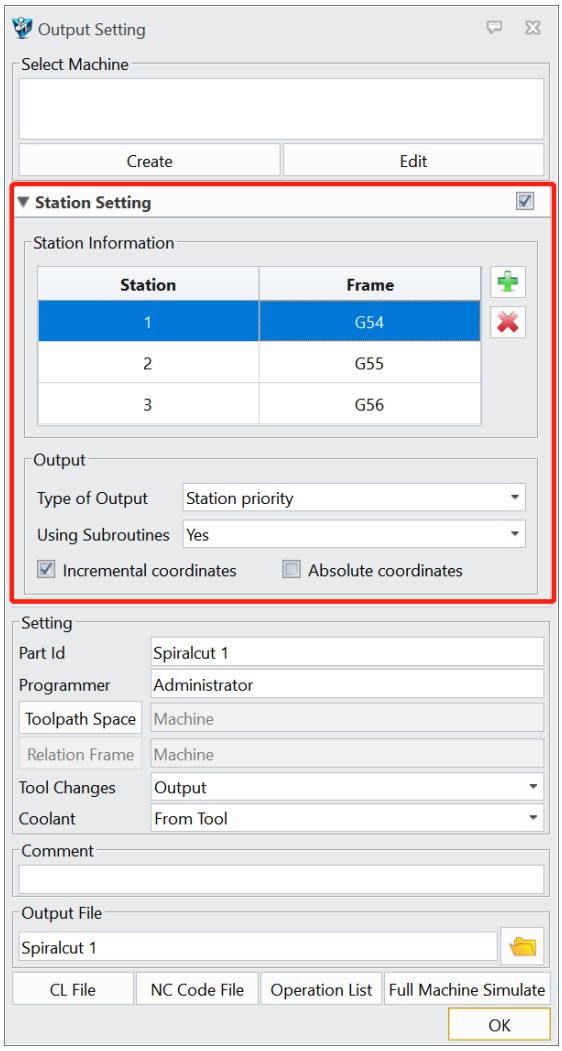
  When loading a tool to the current CAM plan from the tool library, if the system detects that a tool with the same name already exists in the CAM plan, a prompt will pop up.

  Click Overwrite button, it will overwrite the existing tool's parameters, and then click Create button to create a new tool.

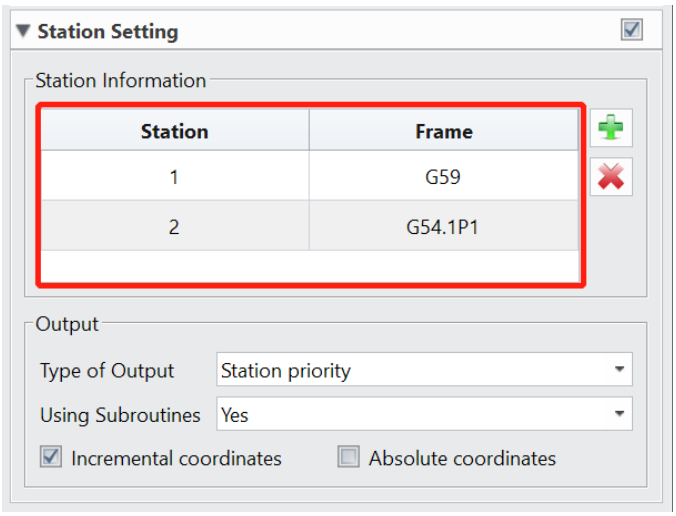


4.3 ★New Multi Work Station

We added multi-workstation in ZW3D 2023 to meet user's need of multi-station machining. The multi-station is displayed in the output setting interface with a closed status by default.



- The multi-station function supports user customized multiple coordinates. But the input coordinate format has requirements. At present, it supports the input of “G+number” and “G+number+P+number” and the letters must be uppercase. Incorrect format or order or other characters cannot be entered.

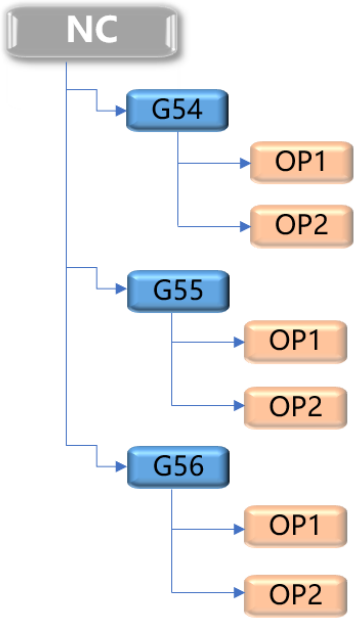
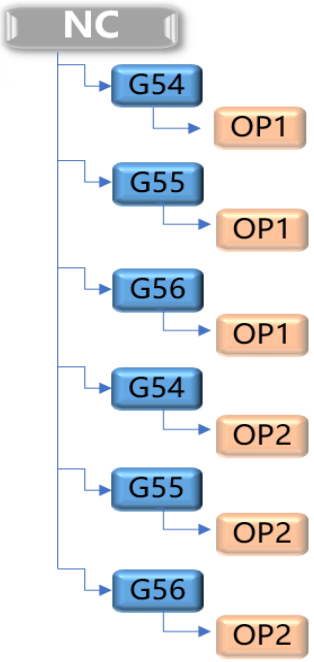


- Support two output modes Station Priority and Operation Priority. Please use it with ZW_subcl.znc files.

  Station priority mode: When NC operation contains multiple operations, the coordinate output is in priority.

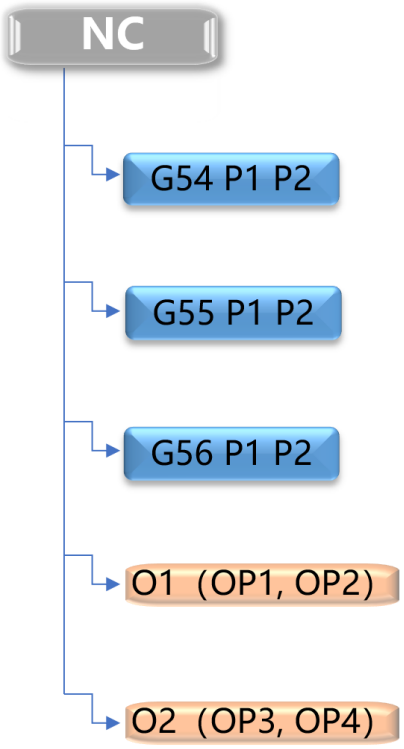
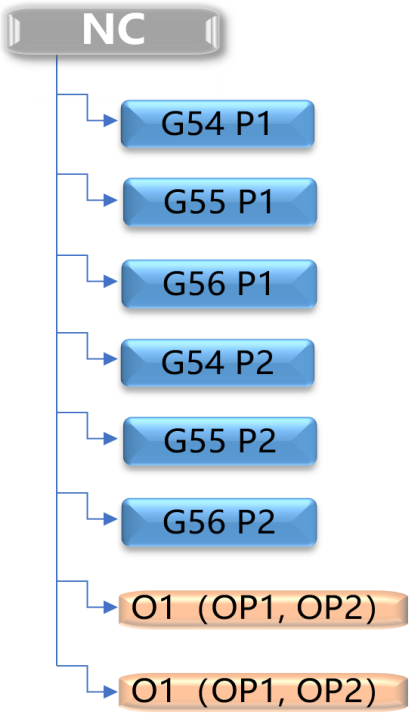
  Operation priority mode: When NC operation contains multiple operations, the operation output is in priority.

  NC code structures under the two modes are as below:

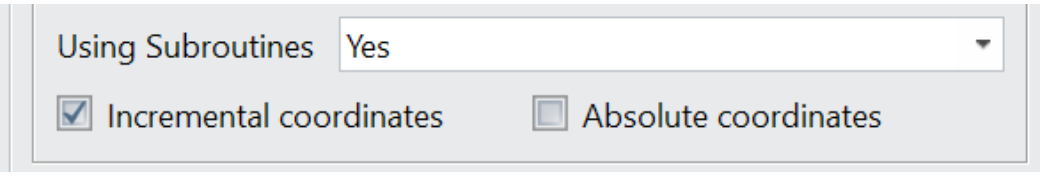
Station priority mode	Operation priority mode
	

- Support output operation in the form of sub-operation

  Sub-operation forms of NC code structures under the two modes are as below:

Sub-operation form under station priority	Sub-operation form under operation priority
	

- Output form under subroutines



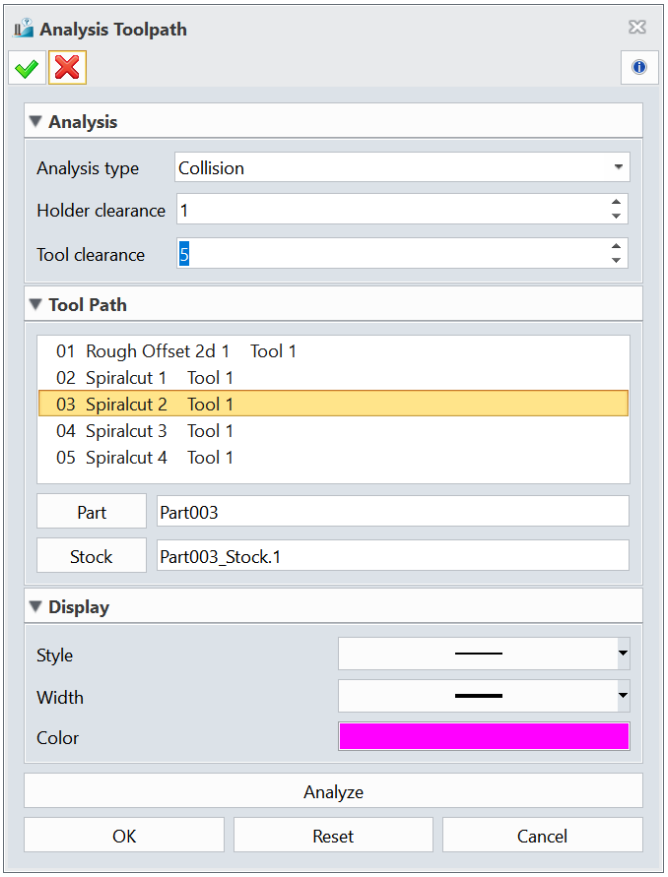
Incremental coordinate——Incremental program. All subsequent coordinates using this instruction are incremental coordinates relative to the previous position

Absolute coordinate——Output in G90, absolute coordinate program.

4.4 Analysis Toolpath Overall Upgrade

The original analysis toolpath had been reformed in the new version of ZW3D including:

- New UI

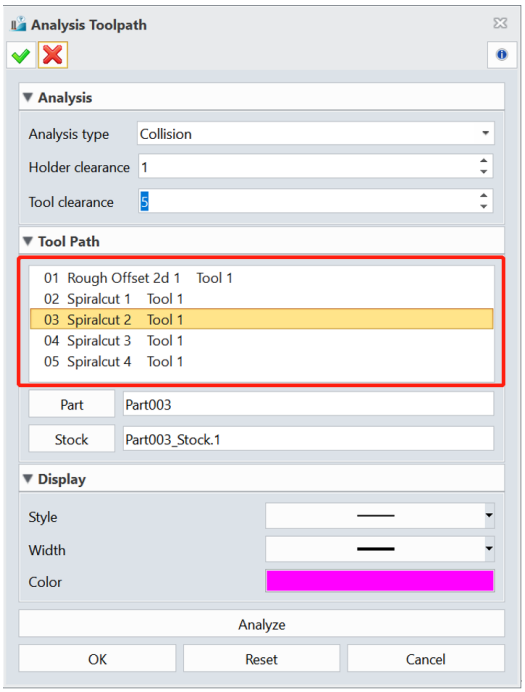


- Support two modes: collision check and gouge check

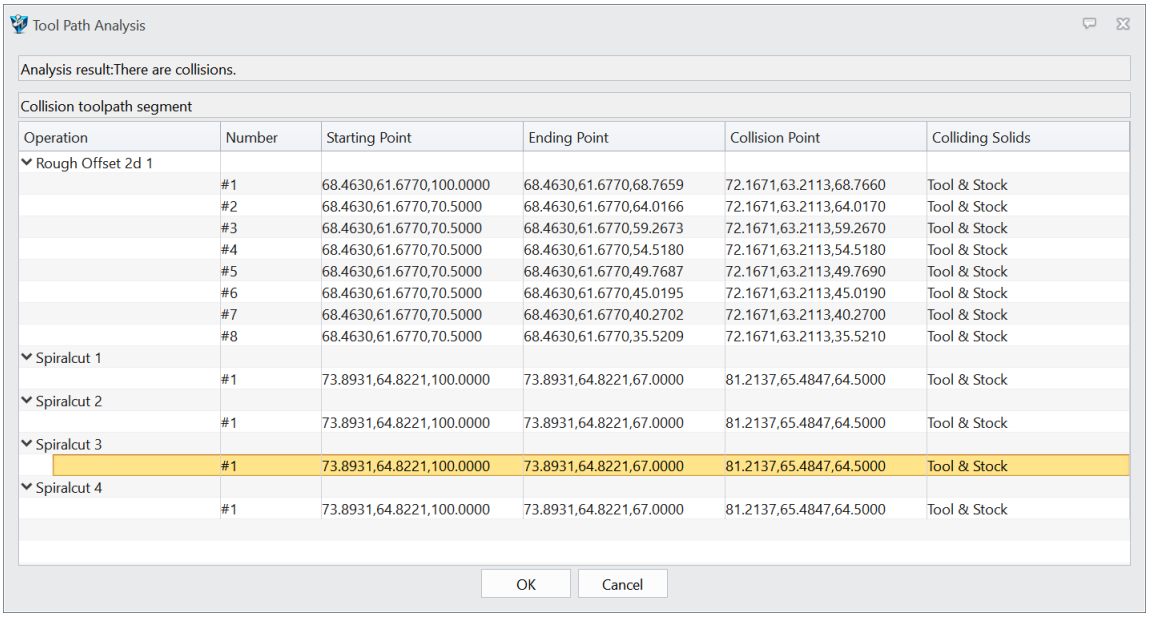
Collision check: During the cutting process, check the collision behavior between tool and holder and process workpiece.

Gouge check: Check the gouge behavior between tool and part.

- Newly support check multiple operations at the same time.

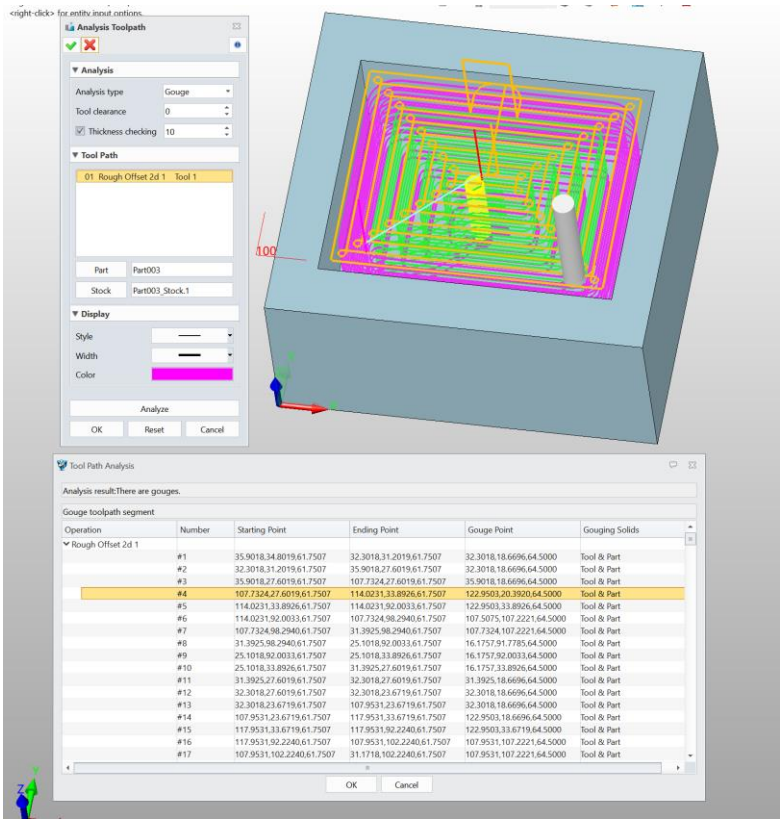


- Newly added the output check result list including operation, collision/gouge number, starting point, ending point, collision/gouge point, collision/gouge type, etc.



Operation	Number	Starting Point	Ending Point	Collision Point	Colliding Solids
Rough Offset 2d 1					
	#1	68.4630,61.6770,100.0000	68.4630,61.6770,68.7659	72.1671,63.2113,68.7660	Tool & Stock
	#2	68.4630,61.6770,70.5000	68.4630,61.6770,64.0166	72.1671,63.2113,64.0170	Tool & Stock
	#3	68.4630,61.6770,70.5000	68.4630,61.6770,59.2673	72.1671,63.2113,59.2670	Tool & Stock
	#4	68.4630,61.6770,70.5000	68.4630,61.6770,54.5180	72.1671,63.2113,54.5180	Tool & Stock
	#5	68.4630,61.6770,70.5000	68.4630,61.6770,49.7687	72.1671,63.2113,49.7690	Tool & Stock
	#6	68.4630,61.6770,70.5000	68.4630,61.6770,45.0195	72.1671,63.2113,45.0190	Tool & Stock
	#7	68.4630,61.6770,70.5000	68.4630,61.6770,40.2702	72.1671,63.2113,40.2700	Tool & Stock
	#8	68.4630,61.6770,70.5000	68.4630,61.6770,35.5209	72.1671,63.2113,35.5210	Tool & Stock
Spiralcut 1					
	#1	73.8931,64.8221,100.0000	73.8931,64.8221,67.0000	81.2137,65.4847,64.5000	Tool & Stock
Spiralcut 2					
	#1	73.8931,64.8221,100.0000	73.8931,64.8221,67.0000	81.2137,65.4847,64.5000	Tool & Stock
Spiralcut 3					
	#1	73.8931,64.8221,100.0000	73.8931,64.8221,67.0000	81.2137,65.4847,64.5000	Tool & Stock
Spiralcut 4					
	#1	73.8931,64.8221,100.0000	73.8931,64.8221,67.0000	81.2137,65.4847,64.5000	Tool & Stock

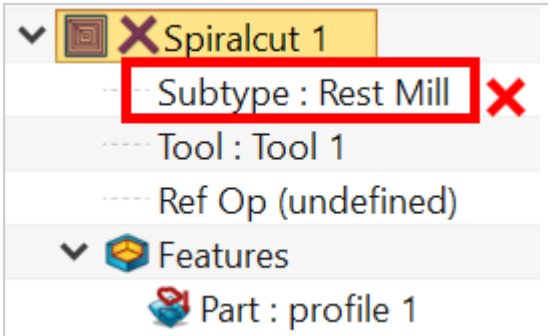
- Support displaying collision/gouge toolpath.



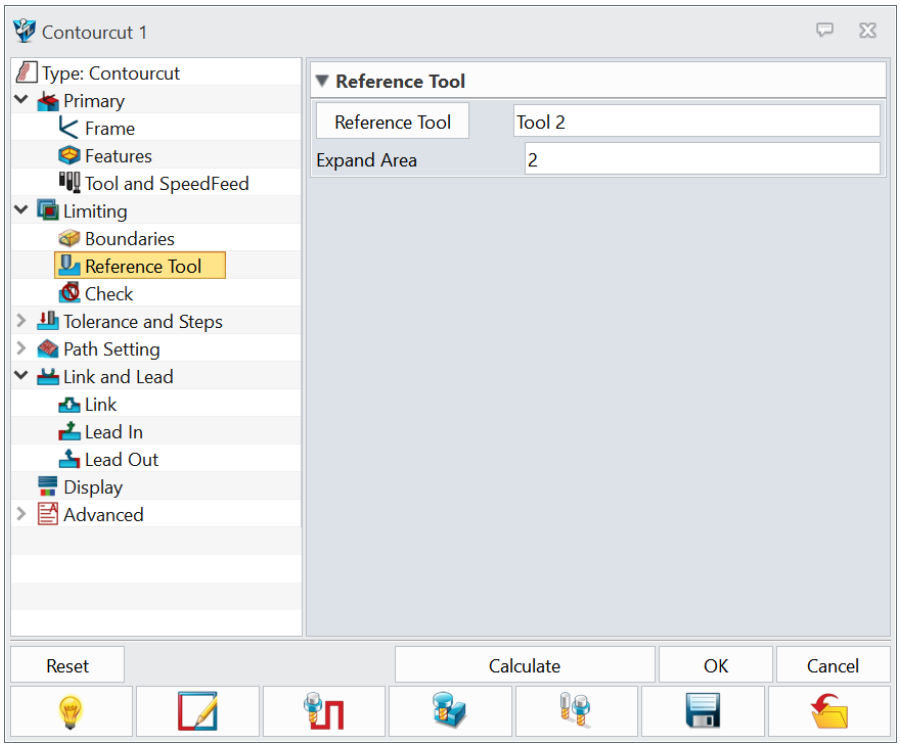
4.5 ★New Reference Tool in 2 Axis Operation

We removed “Rest Mill” subtype in 2 axis operation and added reference tool to generate rest milling tool path in ZW3D 2023.

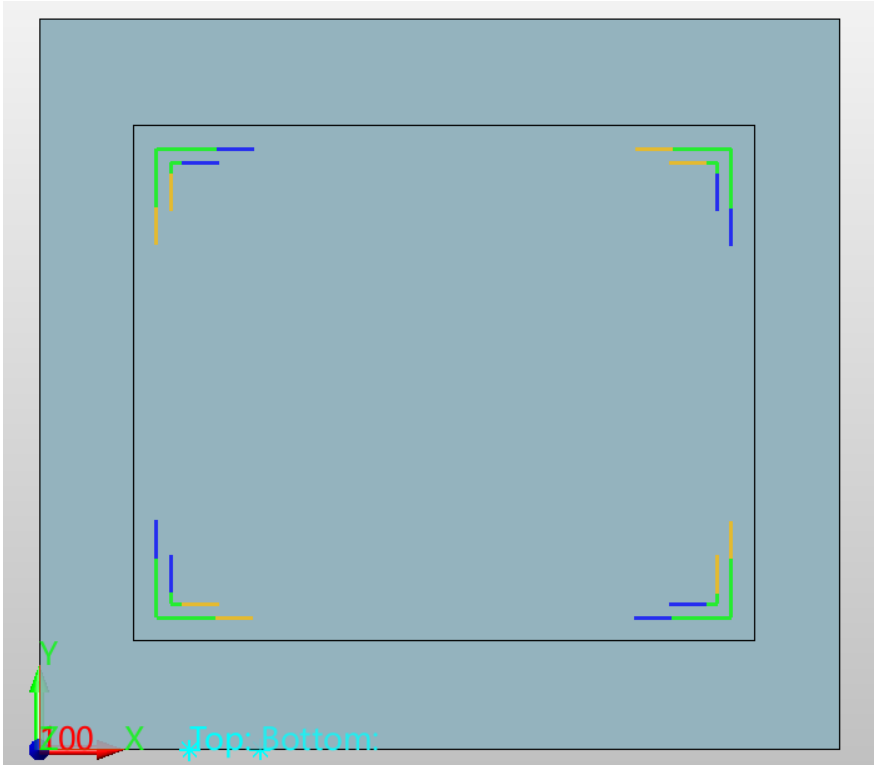
- Remove rest mill subtype in the operation.



- Newly add reference tool which is used to generate rest milling tool path.



The following figure is Contour operation generating rest milling tool path:



4.6 ★FMS Support 5-axis Index Milling Simulation

FMS supports 5-axis index milling machining simulation in ZW3D 2023. Heidenhain, Siemens, Fanuc CNC system and their common index milling instruction are as the below list.

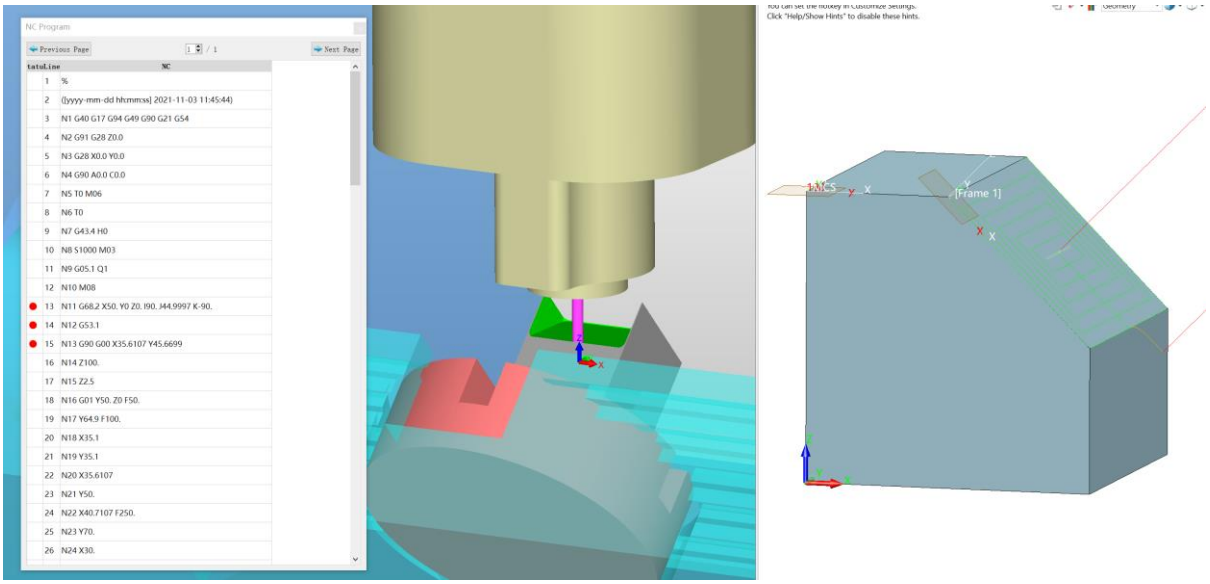
Heidenhain and Fanuc system support two rotation modes, internal and external, separately.

Intrinsic rotation: Revolve around the three axes of object coordinate. During the object revolving, its coordinate axes move around the object.

External rotation: Revolve around the tree axes of world coordinate. During the object revolving, its coordinate axes remain stationary.

CNC	Instruction purpose	FMS supports index milling instruction
Heidenhain	Start index milling (XYZ sequence in extrinsic rotation)	*PLANE SPATIAL SPA30 SPB30 SPC30*
	Start index milling (ZXZ sequence in intrinsic rotation)	*PLANE EULER EULPR90 EULNU45 EULROT0*
	Cancel index milling	*PLANE RESET*
Siemens	Start index milling (XYZ sequence in intrinsic rotation)	*CYCLE800(1,"",0,57,x1,y1,z1,I,J,K,0,0,0,-1)*
	Cancel index milling	*CYCLE800()*
Fanuc	Start index milling (ZXZ sequence in intrinsic rotation)	*G68.2 X50 Y50 Z100 I90 J45 K0*
	Start index milling machining (XYZ sequence in extrinsic rotation)	*G68.2 P1 Q123 X50 Y50 Z100 I90 J45 K0*
	Control tool axis revolution	*G53.1*
	Cancel index milling	*G69*

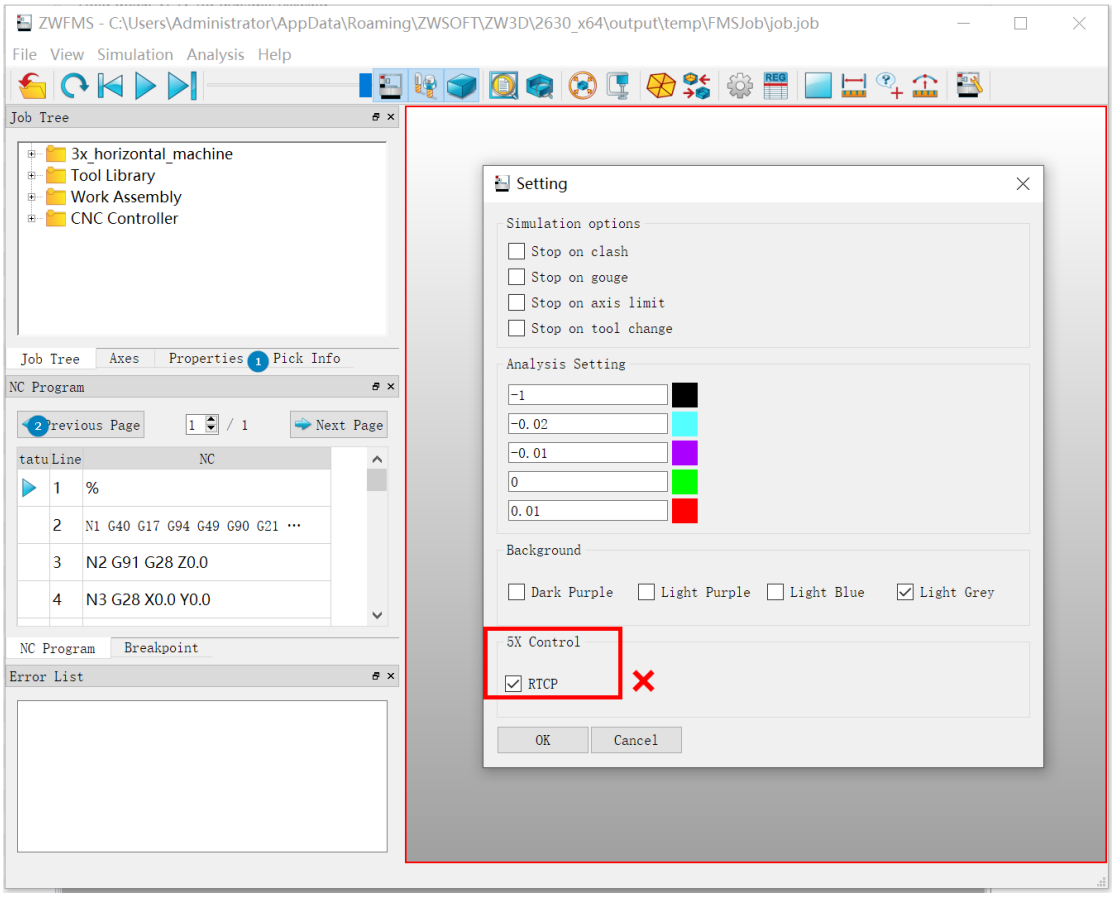
The following figure is Fanuc system index milling FMS:



4.7 ★FMS Support Auto On/Off RTCP

In the old version, when executing 5-axis FMS, user had to start RTCP manually. But in ZW3D 2023, FMS supports opening or close RTCP by CNC instruction.

- Remove RTCP switch in FMS



- Support opening or closing RTCP of Heidenhain, Siemens, Fanuc system CNC.

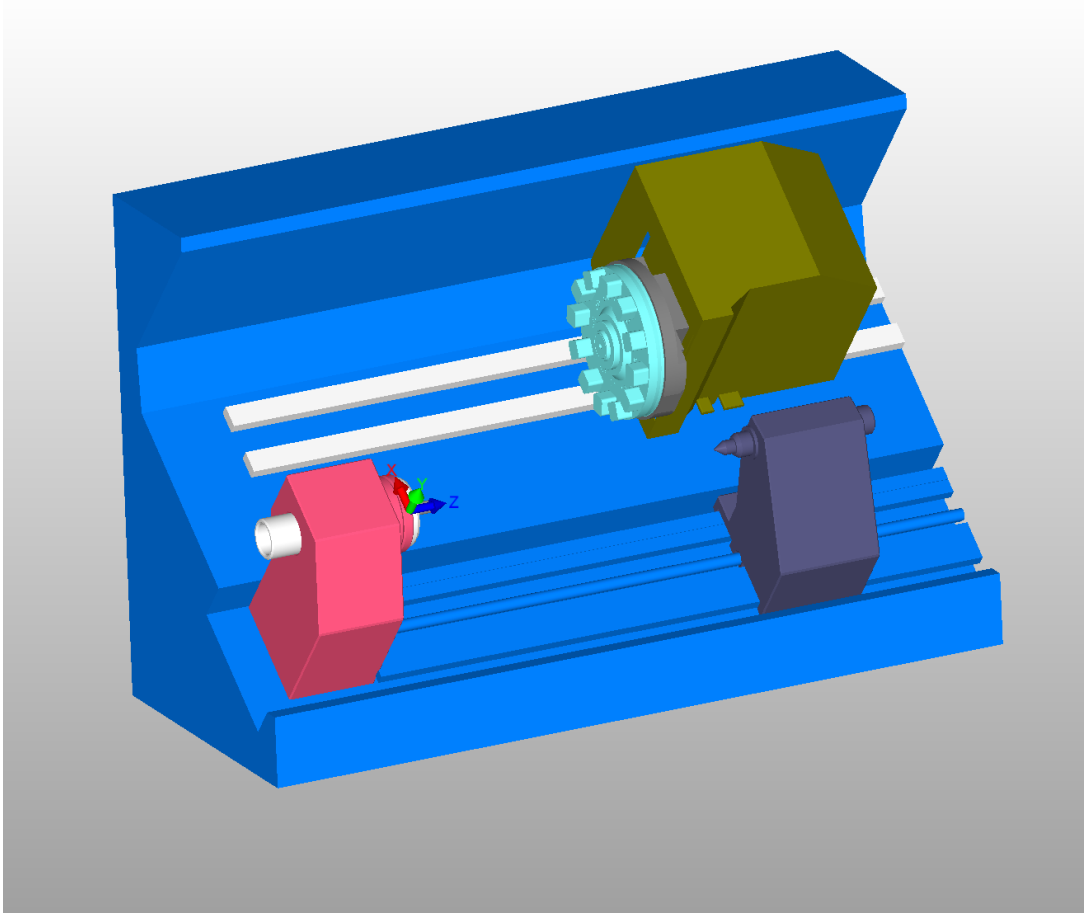
CNC	RTCP start instruction	RTCP close instruction
Fanuc	G43.4	G49
Siemens	TRAORI	TRAFOOF
Heidenhain	M128	M129

4.8 FMS Support Mill-Turn Simulation

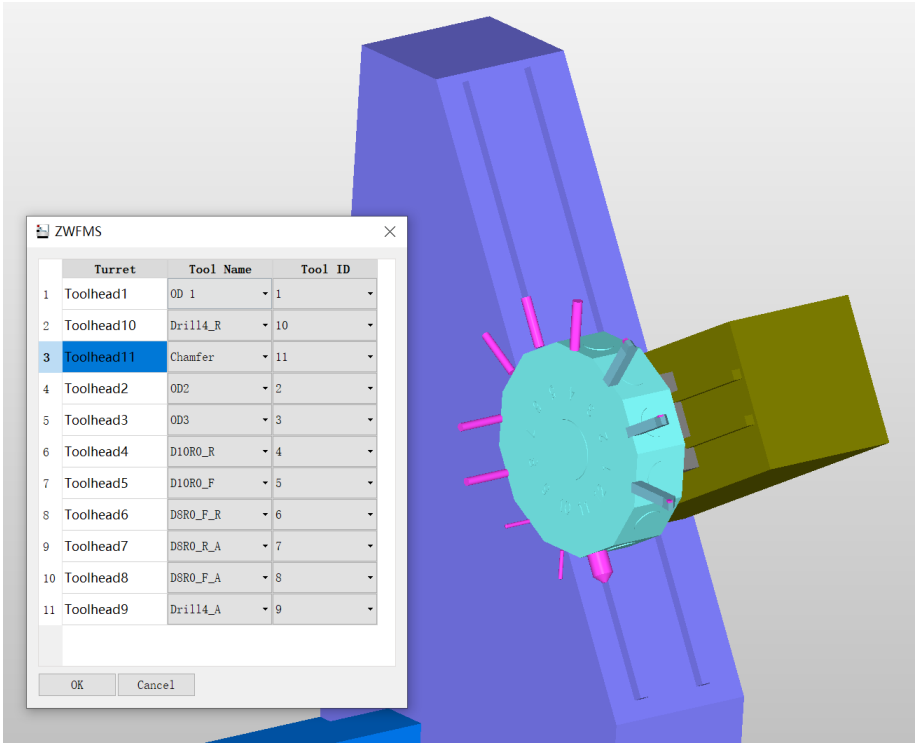
In ZW3D 2023, FMS supports the Mill-Turn machining simulation.

- Add Mill-Turn machine tool case.

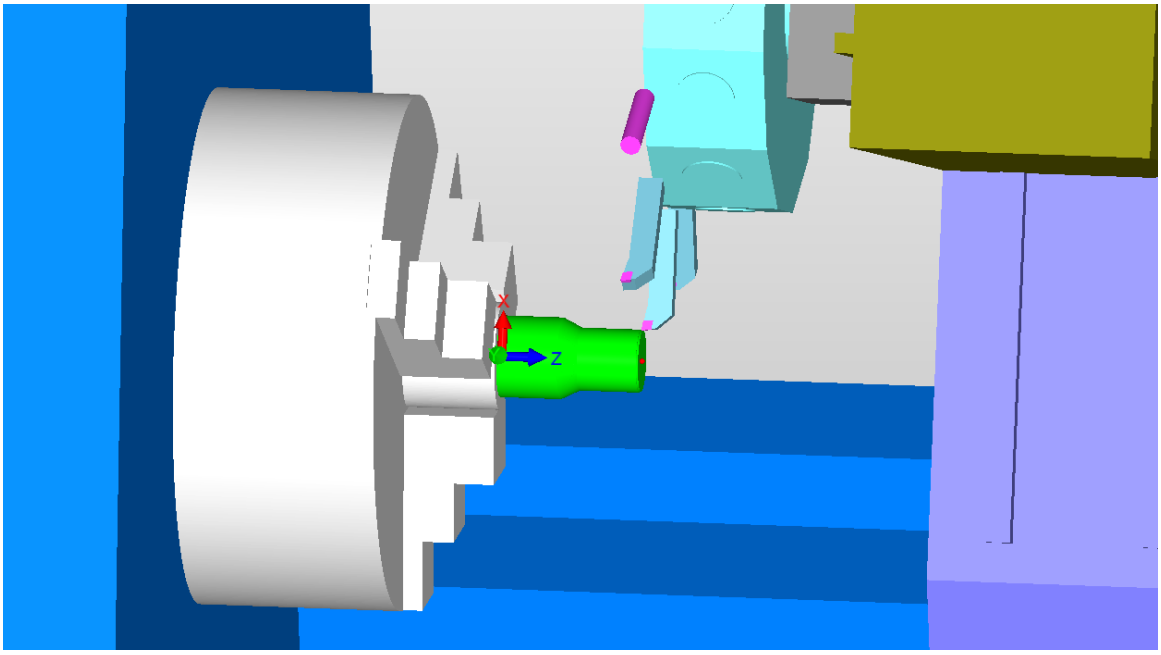
  Add Mill-Turn machine tool: 4x_MillTurn_XYZC_mm, with turret structure.

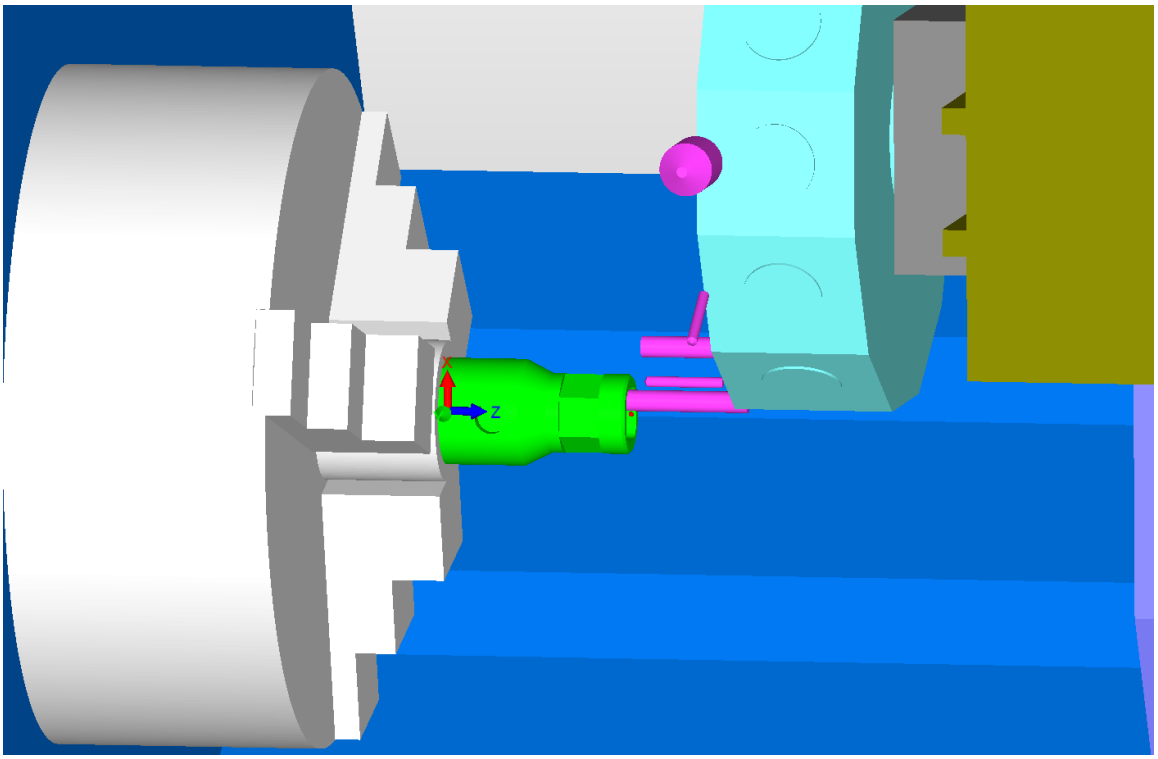


- Add turret setting (Only for Mill-turn machine tool, other type of machines do not need to set)



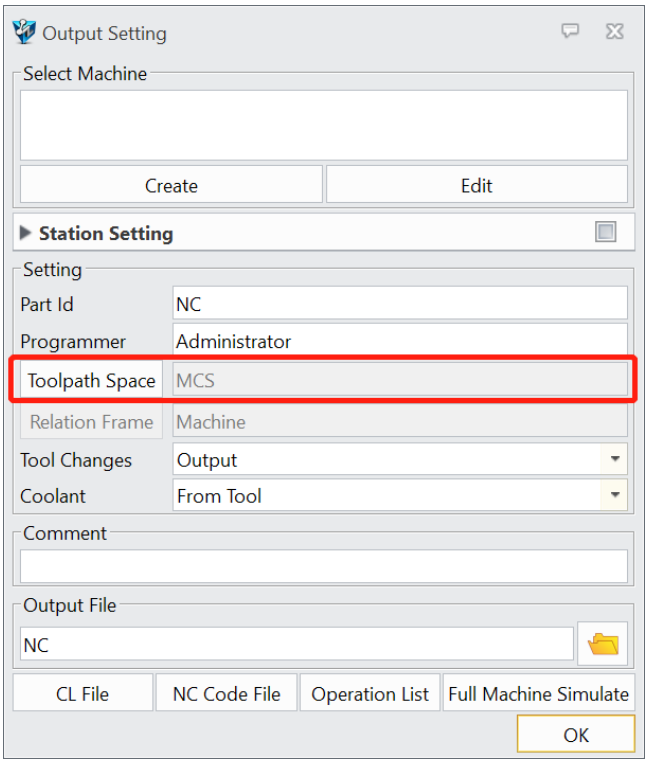
- Support Mill-Turn machining FMS



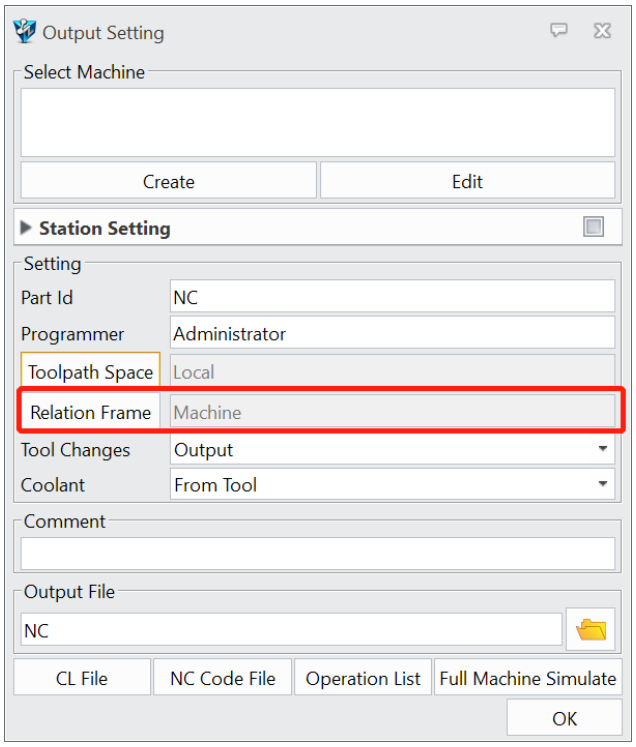


- Output the coordinate change of NC codes and Stl Model files

In the previous ZW3D version, before entering FMS, the system referred to world coordinate to output NC code and Stl model file. In ZW3D 2023, the system outputted NC code and part/stock Stl file according to the coordinate set in Toolpath Space.

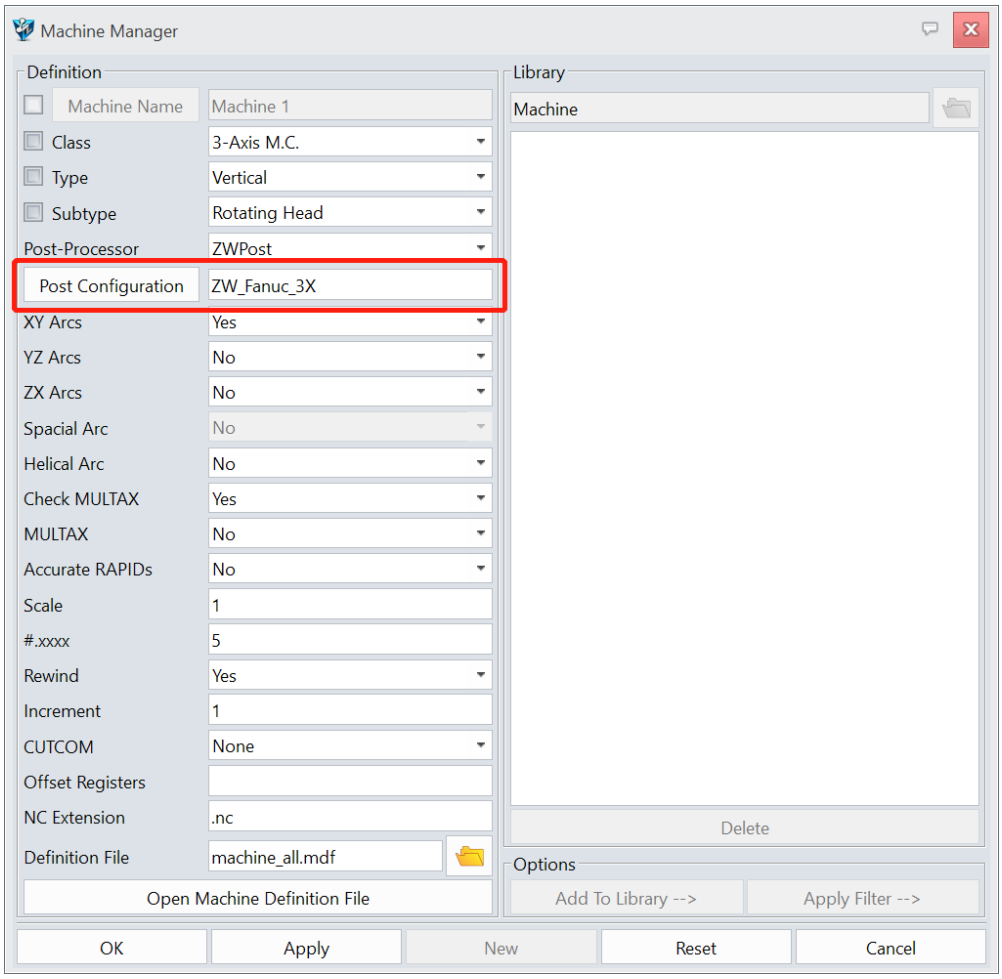


When FMS 5-axis index machining or Mill-Turn machining, Toolpath Space will be set as Local. At this time, it will output part/stock stl file according to Relation Frame. NC code will be output according to the coordinate set in Toolpath Space.



- Support outputting NC code in znc file that set in the device manager.

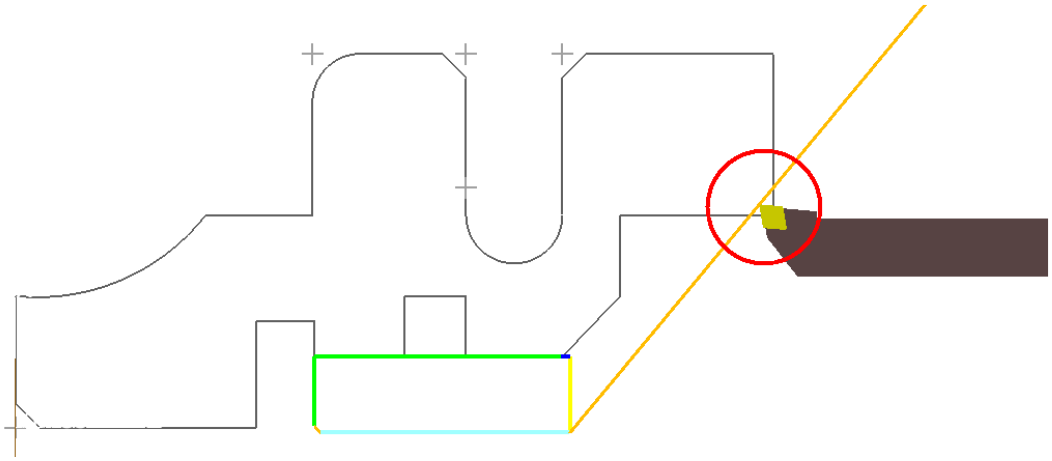
In the previous ZW3D, when user needed to perform FMS, the system would use ZW_FMS_5X.znc and ZW_FMS_Turn.znc files to output NC code by default. In ZW3D 2023, FMS supports device manager set in the post-processing file output NC code for machine tool simulation.



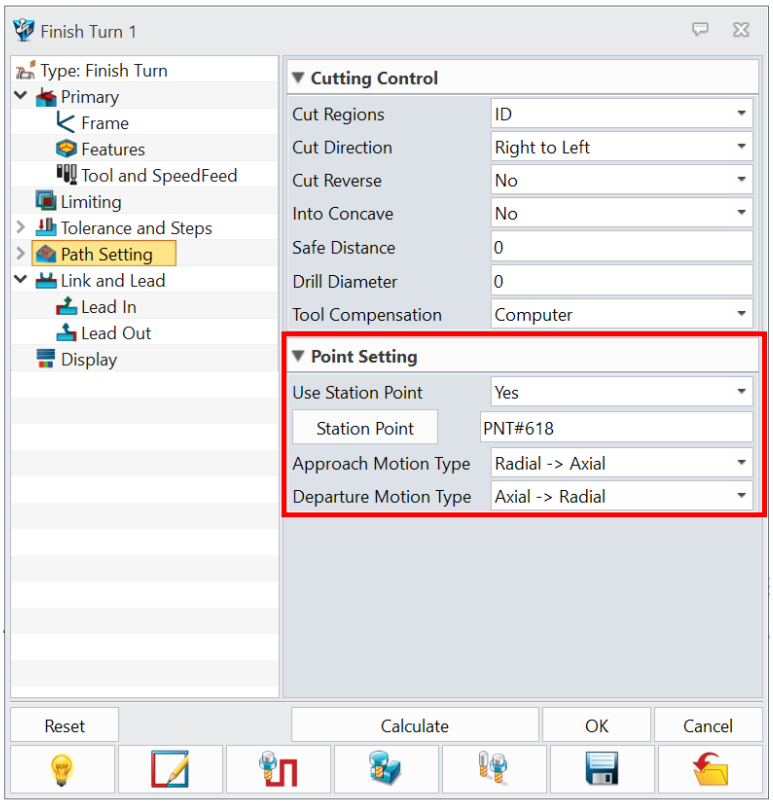
4.9 New Approach/Departure Motion Types in Turning Operation

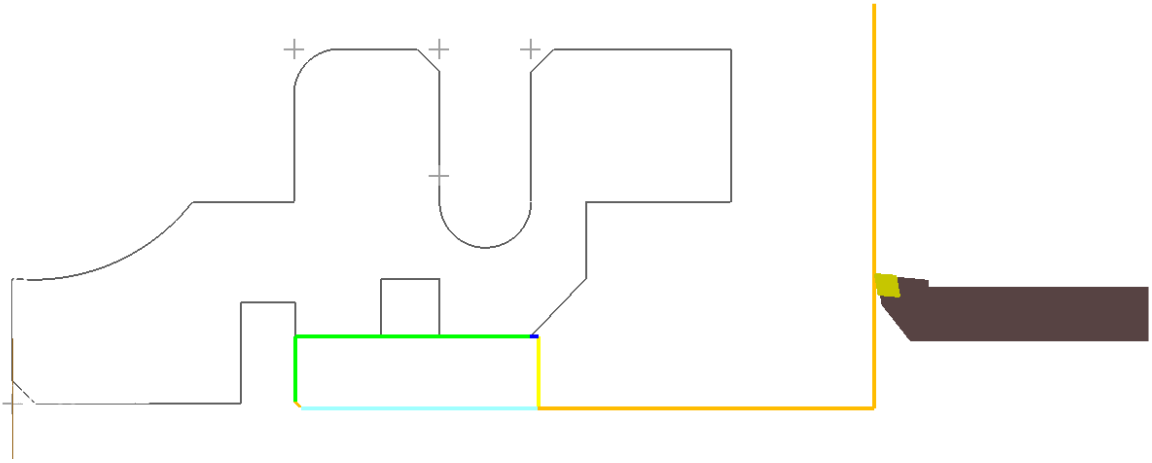
In the old version, from staring point to engage point, there was only one “Direct” approach/departure way. It was easy to cause the tool to collide with the workpiece in the process of movement, which was not conductive to practical use. In ZW3D 2023, we added two options “Radial -> Axial” and “Axial -> Radial” to turning operation, which allowed user to customize tool in the motion way of approach to/departure from workpiece.

In the previous versions, tool could only move in the “Direct” way, which caused collision with workpiece.



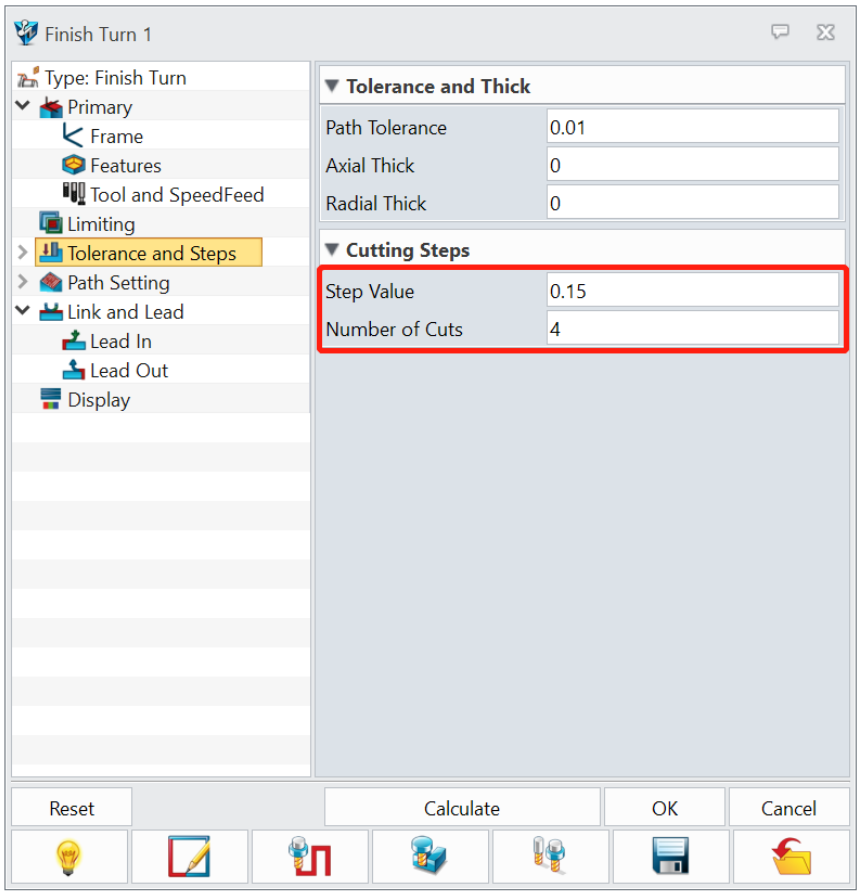
The two options “Radial->Axial” and “Axial -> Radial” in the new version can allow tool to enter or leave workpiece safety.

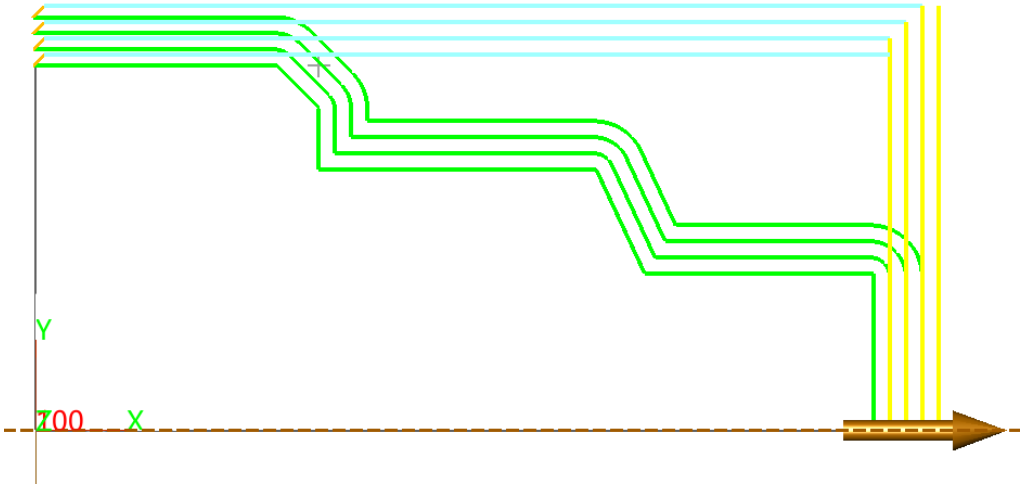




4.10New Multi-level Function in Finish Turing Operation

In ZW3D 2023, we added multi-level to the finish turning operation, so user can generate multi-level finish toolpath on demands.


Finish Turn 1
Type: Finish Turn
Primary
Frame
Features
Tool and SpeedFeed
Limiting
Tolerance and Steps
Path Setting
Link and Lead
Lead In
Lead Out
Display
Tolerance and Thick
Path Tolerance
0.01
Axial Thick
0
Radial Thick
0
Cutting Steps
Step Value
0.15
Number of Cuts
4
Reset
Calculate
OK
Cancel

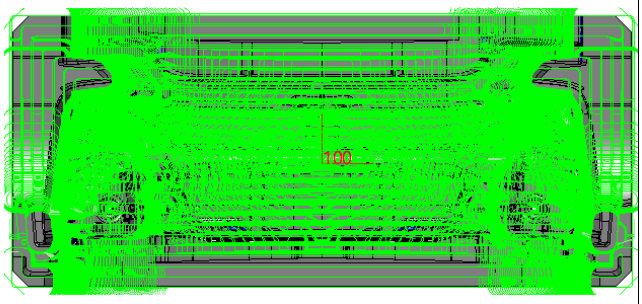

Y
X

4.11 Kernel Algorithm Optimization

4.11.1 ★Speed Up QM Rough Operation Calculation

We applied multithread parallel calculation technique to optimize kernel algorithm and speeded up the generation of QM toolpath based on ZW3D 2022. The QM rough operation calculation speeds up obviously and reduce 70% calculation time, which greatly raises the efficiency.

Calculation Time Comparison		
	ZW3D 2023	ZW3D 2022
	37s	120s

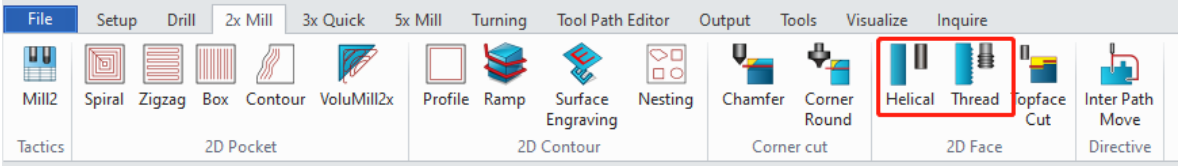


4.12 New Functionalities of Milling

4.12.1 Distinguish Hole Milling and Thread Milling

In previous version, Helical Cut Operation was often utilized in hole milling. User can achieve thread mill cutter and hole milling through selecting hole milling and thread milling, respectively. The two technics are divided into two operations to achieve in ZW3D 2023.

In the new Thread Mill operation, we distinguished Helical Cut operation and Thread operation, which are independently applied to milling internal and external milling, where user can only use thread mill cutter. The original Thread Mill operation is applied to milling hole and cylinder, where user can only use end mill cutter.



Helical Cut operation: Compared with the old version, we removed some uncommonly used parameters during hole milling in Helical Cut operation, such as Max Cut Depth and Z Progress.

Thread Mill operation: It is independently applied to milling thread operation. We added “Rotation” option to define thread based on the original Thread Mill operation. The thread milling operation order in Z progress is controlled by two parameters “Cut Direction” and “Rotation”. When Thread Mill operation generates toolpath, it needs to ensure the thread mill cutter pitch being the same with the hole or boss feature pitch.

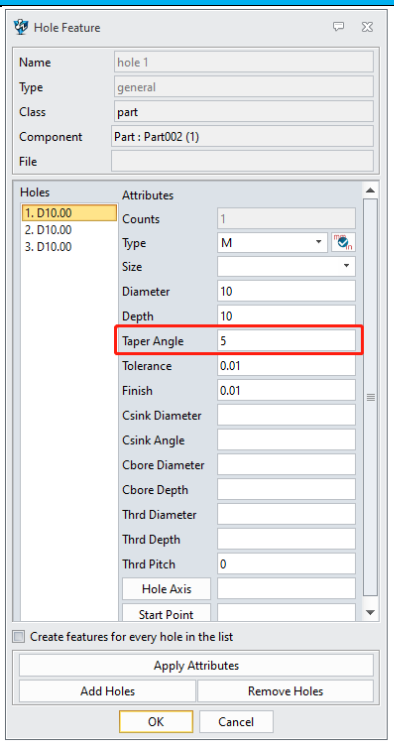
Z progress Operation Order Rule		
Rotation / Cut Direction \ Z Progress	Right Hand	Left Hand
CW	Clockwise, from up to down	Clockwise, from down to up
CCW	Counterclockwise, from down to up	Counterclockwise, from up to down

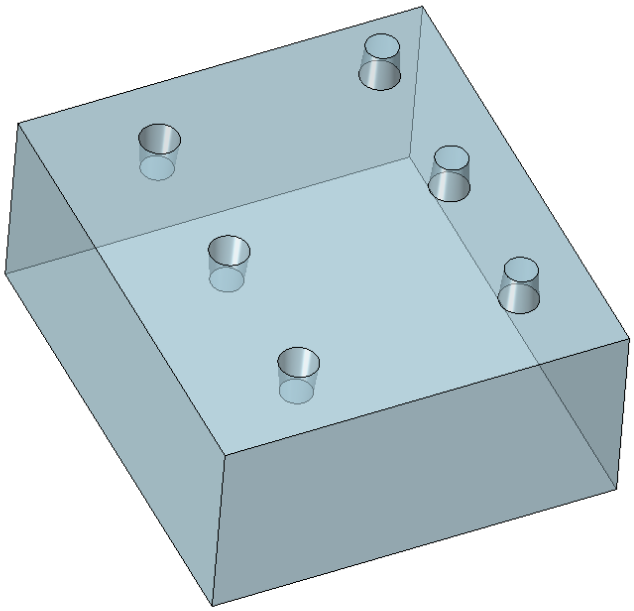
4.12.2 ★Support Taper Hole and Taper Boss Milling

Both hole feature and boss feature support defining taper hole and boss, which allows Helical Cut operation and Thread Mill operation to generate taper milling toolpath in ZW3D 2023.

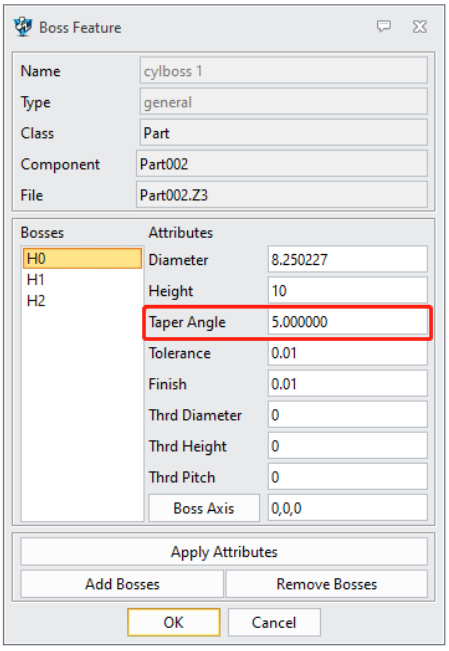
We added “Taper Angle” parameter to both hole feature and boss feature, which is used to record the taper angle of hole or boss, respectively.

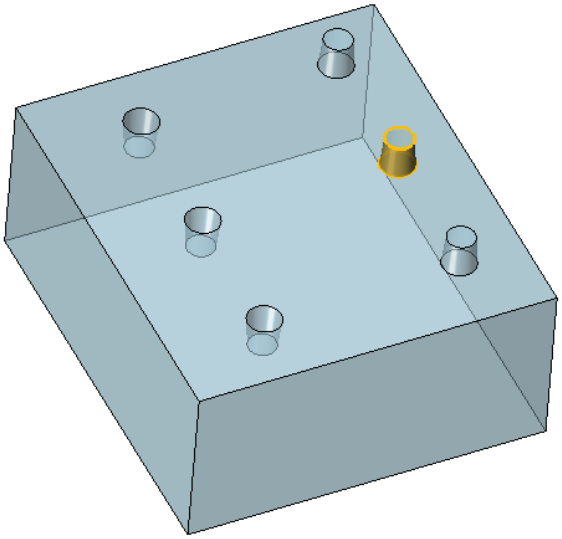
Add “Taper Angle” to Hole Feature



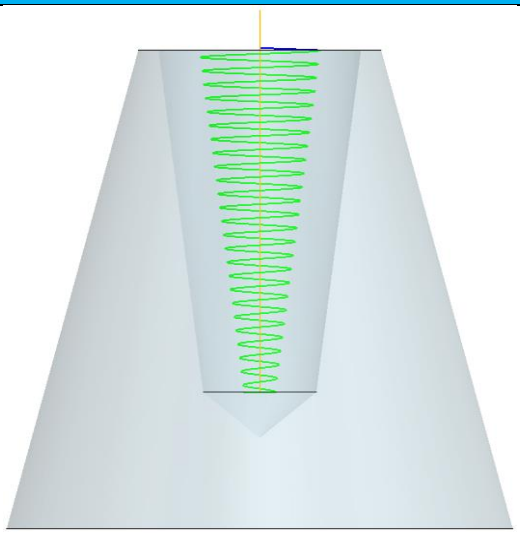
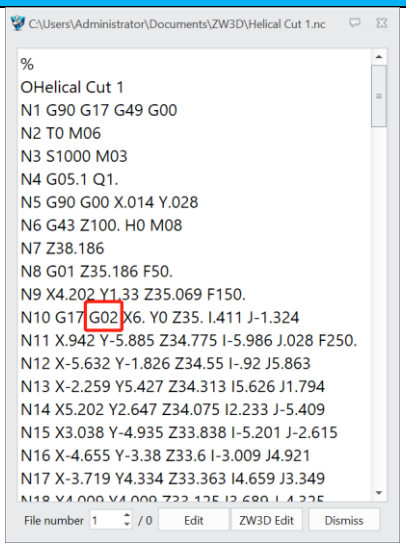
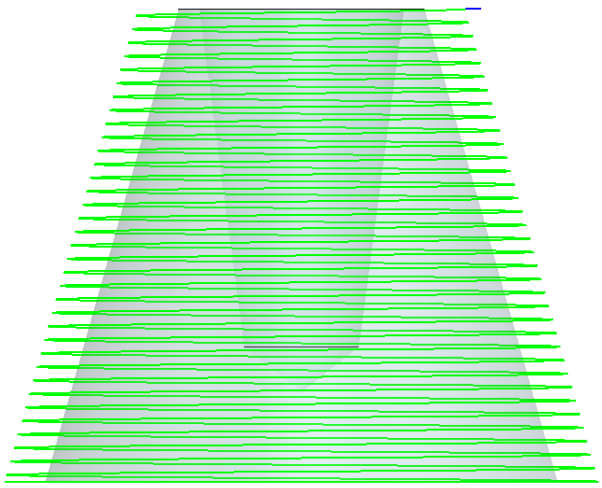
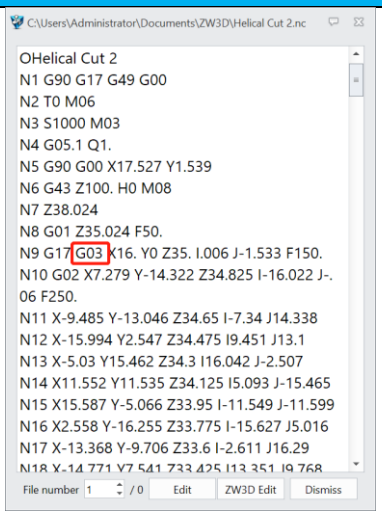


Add “Taper Angle” to Boss Feature



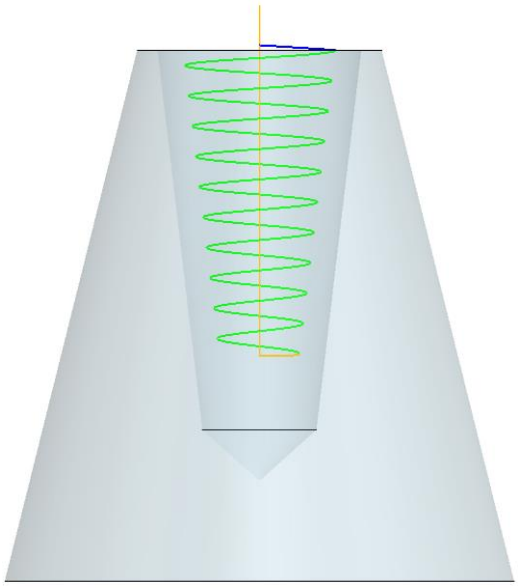


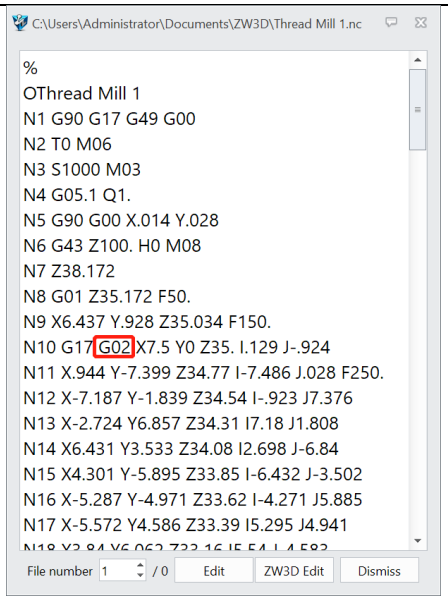
Use Helical Cut operation to generate the milling toolpath of taper hole and taper boss, respectively, and export NC code in the way of arc.

Use Helical Cut operation to mill taper hole, and export toolpath in the way of arc	
	 C:\Users\Administrator\Documents\ZW3D\Helical Cut 1.nc % OHelical Cut 1 N1 G90 G17 G49 G00 N2 T0 M06 N3 S1000 M03 N4 G05.1 Q1. N5 G90 G00 X.014 Y.028 N6 G43 Z100. H0 M08 N7 Z38.186 N8 G01 Z35.186 F50. N9 X4.202 Y1.33 Z35.069 F150. N10 G17 G02 X6. Y0 Z35. I.411 J-1.324 N11 X.942 Y-5.885 Z34.775 I-5.986 J.028 F250. N12 X-5.632 Y-1.826 Z34.55 I-.92 J5.863 N13 X-2.259 Y5.427 Z34.313 I5.626 J1.794 N14 X5.202 Y2.647 Z34.075 I2.233 J-5.409 N15 X3.038 Y-4.935 Z33.838 I-5.201 J-2.615 N16 X-4.655 Y-3.38 Z33.6 I-3.009 J4.921 N17 X-3.719 Y4.334 Z33.363 I4.659 J3.349 File number 1 / 0 Edit ZW3D Edit Dismiss
Use Helical Cut operation to mill taper boss, and export toolpath in the way of arc	
	 C:\Users\Administrator\Documents\ZW3D\Helical Cut 2.nc OHelical Cut 2 N1 G90 G17 G49 G00 N2 T0 M06 N3 S1000 M03 N4 G05.1 Q1. N5 G90 G00 X17.527 Y1.539 N6 G43 Z100. H0 M08 N7 Z38.024 N8 G01 Z35.024 F50. N9 G17 G03 X16. Y0 Z35. I.006 J-1.533 F150. N10 G02 X7.279 Y-14.322 Z34.825 I-16.022 J-.06 F250. N11 X-9.485 Y-13.046 Z34.65 I-7.34 J14.338 N12 X-15.994 Y2.547 Z34.475 I9.451 J13.1 N13 X-5.03 Y15.462 Z34.3 I16.042 J-2.507 N14 X11.552 Y11.535 Z34.125 I5.093 J-15.465 N15 X15.587 Y-5.066 Z33.95 I-11.549 J-11.599 N16 X2.558 Y-16.255 Z33.775 I-15.627 J5.016 N17 X-13.368 Y-9.706 Z33.6 I-2.611 J16.29 File number 1 / 0 Edit ZW3D Edit Dismiss

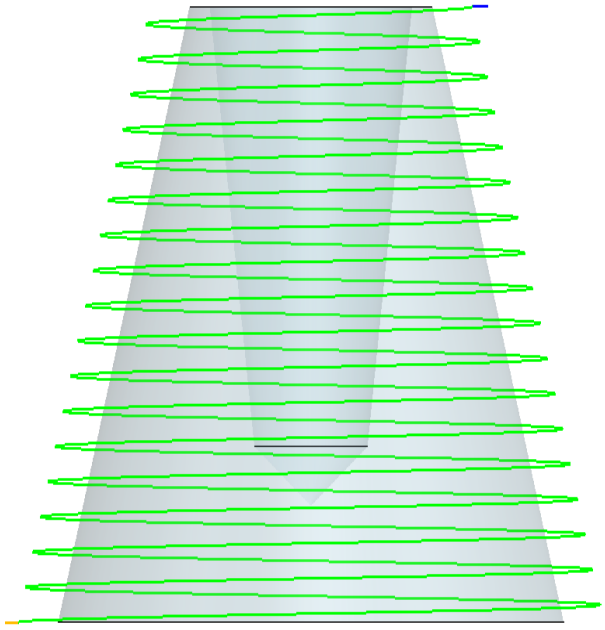
Use Thread Mill operation to generate the milling thread toolpath of taper hole and taper boss.

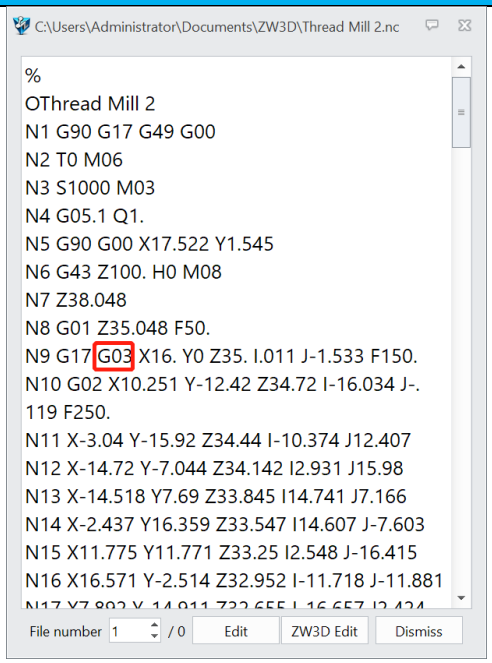
Use Thread Mill operation to mill internal thread of taper hole, and export toolpath in the way of arc





Use Thread Mill operation to mill external thread of taper boss, and export toolpath in the way of arc





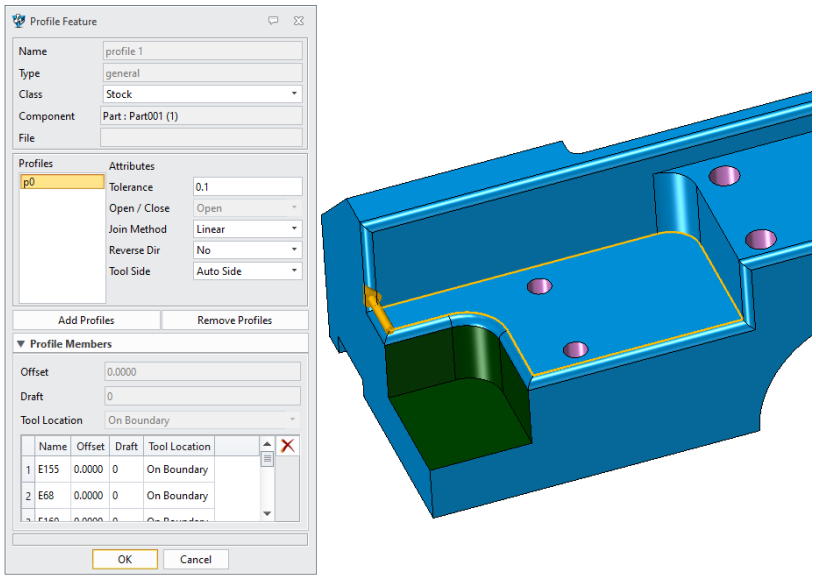
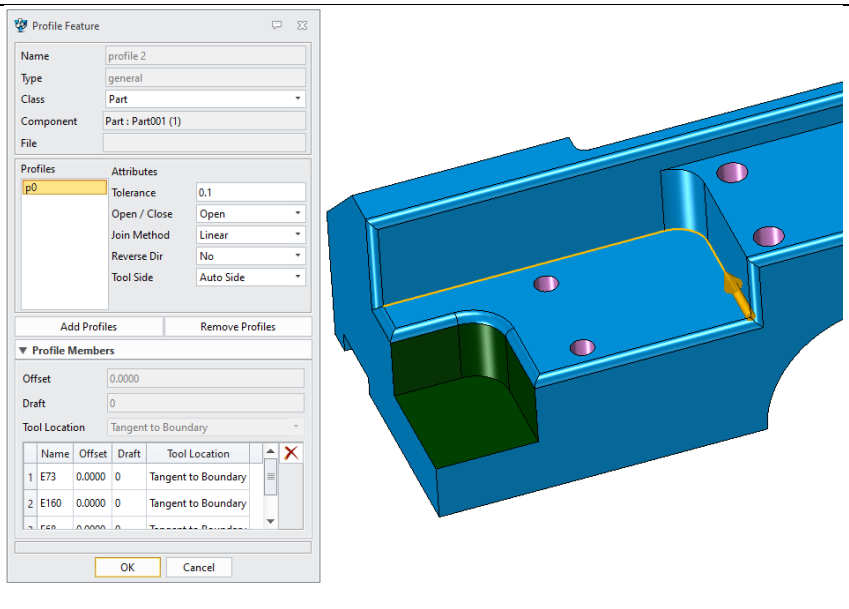
4.12.3 ★ 2D Pocket Operation Supports External Lead-in Open Pocket

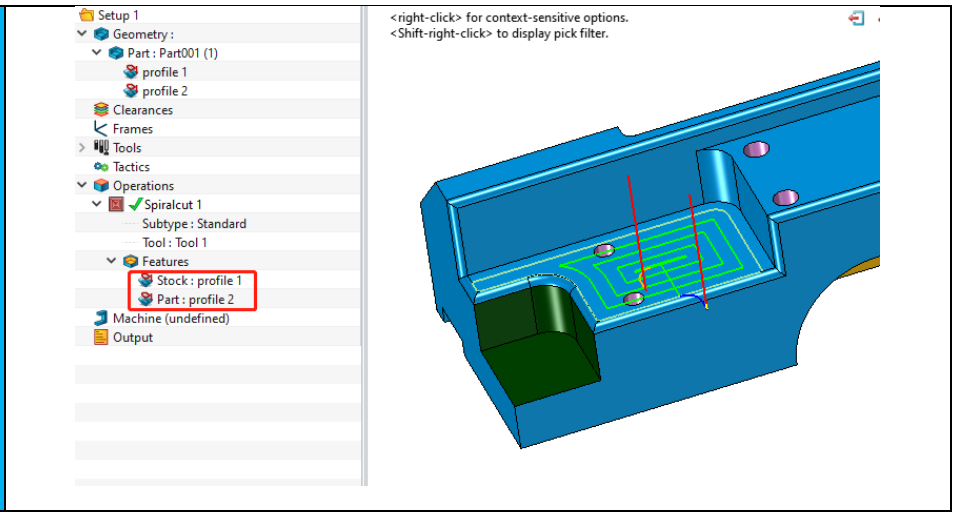
As for 2D pocket operations, when user defines the shape of open pocket through the type (stock, part) of Profile feature in ZW3D 2023, the system will automatically work out the processing range and

find out the part boundary (profile of stock type) that can be crossed and then start to lead in on the boundary, achieving the external lead-in of open pocket. When define stock profile, the selected profile must be closed; when define part profile, the selected profile can be open or closed. Processing from open pocket's external lead-in can reduce tool wear and reduce production cost.

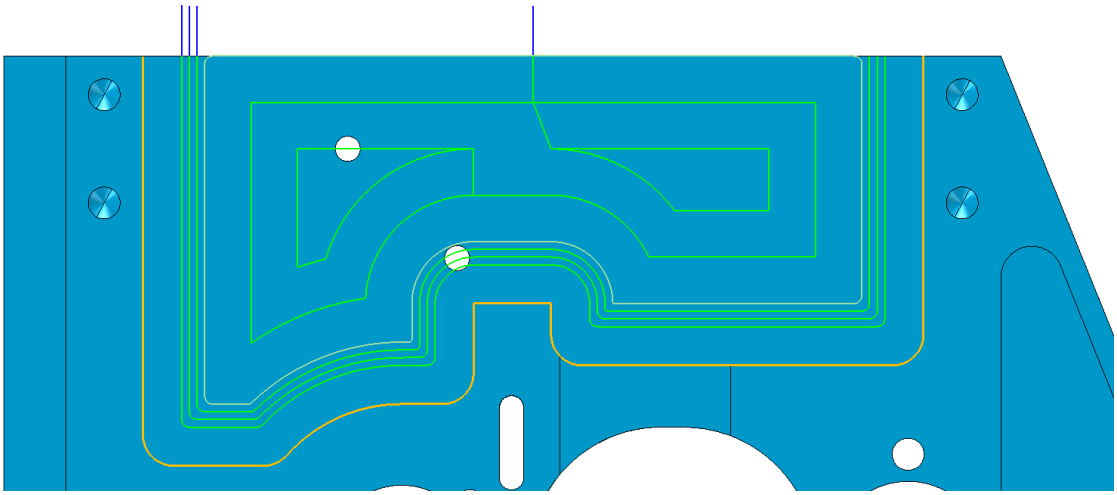
"Stock" type profile——Tool can pass through the profile

"Part" type profile——Tool cannot touch the profile

Define closed stock profiles	
Define open part profiles	

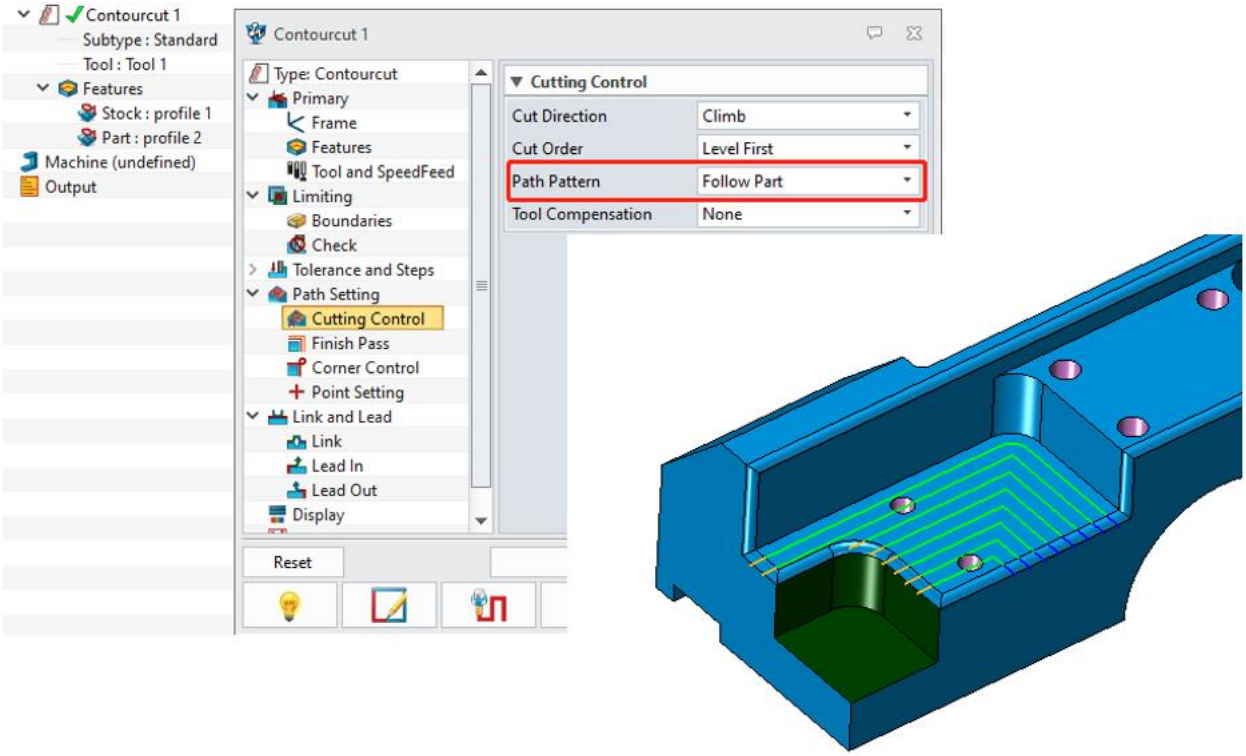
The system identifies the shape of open pocket and generate the toolpath style of external lead-in	

In the new version, when 2D pocket operation defines both “Stock” and “Part” profile features, the cleanup toolpath of operation can only be generated on the boundary of “Part” type. As the following figure shows, the Multiple levels cleanup can only be generated on the profile boundary of “Part” type to reduce the realistic processing time.



4.12.4 ★New Follow Part Path Pattern in 2X Contourcut operation

In ZW3D 2023, on the base of 2X pocket operation supporting open pocket external lead-in, we added “Follow Part” path pattern in the contourcut operation. As “Part” type profile works on offset to generate toolpath, it can fulfill in the entire open pocket so that to achieve more efficient and safter pocket rough operation.

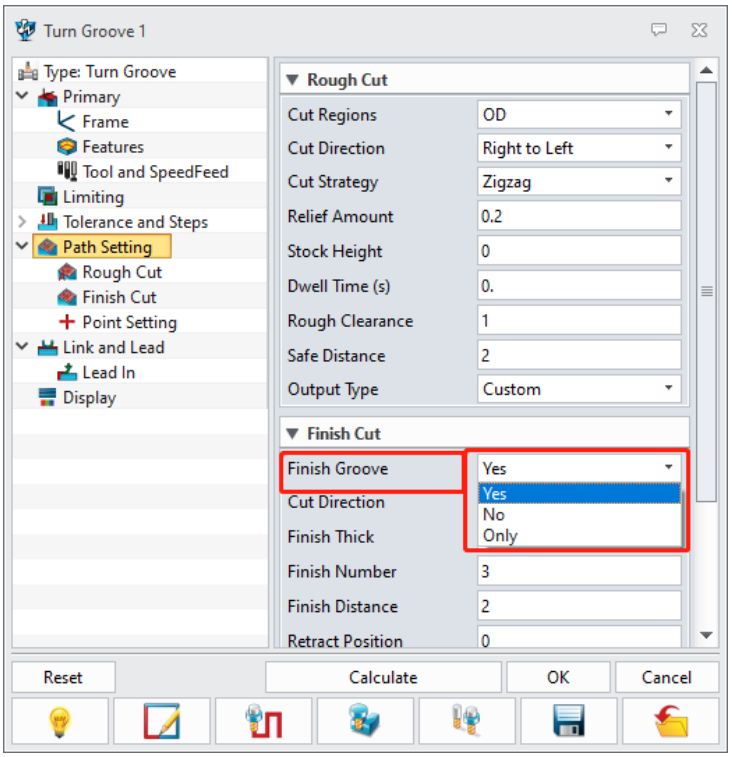


4.13 New Functionalities in Turning Model

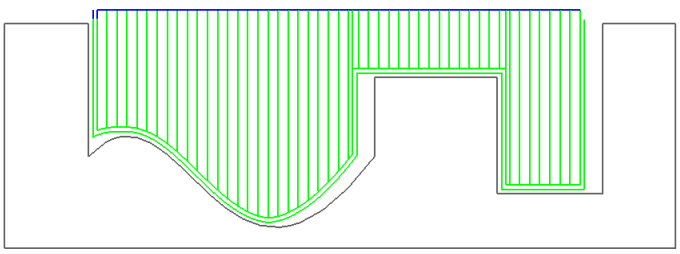
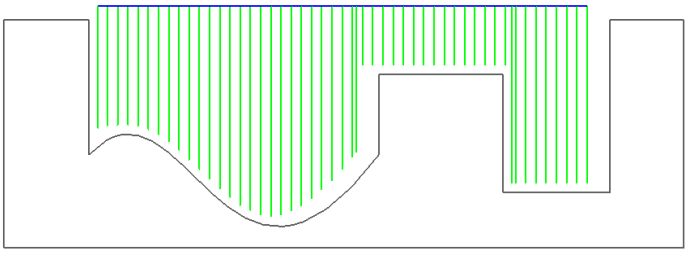
4.13.1 Finish Operation in Groove Operation Optimization

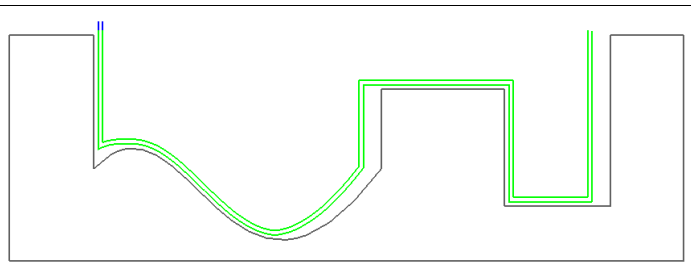
In the new version, we optimized the Groove operation in the turning model and added “Only” Finish Groove, Cut Direction, Finish Number, Finish Distance, to meet user’s actual need.

We added “Only” to the finish groove operation, which provides more processing strategy.



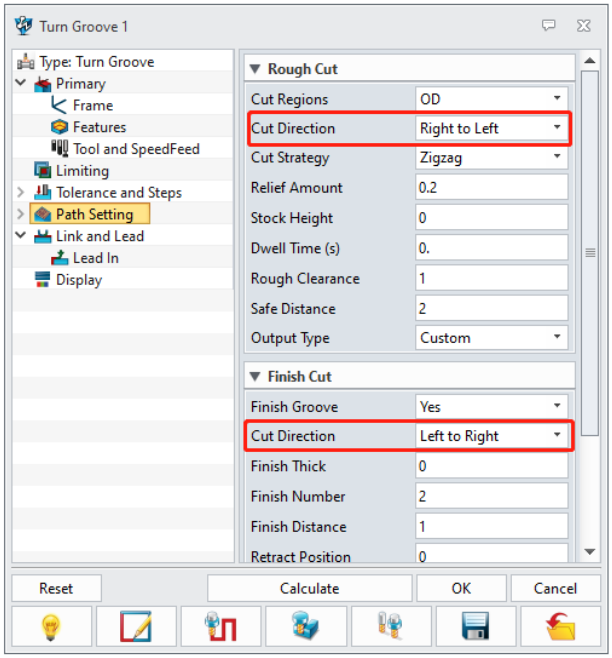
The available options of finish operation groove include the following:

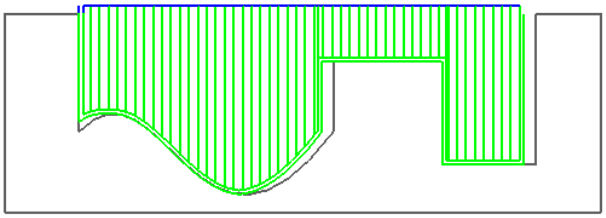
Finish Operation Groove Option	Toolpath Style Sample
"Yes" — — Groove operation generates rough groove and finish groove toolpath	
"No" — — Groove operation only generates rough groove toolpath	

“Only” — — Groove operation only generates finish groove toolpath	

We added the parameters “Cut Direction” “Finish Number” and “Finish Distance” to the finish groove, which can be used to control the style of finish groove toolpath.

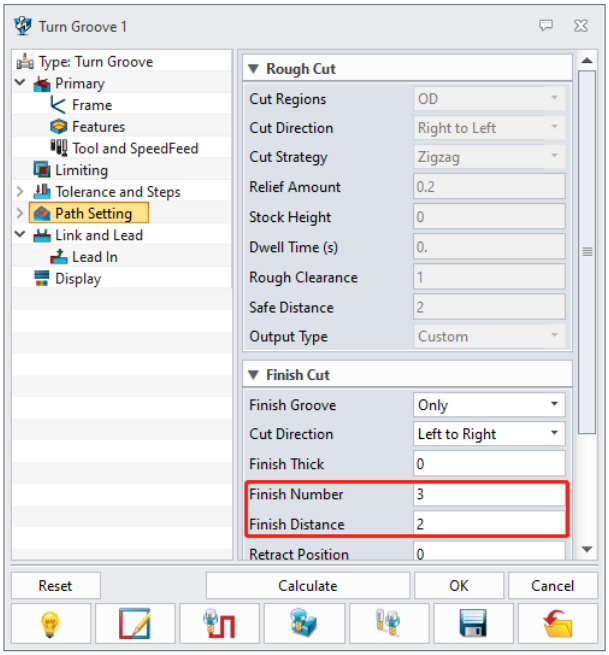
Cut Direction—— Use to control the processing direction of finish groove toolpath. The Finish groove direction is the oppositive to the rough groove direction, as the following figure shows:

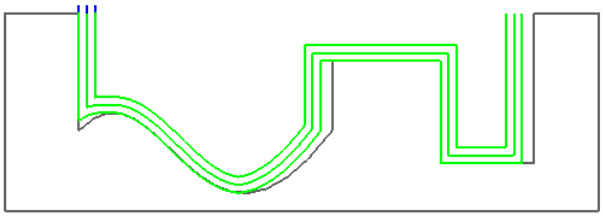




Finish Number——Use to set the finish groove number

Finish Distance——Use to set the distance between each finish groove toolpath





4.14 Workflow and Usability Optimization

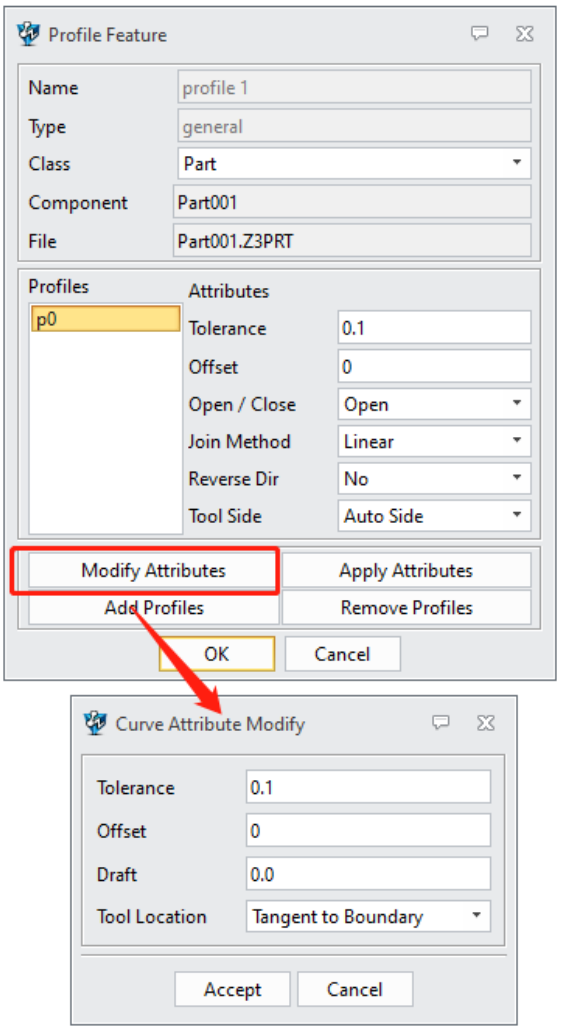
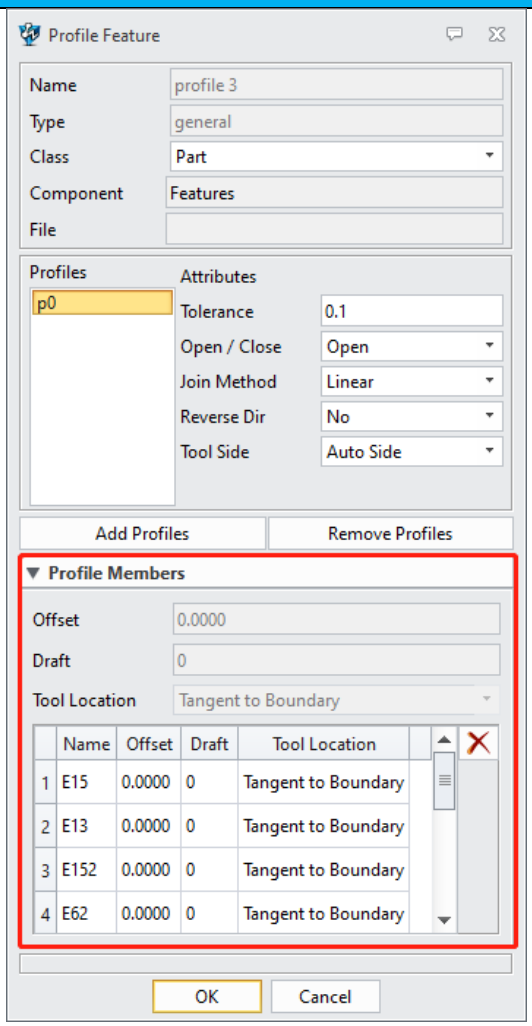
4.14.1 ★Profile Feature Curve Attribute Optimization

In this version, we optimized the functionality of curve attribute of Profile feature and added the curve attribute interface that was not easy to trigger to the profile feature dialogue, which was used to set the position of toolpath on each boundary and remove the “Tool Location” functionality from 2X operation and control each boundary position of toolpath in the profile feature dialogue. The tool position includes 3 types: on boundary, tangent to boundary, and past boundary.

In the expansion menu for profile members, you can set the boundary property for each member of the selected profile --- offset, draft, and tool location.

The tool position options include on boundary, tangent to boundary, past boundary.

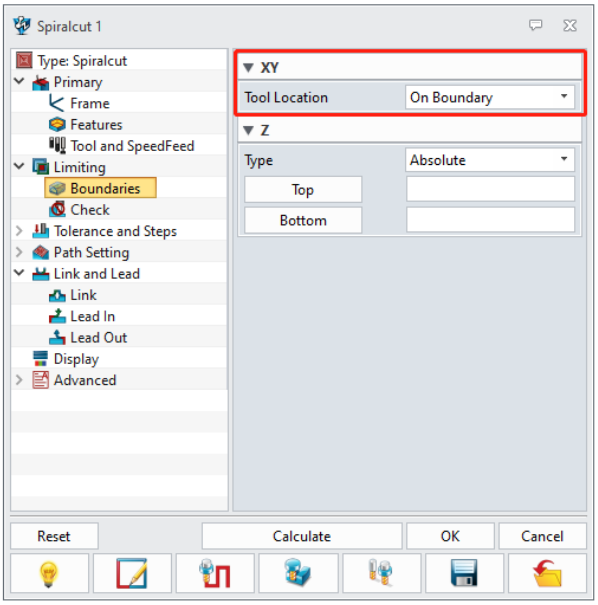
The new profile boundary attribute interface will be put in the same window with the profile feature for more direct modification. It can achieve toolpath of different processing ranges and different styles by setting different boundary attribute and satisfy user’s requirement in different cases.

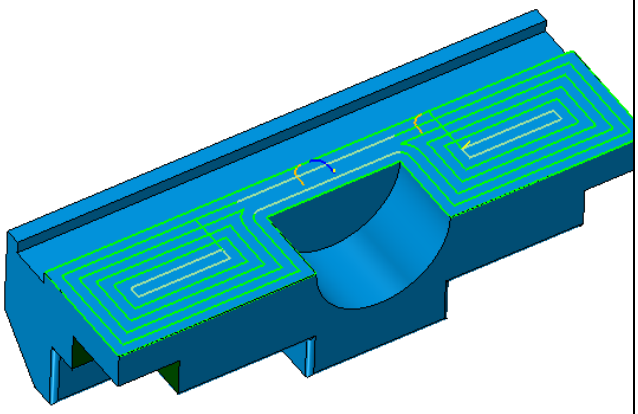
2022	2023
	

Profile Feature
Name
profile 1
Type
general
Class
Part
Component
Part001
File
Part001.Z3PRT
Profiles
p0
Attributes
Tolerance
0.1
Offset
0
Open / Close
Open
Join Method
Linear
Reverse Dir
No
Tool Side
Auto Side
Modify Attributes
Apply Attributes
Add Profiles
Remove Profiles
OK
Cancel
Curve Attribute Modify
Tolerance
0.1
Offset
0
Draft
0.0
Tool Location
Tangent to Boundary
Accept
Cancel

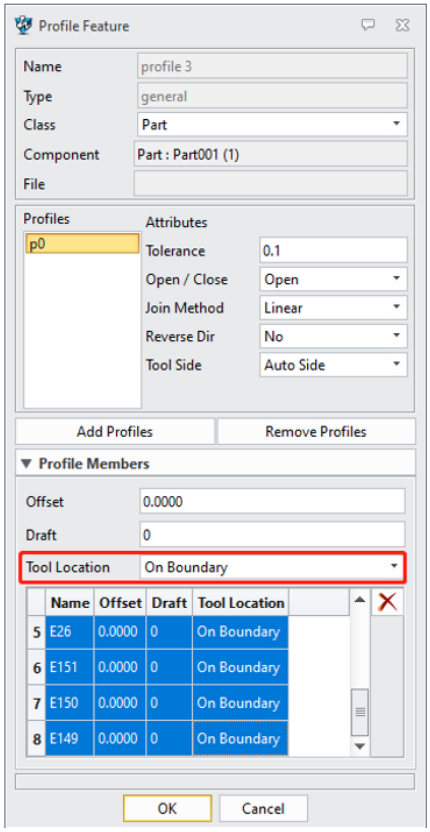
Profile Feature
Name
profile 3
Type
general
Class
Part
Component
Features
File
Profiles
p0
Attributes
Tolerance
0.1
Open / Close
Open
Join Method
Linear
Reverse Dir
No
Tool Side
Auto Side
Add Profiles
Remove Profiles
Profile Members
Offset
0.0000
Draft
0
Tool Location
Tangent to Boundary
Name
Offset
Draft
Tool Location
1 E15 0.0000 0 Tangent to Boundary
2 E13 0.0000 0 Tangent to Boundary
3 E152 0.0000 0 Tangent to Boundary
4 E62 0.0000 0 Tangent to Boundary
OK
Cancel

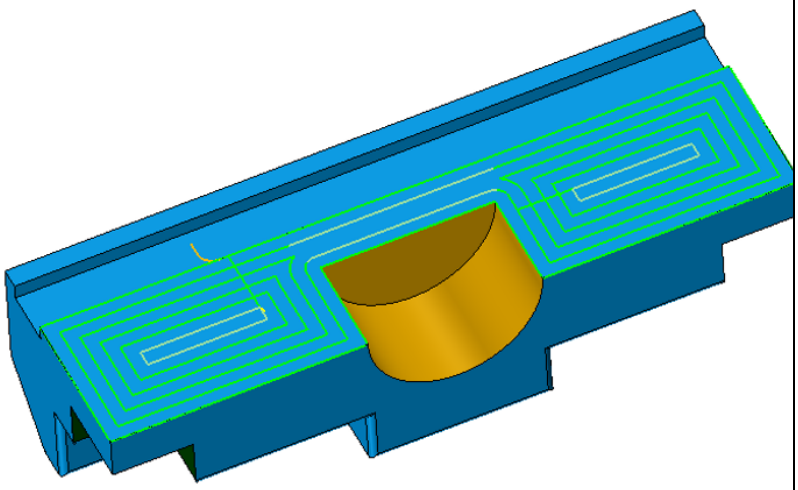
In ZW3D 2022, control tool position on the profile through the tool position in operation





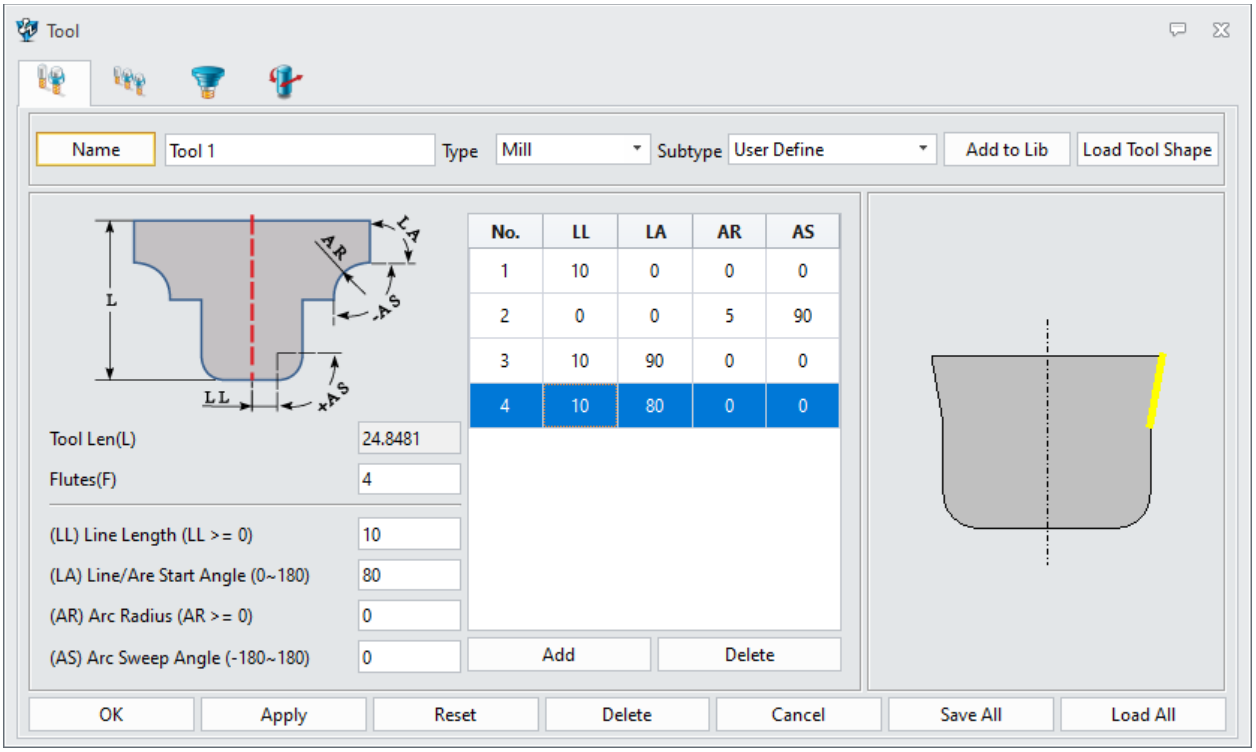
In ZW3D 2023, control tool position in profile through profile feature





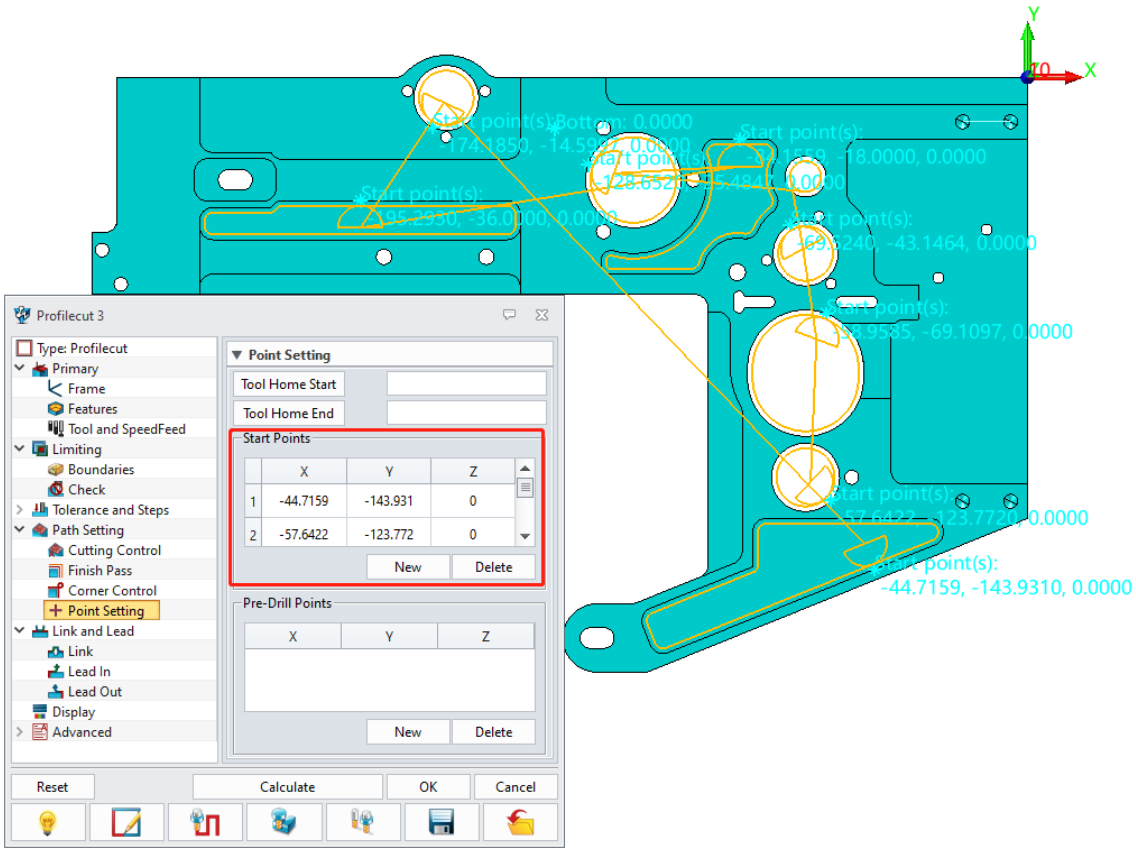
4.14.2 ★Update User Customized Tool Functionality

In ZW3D 2023, we updated the user's customized tool functionality and designed a full new operation interface with adding clear parameter samples and applying the table method to manage tool profile parameter. Meanwhile, the right-side tool display interface interacts with the table data. Select any row data in the table and highlight the corresponding tool profile simultaneously. The table supports double-clicking to edit parameters, which is convenient for user to modify tool parameter and improves work efficiency.



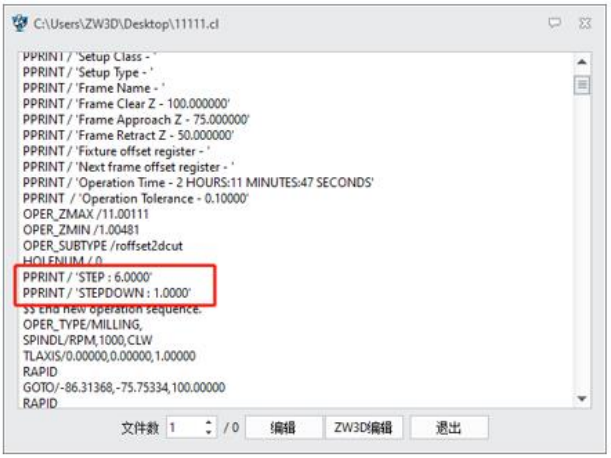
4.14.3 Optimized Start Point, Pre-Drill Point and Synchro Z Level in Operation

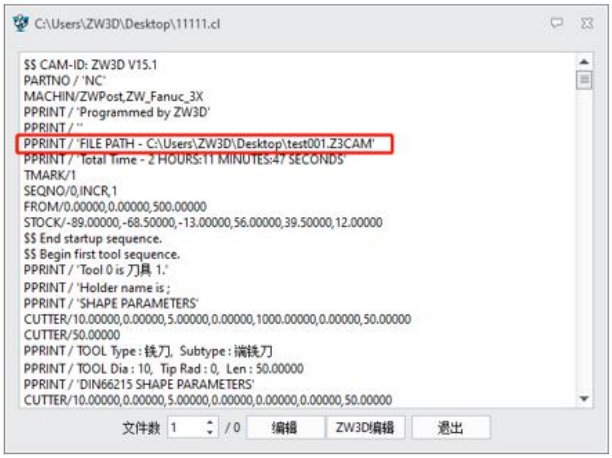
In ZW3D 2023, we optimized the point setting interface of the operation including 2X/3X operation start point, pre-drill point and synchro Z level and utilized table control to manage point data, making each point parameter more direct. The new version supports setting more start points and pre-drill points, which can be effectively applied to each independent processing areas. Meanwhile, the functionalities of "New" and "Delete" allows user to quickly edit point data. As the following figure shows, each processing area can be independently set a start point.



4.14.4 CL New Export Variables in Files

ZW3D 2023 supports “Stepdown”, “Step” and NC file path in export operation in CL files through configurating .mdf files to do that. It is convenient for user to search the cutting parameter of toolpath from CL file and export file storage path.



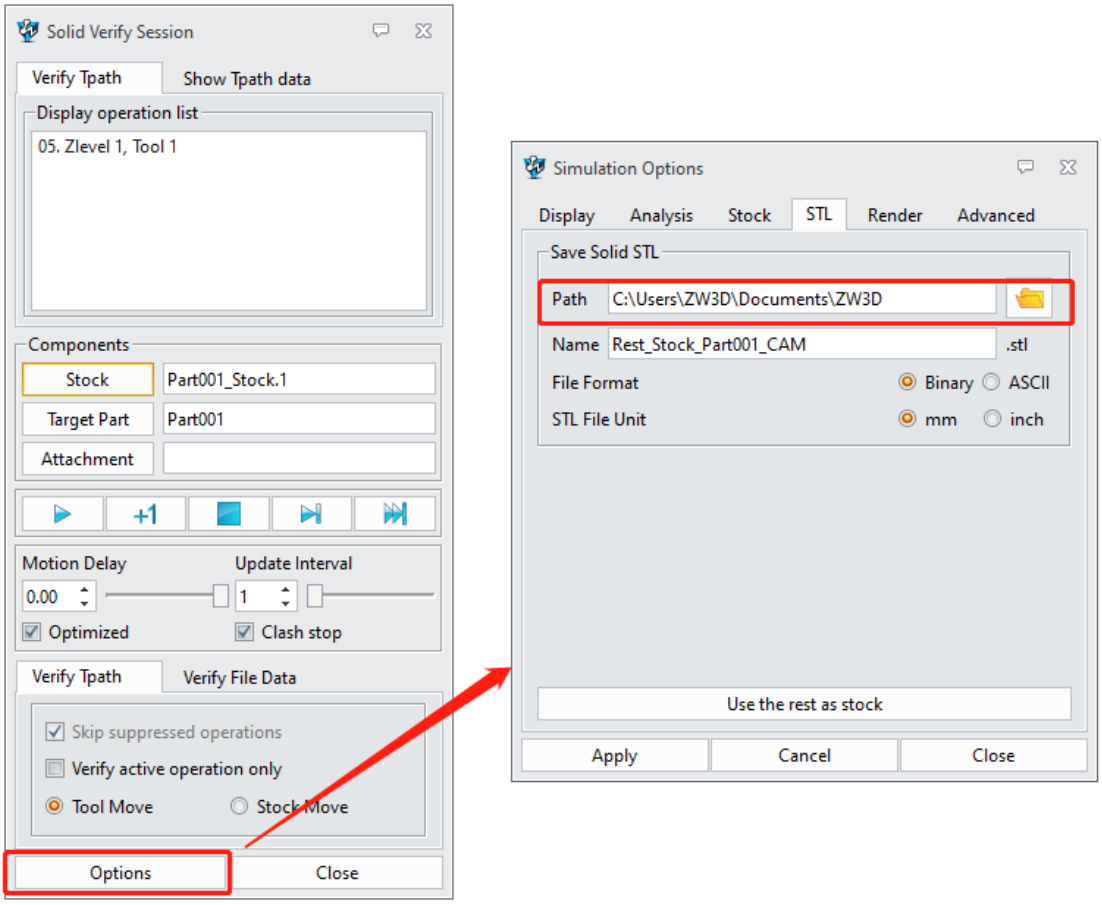


4.14.5 Support Open Operation List in WPS

In previous ZW3D, the export operation list can only be opened in Microsoft Excel. But in ZW3D 2023, the operation list table can be open in WPS.

4.14.6 STL Rest Stock Creation Optimization

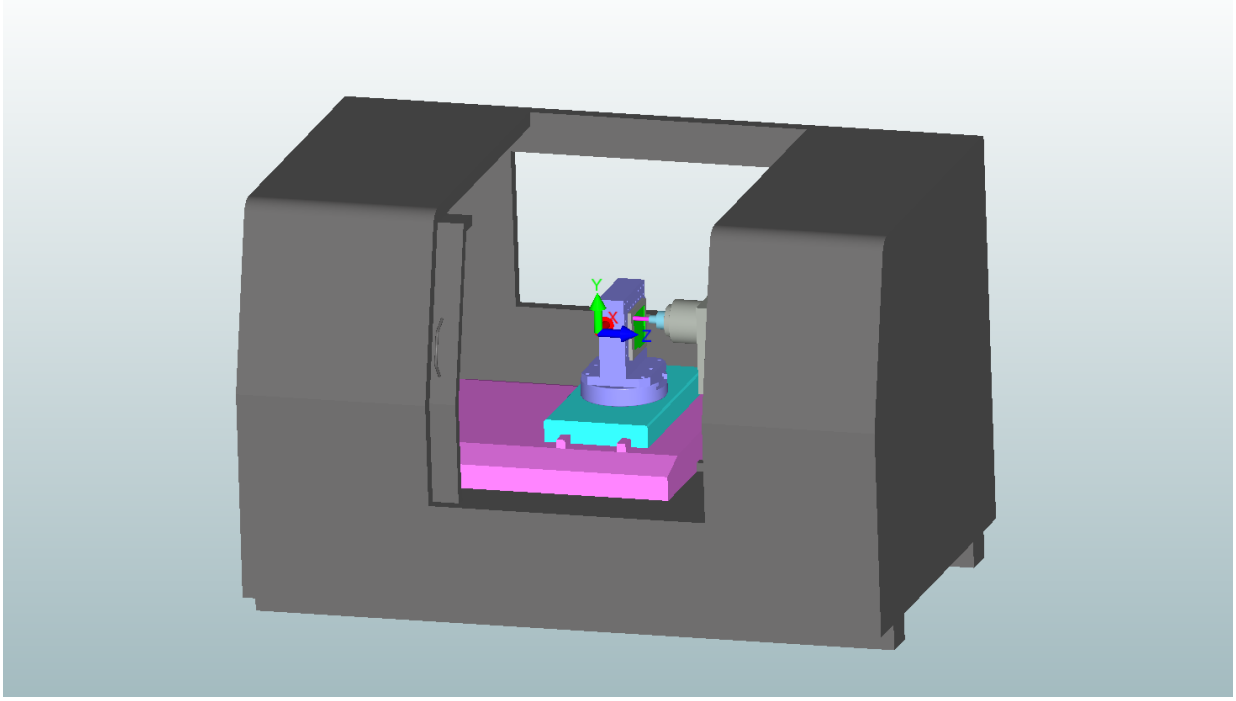
In ZW3D 2023, we added the functionality of customized STL solid path in Solid Verify, allowing user to self-define saving path of stock which is convenient for user to manage stock files.



4.15 FMS

4.15.1 New Horizontal Machine

We added a new 3-axis horizontal machine to FMS. See 3x_horizontal_machine.mch sample shown as below figure:



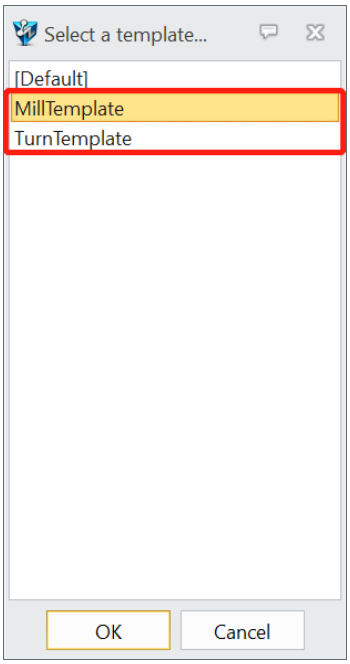
User can perform machine simulation by selecting different CNC control system and combing with postprocessing znc files according to the realistic needs. CNC controllers and corresponding postprocessing files are listed as below:

CNC Controller	Znc
ZW_Fanuc_Mill	ZW_FMS_5X.znc
ZW_Heidenhain_Mill	ZW_HEIDENHAIN_5X_FMS.znc
ZW_Sinumerik_Mill	ZW_SINUMERIK_5X.znc

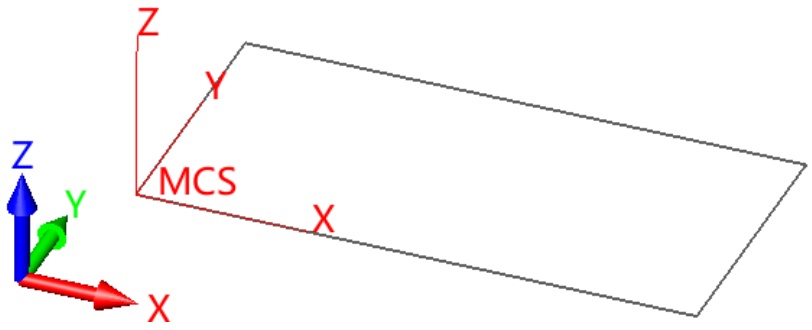
4.16Others

4.16.1New Milling and Turning CAM Plan Template

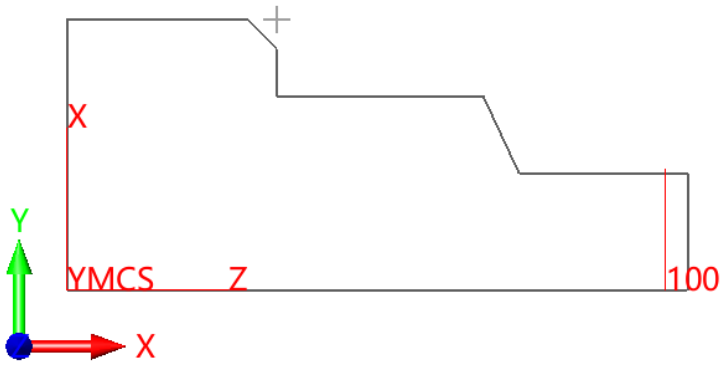
In ZW3D 2023, we added two CAM templates：Mill Template and Turn Template in the entry of CAM Plan.



Both Mill Template and Turn Template have an MCS (programming coordinate system) that are preconfigured and activated. MCS orientation in Mill Template are the same with the world coordinate system.

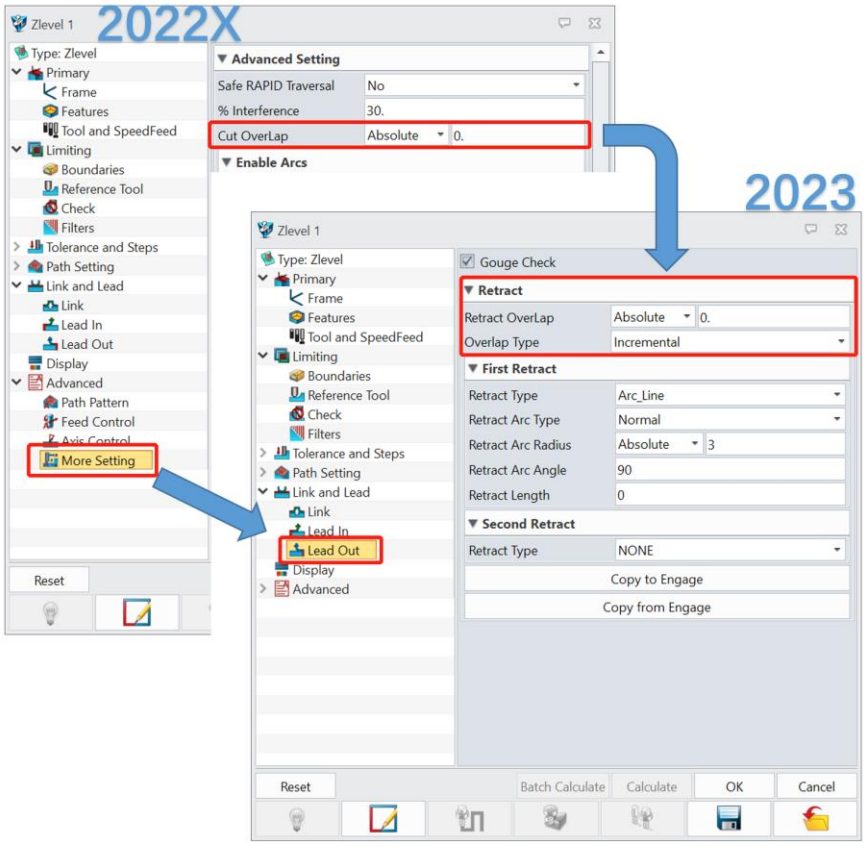


MCS orientation in Turn Template are not identical with the word coordinate system. Z axis of MCS is the same with X-axis orientation of the world coordinate system, and X-axis with Y-axis of the world coordinate system, which are convenient to user for turning programming.

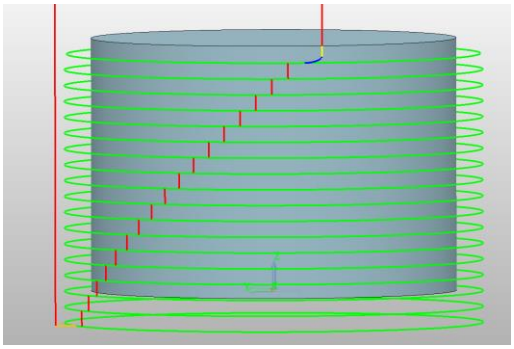
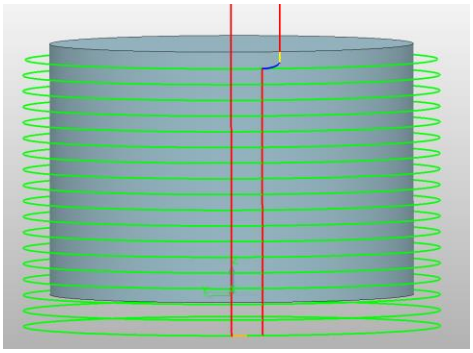


4.16.2New Overlap Style in ZLevel Operation

- We moved the function of Overlap from More Setting page to Lead Out page in Zlevel operation.

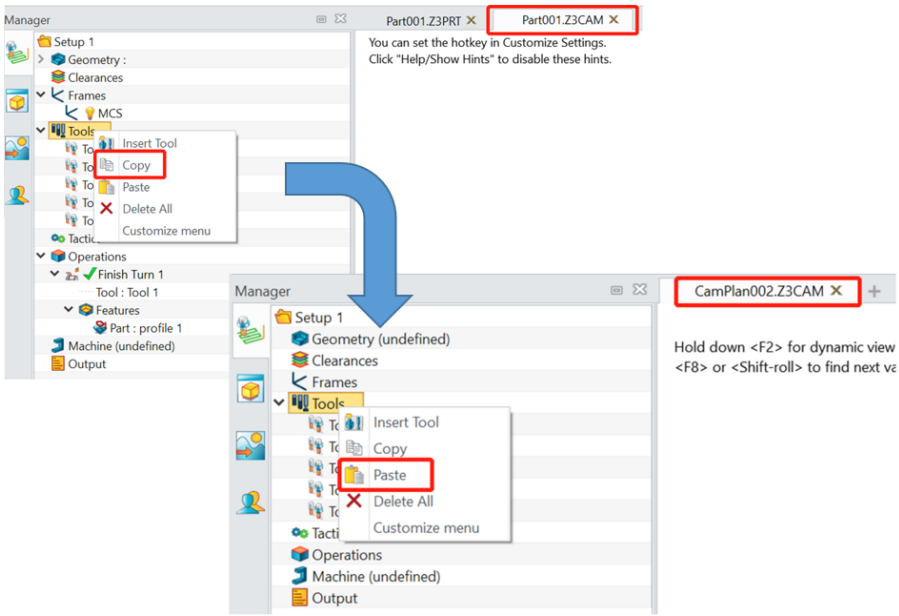


- Newly add stationary style in Overlap:

Incremental	Stationary
	

4.16.3New Tool Copy/Paste

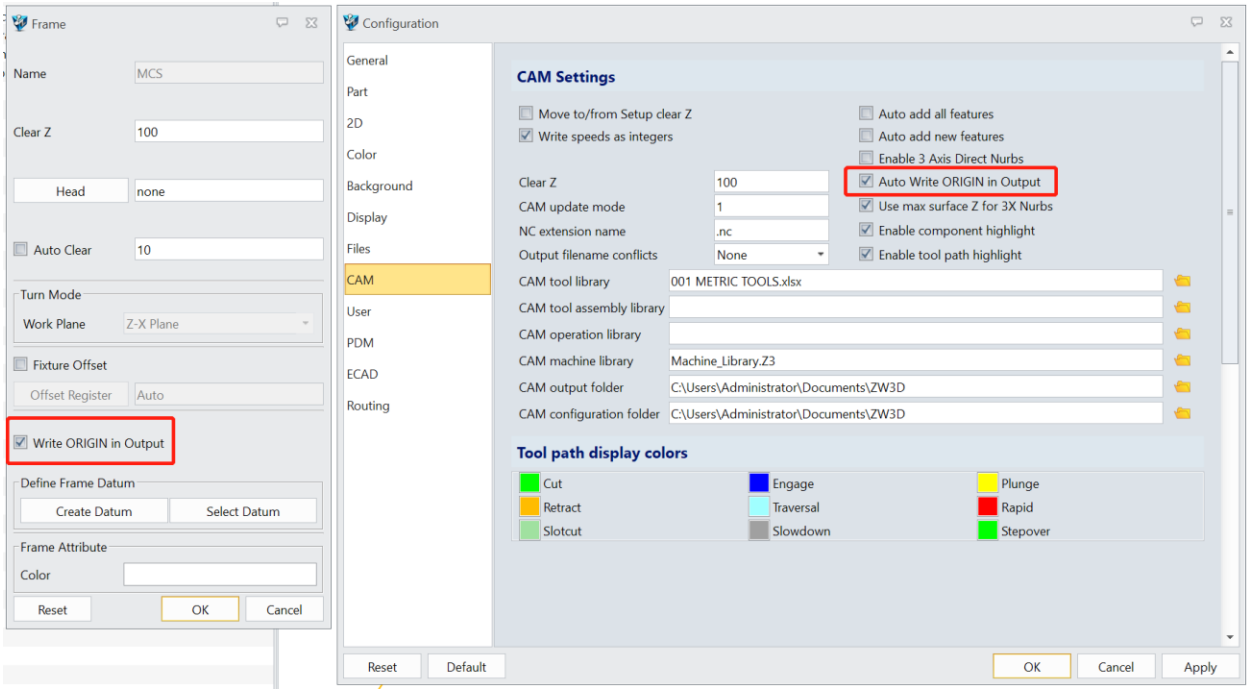
In the tool node of machining manager, we added the function of copy/paster to support copying/pasting a tool from a CAM plan to another CAM plan.



4.16.4Enable Auto Write ORIGIN in Output by Default

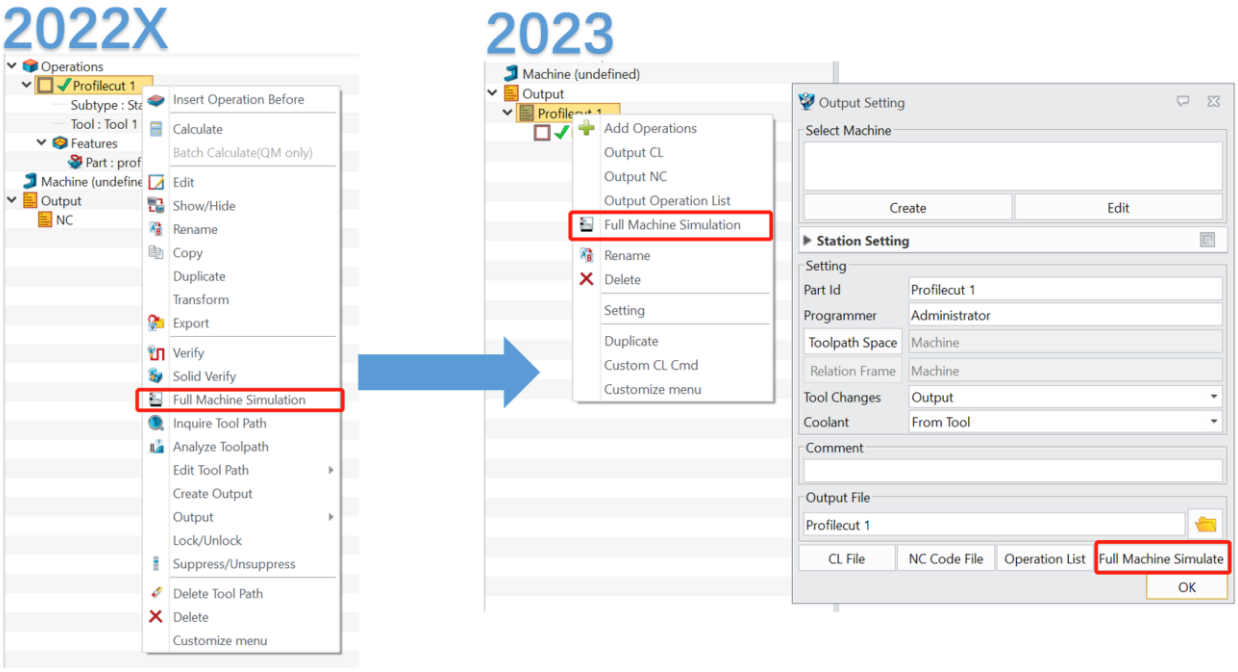
Output origin refers to output the reference coordinate position in the world coordinate system in the CL file, which is used for subsequent coordinate system conversation and suitable for 5-axis fixed orientation machining and turning combined scenarios. In ZW3D 2023, “Auto write ORIGIN in output” in

configuration is enabled by default and the option “Write ORIGIN in output” in the new frame is checked by default.



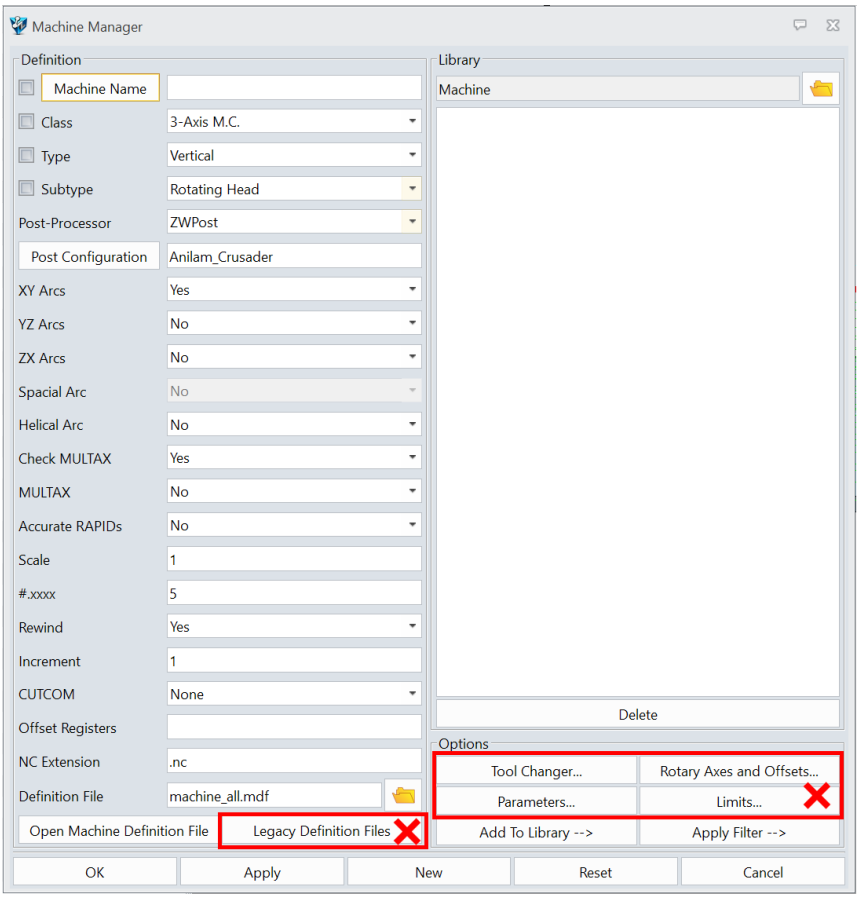
4.16.5Change FMS Start Position

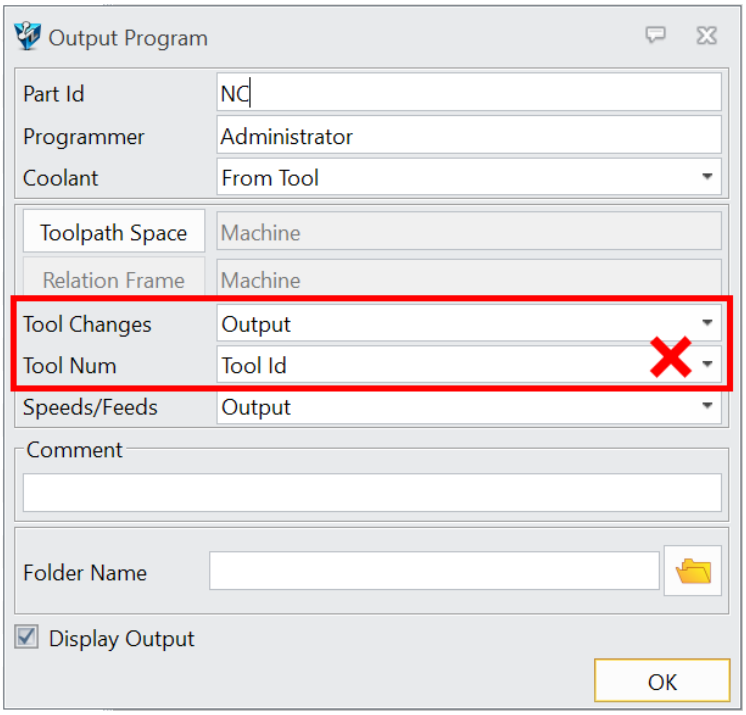
In previous version, FMS entrance located at the context menu of operation while in the new version, it was changed to the context menu of output node and at the lower right corner of output setting.



4.16.6Remove Some Functions from Machine Manager and Output Program Setting

In ZW3D 2023, we removed some functions from machine manger and output program as the below shown:


Machine Manager
Definition
Machine Name
Class
3-Axis M.C.
Type
Vertical
Subtype
Rotating Head
Post-Processor
ZWPost
Post Configuration
Anilam_Crusader
XY Arcs
Yes
YZ Arcs
No
ZX Arcs
No
Spacial Arc
No
Helical Arc
No
Check MULTAX
Yes
MULTAX
No
Accurate RAPIDs
No
Scale
1
#.xxxx
5
Rewind
Yes
Increment
1
CUTCOM
None
Offset Registers
NC Extension
.nc
Definition File
machine_all.mdf
Open Machine Definition File
Legacy Definition Files
Library
Machine
Delete
Options
Tool Changer...
Rotary Axes and Offsets...
Parameters...
Limits...
Add To Library -->
Apply Filter -->
OK
Apply
New
Reset
Cancel


Output Program
Part Id
NC
Programmer
Administrator
Coolant
From Tool
Toolpath Space
Machine
Relation Frame
Machine
Tool Changes
Output
Tool Num
Tool Id
Speeds/Feeds
Output
Comment
Folder Name
Display Output
OK