

ZW3D WHAT'S NEW

V 2024 SP



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ZW3D™ V2024 What's New

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Highlights of ZW3D 2024

Basic:	Enable New Render Engine by Default
	Display Element Information in Real Time
Translator:	Support PMI Data Exchange
	JT Format Support Exchanging PMI Data
Sketch & Wireframe Design:	Constraint Group
	Offset Constraint Improvement
	Spiral Helix Improvement
	Silhouette Curves Improvement
Shape Design:	"Pattern Feature" Efficiency Improvement
Sheet Metal Design:	New Miter Cut Functionality
	New Sketched Rip
Assembly Design:	New Spur Gear
	Assembly Partially Loading
	Support Part Design Behaviors in Assembly Environment
	Support In-place Editing Component Properties on Assembly Tree
	Improvement of Assembly Efficiency
	Top-down Design Based on Master Layout
	Link Manager Improvement

[Flexible Part](#)

Mold Design

[Support Parting Update after Product Model Changed](#)

[“MoldBase Modify” Command Optimization](#)

[New Trim Tool](#)

Drawing Sheet

[2D Drawing Sheet](#)

Design:

[New Complex Line Type](#)

[New Dimension Attaching Alignment](#)

[New Magnetic Line](#)

Piping Design:

[Optimized Spool Drawing Coordinate Dimension](#)

[Default Enable Display Optimization](#)

[Connection Point Improvement](#)

[New Piping ISO Drawing](#)

[Spool Drawing Improvement](#)

Structure:

[Structure](#)

Harness Design:

[New Create Port](#)

[New Define Wire & Cable](#)

[New Harness Route](#)

[Path Setting in Leading Part](#)

[New Crate Protection](#)

CAM

[Independent Shortest Tool Calculation](#)

[New Limiting in Turning Operation](#)

[Tool Path Trim Supports Drill Operation](#)

[Apply to New Render Engine](#)

[QM Step Max Roughing](#)

[Whole Workflow Improvement of Lathe Tool](#)

[Pencil Finish Improvement](#)

[Corner Finish Improvement](#)

[New QM Cluster Computing Function](#)

Simulation

[New Fatigue Analysis, Random Vibration and Parameter Optimization](#)

[Large Displacement Analysis](#)

[Large Strain Analysis](#)

[New Thermal Effect Load](#)

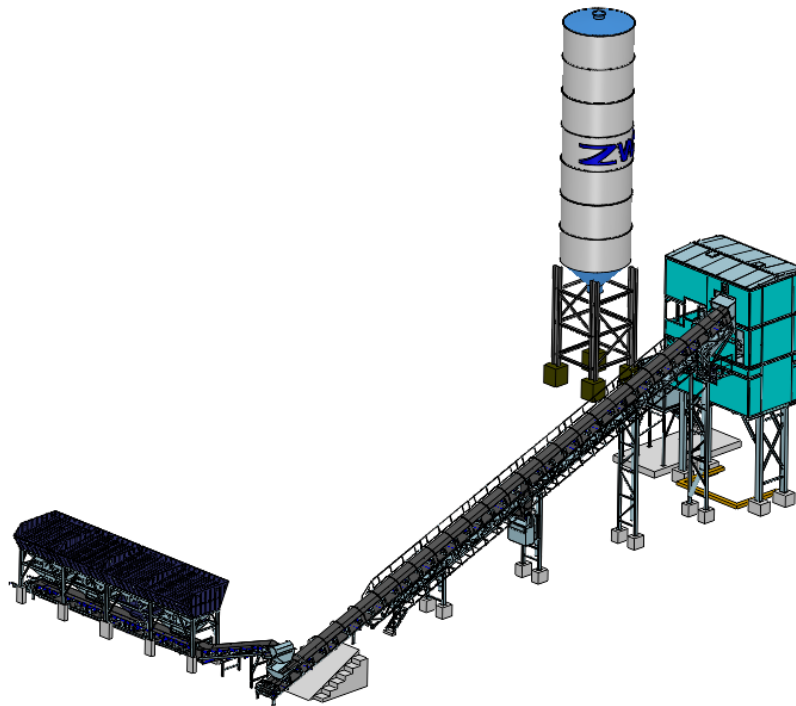
[New Small Slip Friction Contact](#)

[New Mesh Advanced Features](#)

1 Basic

1.1 ★Enable New Render Engine by Default

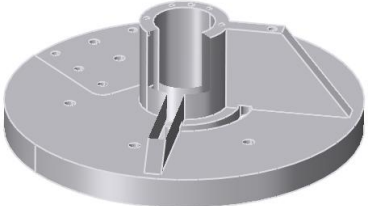
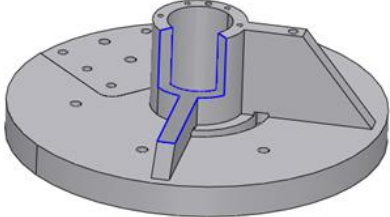
The new render engine is enabled by default in the premise of unchanging the original display effect. The display efficiency is significantly improved, especially in supporting higher frame rate while user is viewing and inquiring drawings and modelling multiple-shapes and big assembly files.



1.2 Section Display Improvements

1.2.1 Lightweight Model Section Display Improvement

The section display effect of lightweight model section has been optimized. The section display is consistent with the ordinary parts, which makes it more convenient to observe the inside structure of lightweight model.

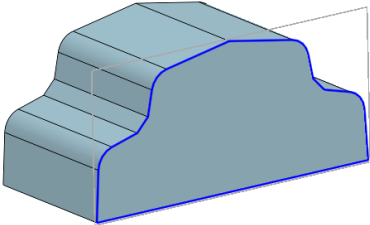
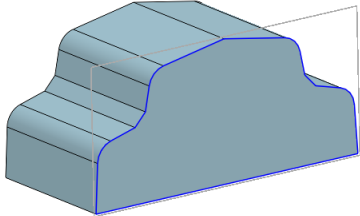
Effect in ZW3D 2023	Effect in ZW3D 2024
	

→ Where it is

Part/Assembly Environment >> Inquire >> Inspect Model >> Section

1.2.2 Default Line Width of Section Curves

To better observe the inside information of section, we changed the default line width value to 0.18mm.

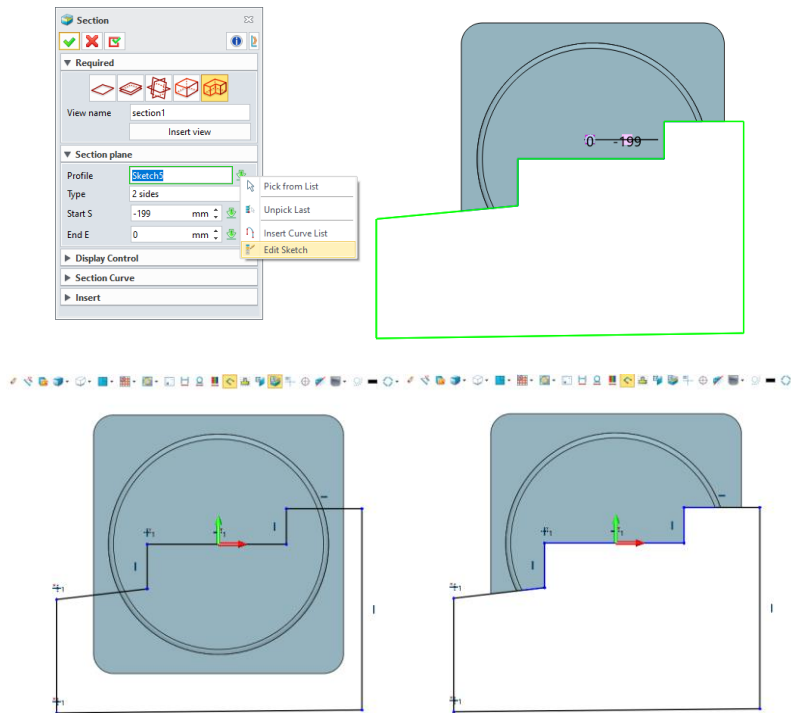
Display Effect in ZW3D 2023	Display Effect in ZW3D 2024
	

→ Where it is

Part/Assembly Environment >> Inquire >> Section >> Section Curve >> Line width

1.2.3 Support Turning On/Off Section View while editing sketch in Section

Users may need to refer to the non-sectioned model when they're editing the profile of section view. Therefore, ZW3D supports turning on/off section view when editing the sketch section profile.



→ Where it is

Part/Assembly Environment >> Inquire >> Inspect Model >> Section >> Section with profile >> Edit Sketch

1.3 Datum Enhancements

1.3.1 Display Datum Name

ZW3D supports displaying datum name in drawing area, including Default CSYS, Plane and Axis. User can locate to the needed datum plane more precisely.

Display name	Do not display name
<p>Default CSYS</p>	
<p>Plane1</p>	

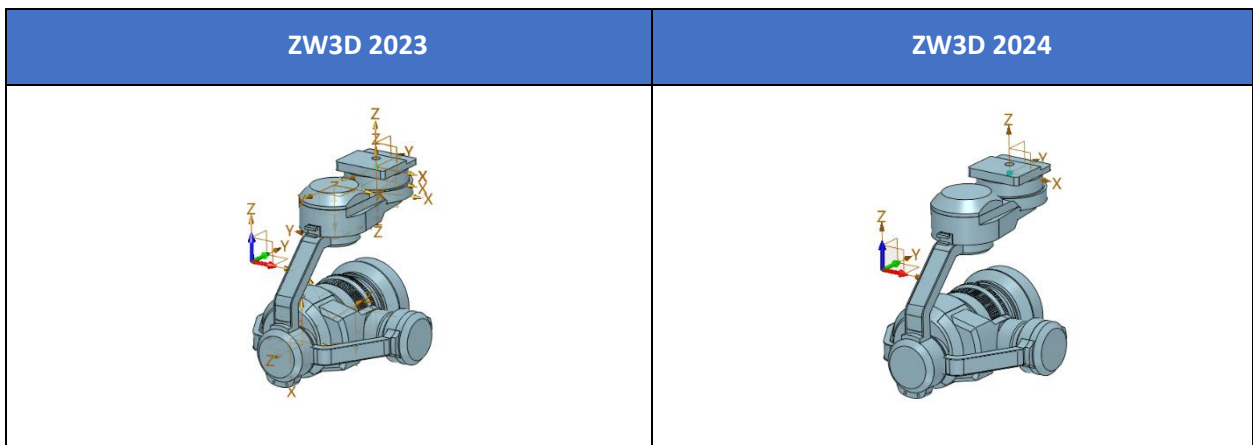


→ Where it is

Part/Assembly Environment >> Manager >> Visual manager >> Datum >> Name Display

1.3.2 “Show Datum” Improvement

ZW3D improved the display logics of datum in assembling parts, so that only the datum with top-level parts can be displayed and optimized the UI of assembling operation.



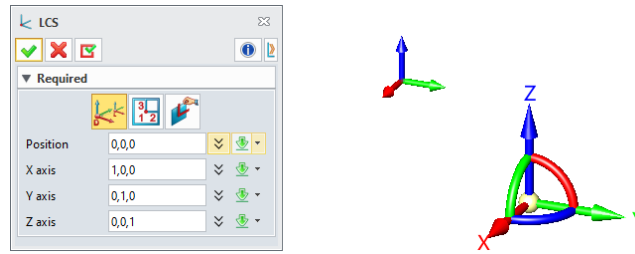
→ Where it is

Assembly Environment >> Insert >> Placement >> Show Datum

1.3.3 LCS Improvement

1.3.3.1 New “Dynamically create WCS”

When user changes LCS, the system will take the changed LCS as reference to locate the coordinate value. At this moment, it’s difficult for user to restore LCS to the origin of WCS. Therefore, ZW3D added “Dynamically create WCS” mode. Under this mode, the coordinate values are based on WCS, and user can locate to the origin of WCS by inputting coordinates 0,0,0.

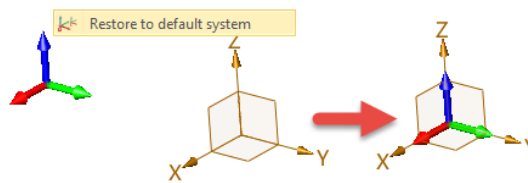


→ Where it is

Part Environment >> Datum >> LCS

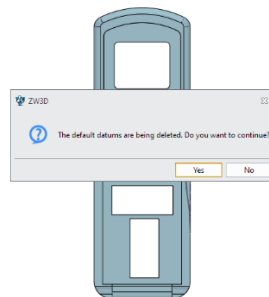
1.3.3.2 New “Restore to Default System” in LCS

Right-click on WCS and select “Restore to default system”, the WCS will coincide with the absolute coordinates system.



1.3.4 Delete Default Coordinate Hint Improvement

When deleting a default coordinate or a default datum plane from the history tree, a new pop-up prompt is added to enable users to confirm whether to delete the default coordinate or the default datum plane, so that avoid deleting the default coordinate by mistake.



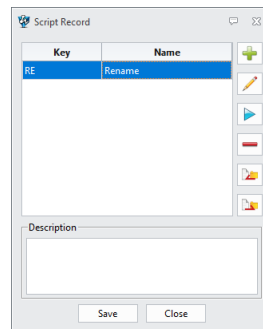
→ Where is it

Part/Assembly >> Manager >> History Manager

1.4 Script Record

1.4.1 Script Record Usability Improvement

We improved the usability of Script Record which supports picking objects first and then starting script record. Take renaming feature as an example: pick the feature to rename first, and then start the setting script record (for example set as RE then press RE directly) in which we can directly rename.

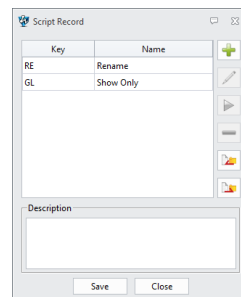


→ Where it is

Utilities >> Script Record

1.4.2 Script Record Supports Import & Export

User can import or export the setting script records for better management. For instance, User A can export the script records after setting a series of script records and distribute to other users. Thus, other users can use the setting script records simply by importing these script records.



→ Where it is

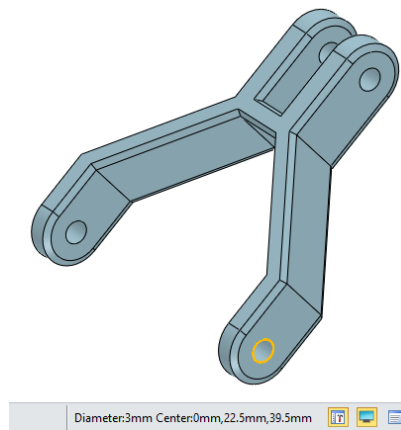
Utilities >> Script Record

1.5 Inquire

1.5.1 ★Display Element Information in Real Time

ZW3D newly added “Display element information in real time” . The function is disabled by default and user can enable it if necessary.

Once “Display element information in real time” is enabled, the physical attributes information of the geometry will be displayed directly in the lower right corner of model display area when you pick geometric object. Different information would be displayed according to different geometric types. For instance, when select a circle, it will display circular center coordinate and diameter; when select curve edge, it will display curve length; when select two or more geometric objects, it will display the current coordinate location of the geometric objects.



→ Where it is

[Configuration >> Part >> General >> Display element information in real time](#)

1.5.2 Promote Mass Inquire Efficiency

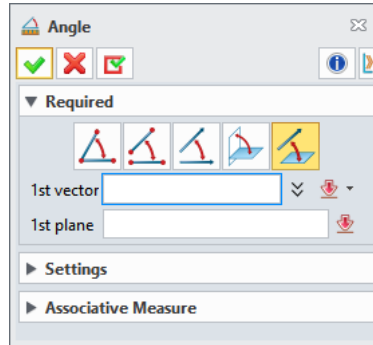
ZW3D 2024 optimizes mass properties mechanism. As for surface model with many control points, the inquire efficiency of mass properties has been improved over 10+ times.

→ Where is it

[Part/Assembly >> Inquire >> Mass Properties](#)

1.5.3 Inquire Angle

Add measurement methods of vector and plane in “Angle”.

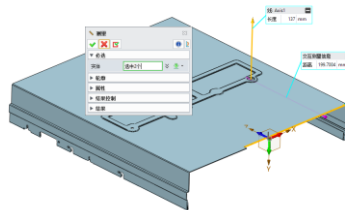


→ Where is it

Part/Assembly >> Inquire >> Angle

1.5.4 Measure

“Measure” function allows you to select an axis as a measure object.



→ Where is it

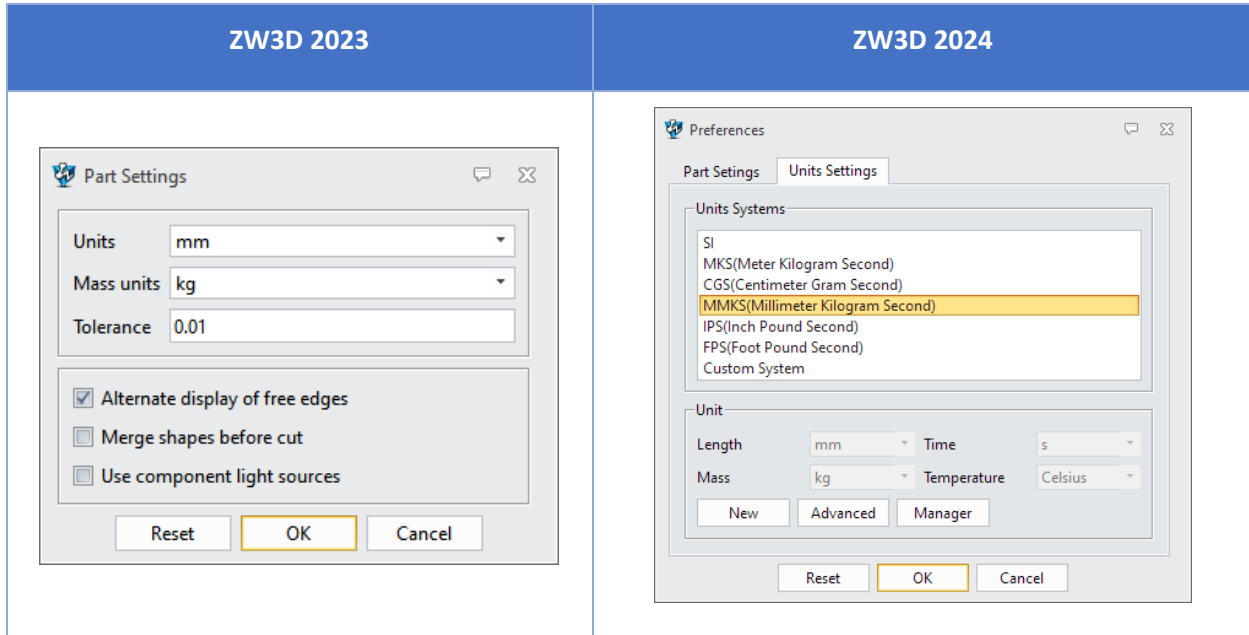
Part/Assembly >> Inquire >> Measure >> Measure

1.6 Unit Upgrade

1.6.1 New Unit Settings

New unit settings are used to unify unit management in ZW3D 2024. The unit settings support changing file unit system which contains multiple basic units and support exporting units.

Moreover, users can create and edit the self-defined unit.

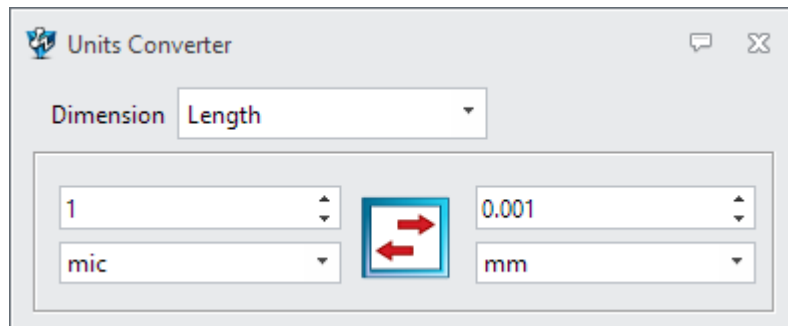


→ Where is it

Part/Assembly >> Tools >> Settings >> Preferences

1.6.2 New Unit Converter

New unit converter provides users a quick and direct tool to convert unit in ZW3D 2024.



→ Where is it

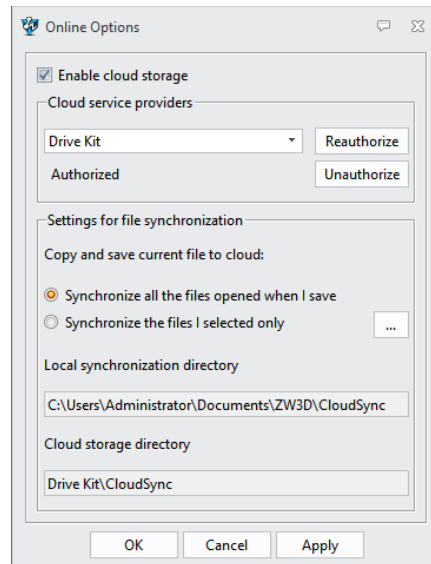
Part/Assembly >> Tools >> Utilities >> Unit Converter

1.7 Cloud Service

ZW3D 2024 supports cloud service and shortcut to link cloud disk where users can upload and download files. Only Drive Kit is supported to link in this version.

1.7.1 Online Function

Online function can be divided into two parts. The first one is to obtain authorization and the second is to set file synchronization.



→ Where is it

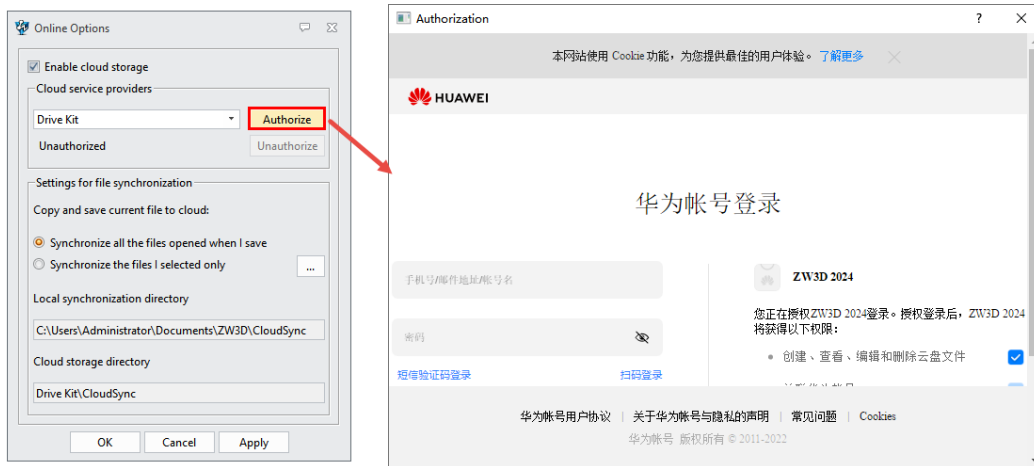
Home Page >> Menu Bar >> Cloud >> Online Options

1.7.1.1 Obtain Authorization

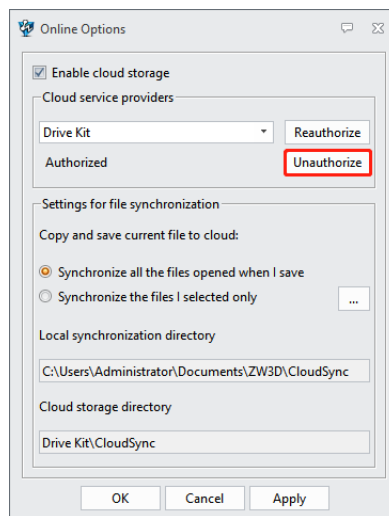
The authorization is to login the cloud disk account and obtain the read and write functionalities. If to enable cloud service, you need to check “Enable cloud storage” first, then you can operate the following account to get authorization. The option includes the following functionalities:

- Display whether you are authorized: display “Unauthorize” when you’re unauthorized; display “Authorized” when you’ re authorized.

- Click “Authorize” and jump to cloud account registration and login interface. Again click “Authorize” to re-authorize.



- Support to unauthorize after getting authorized.



→ Where is it

Home Page >> Menu Bar >> Cloud >> Online Options

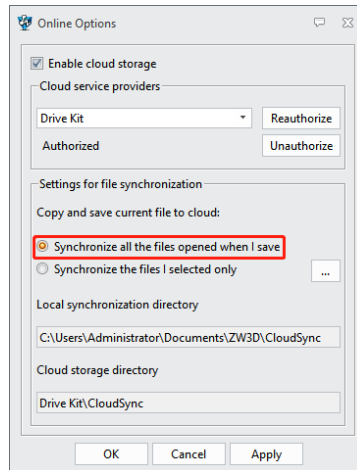
1.7.1.2 File Settings for Synchronization

There are two file synchronization settings including “Synchronize all the files opened when I save” and “Synchronize the files I selected only”. “Synchronize all the files opened when I save” is set by default.

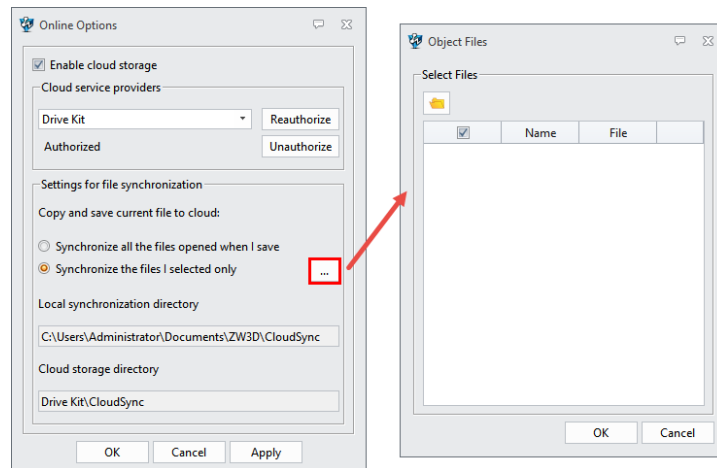
Display the local save path and the cloud disk save path below the two options and you are not allowed to modify the two paths.

The status of the two synchronization methods is as follow:

- Synchronize all the files opened when I save: it can upload the current files to the cloud disk automatically when user clicks “Save” button.



- Synchronize the files selected only: click “...” button beside and you can select files to upload to the cloud disk in the objective file popup.

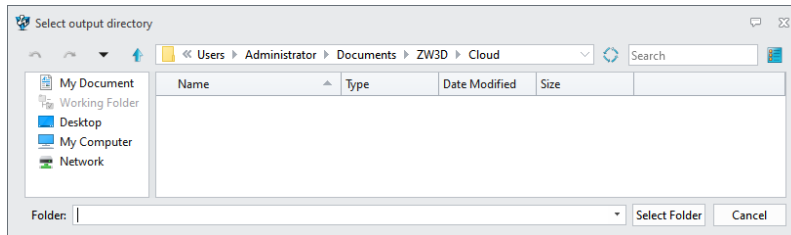


→ Where is it

Home Page >> Menu Bar >> Cloud >> Online Options

1.7.2 Save to Cloud

Cloud service provides the function of Save to Cloud which can save the current files to the objective folder in the cloud disk by manual and allow to create and delete the folder in the cloud disk when saving files.



→ Where is it

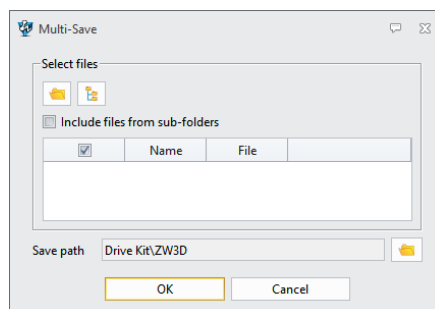
[Home Page](#) >> [Menu Bar](#) >> [Cloud](#) >> [Save to Cloud](#)

1.7.3 Upload Multiple Files by Multi-Save

Cloud service provides the function to upload multiple files which supports uploading multiple files to the cloud disk and allow you to select the specific location for the target disk.

There are two ways to upload multiple files: one is to select multiple files, and another is to select multiple folders. When multiple folders are selected, it will automatically recognize all internal files and list in the below.

- You can delete the files in the list.
- You can select which files to be uploaded. If checked it, upload files; unchecked it, do not upload files for now.



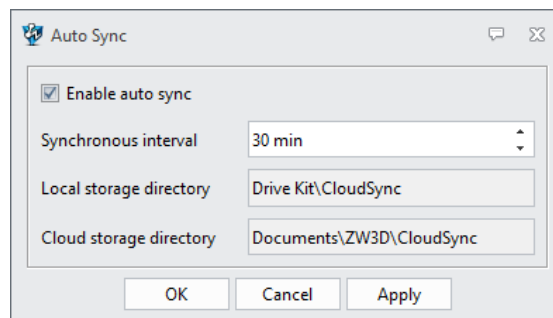
→ [Where is it](#)

[Home Page](#) >> [Menu Bar](#) >> [Cloud](#) >> [Multi-Save](#)

1.7.4 Auto Sync

Cloud service provides the function of Auto Sync (automatic synchronization). If you want to use, you need to enable auto sync first. You can upload and download all files according to the specified interval time.

- Default 30 minutes as sync time which can be adjusted.
- Set target cloud's storage directory and local storage directory as default state which cannot be adjusted.



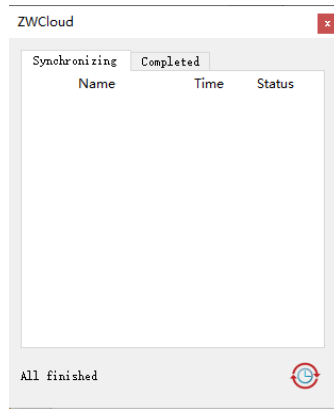
→ [Where is it](#)

[Home Page](#) >> [Menu Bar](#) >> [Cloud](#) >> [Auto Sync](#)

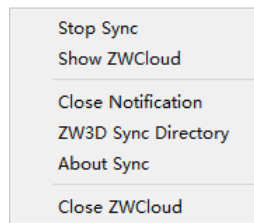
1.7.5 ZWCloud

Cloud service provides the function of ZWCloud which can be activated only after enabling cloud storage and obtaining authorization. Click the icon where you can check the file synchronization state in the popup dialog including:

- the synchronizing file's name, uploaded time, and status.
- allow to suspend or continue to synchronize.



- You can right click the shortcut menu which includes operations of Stop Sync, Show ZWCloud, Close/Open Notification, ZW3D Sync Directory, About Sync, and Close ZWCloud.



→ Where is it

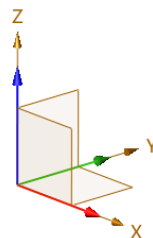
Home Page >> Menu Bar >> Cloud >> ZWCloud (on the lower right taskbar)

1.8 UI Improvements

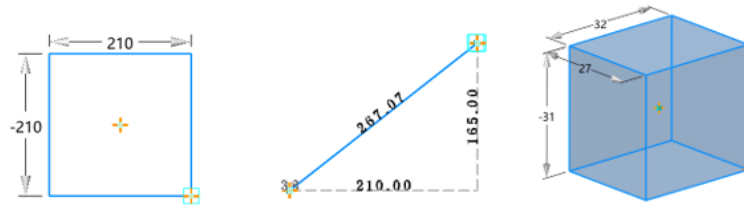
1.8.1 Display Element Improvement

A few selected UI elements have been optimized their display effect to offer a better user experience.

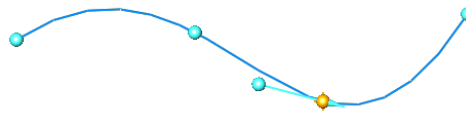
- Default set 2 times of coordinate axis datum plane, and slimmer coordinate axis.



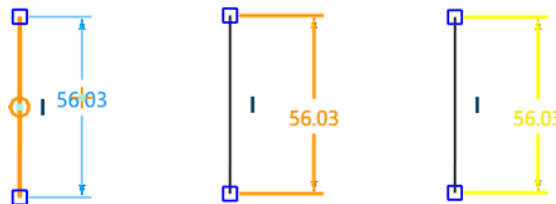
- Adjust preview color to blue and grey arrow during the creating process. When the graphic creation is too small, set a number to point to line. The angle during the creating process is not a current major value. Change the color into grey to indicate the primary and secondary relationships of numeric information.



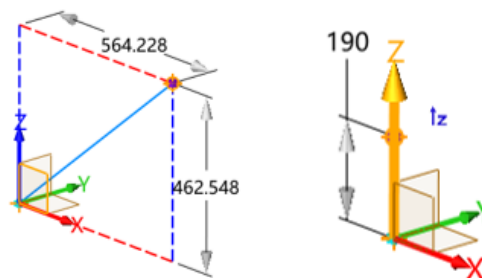
- Set new color for spline's control points. Take blue as the default color. The highlight state is the same as the system in orange.



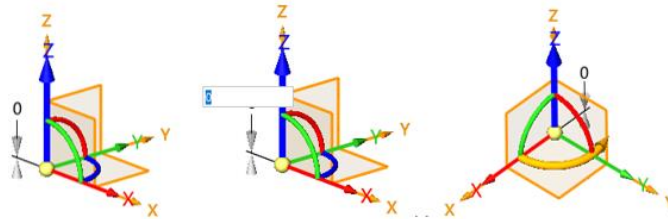
- Change the dimension click process to blue and adjust spending status preview highlight to yellow to make it clear.



- Adjust arrow to grey arrow style with a weak color, implying it is always changeable. The original two lines is changed into bounding dashed box to highlight the effect on the same surface. The activated axis status is the same as the system, which is changed to orange and click to zoom in.

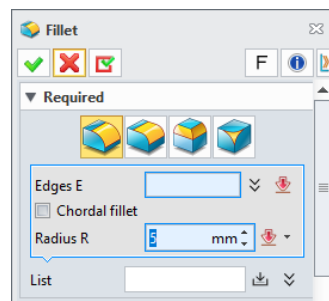


- Specify the number and user can directly click to change the number for rotating axis Z. The arrow still exists after rotating, which implies the last dragged axis.

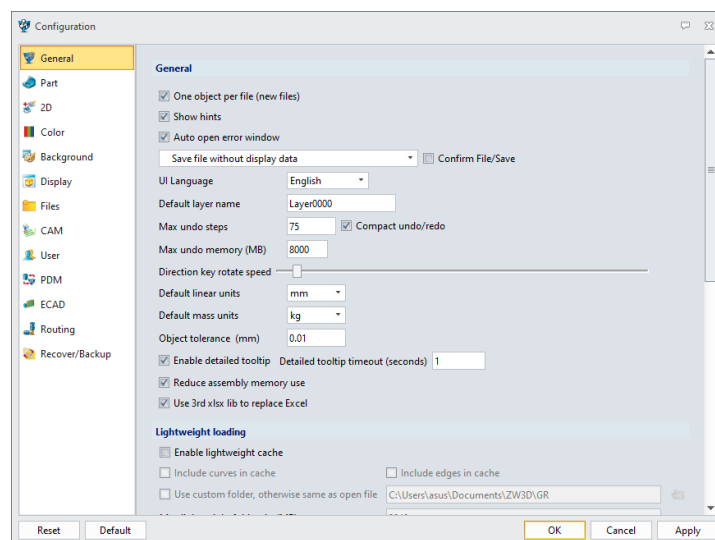


1.8.2 Command Panel

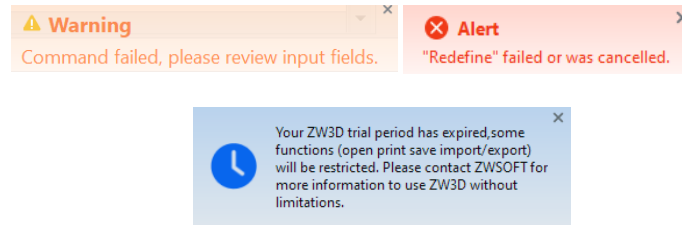
- The control click is changed to blue in command panel.



- Remove extra borders in configuration popup and add icons at the left side and unify the size and space between the control.



3. We upgrade the hint pop-up window by enhancing the bubble floating style.



1.8.3 Optimize Command Parameter Input Control

Added “Mouse wheel does not change the value” to the customize mouse. If the option is checked, command activated parameter values will not be modified while rolling mouse wheel in shape design process.

→ [Where it is](#)

[Tools >> Customize >> Mouse >> Mouse Actions >> Mouse wheel does not change the value](#)

1.9 File Management

1.9.1 File Path Extension

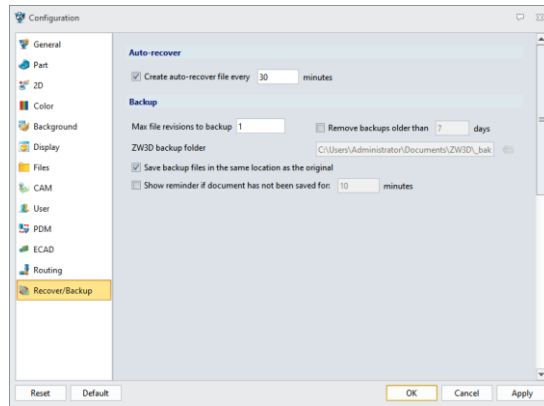
ZW3D 2024 extends the limits on the file path length by supporting 259 maximum characters, so that user can create/open/save/import/export files under long path.

→ [Where is it](#)

[File >> New/Open/Save/Import/Export](#)

1.9.2 New Recover/Backup in Configuration

ZW3D 2024 unifies the configuration of file recover and backup in one page and removes the option “Auto backup per file per day after save”.



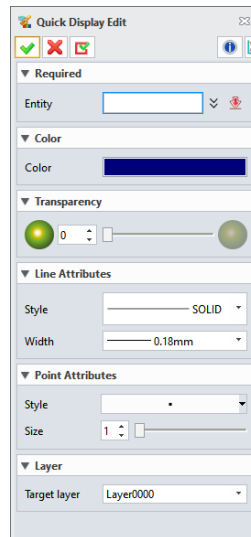
→ Where is it

Configuration >> Recover/Backup

1.10 Misc.

1.10.1 New Quick Display Edit

The new function Quick Display Edit supports in the mode of shape operating the selected object's color, transparency, line attributes, point attributes, and target layer.

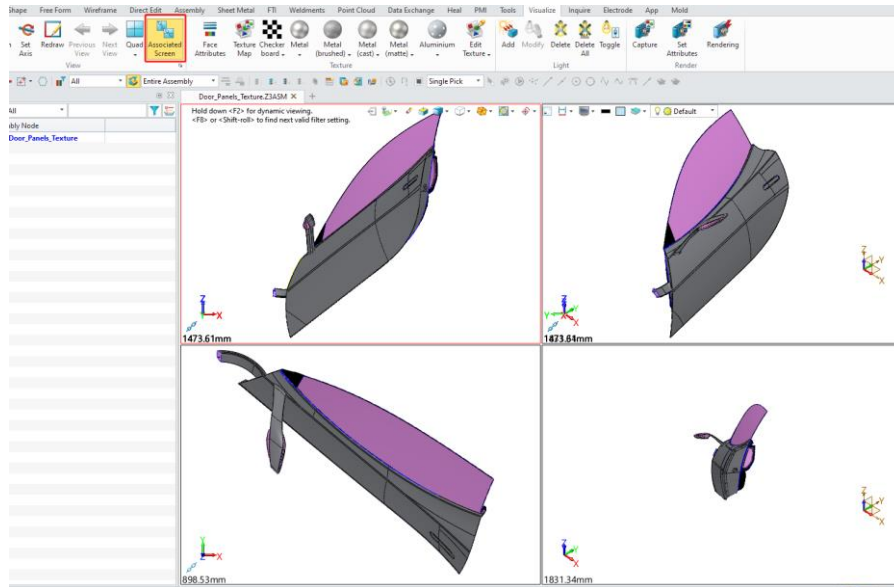


→ Where is it

Part >> Tools >> Attributes >> Quick Display Edit

1.10.2 New Associated Screen

When the function of associated screen is activated, for the standard view, the rotation function is forbidden, and the associated screen scale function is synchronized; for the drag function, the left and right views are dragged association up and down, the up and down views are dragged association left and right.

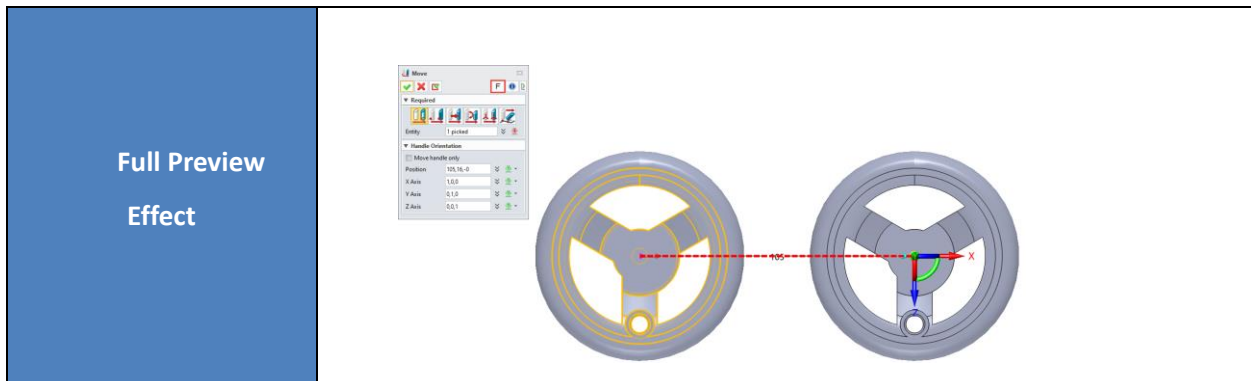


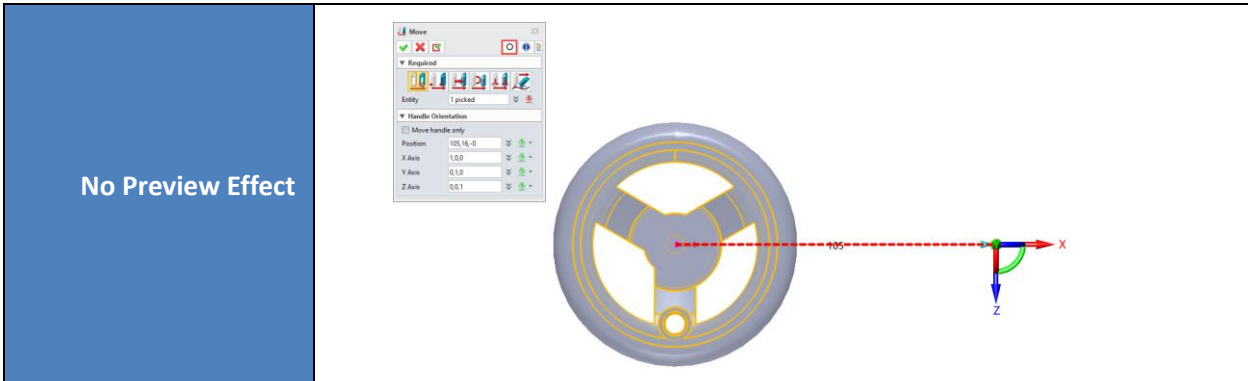
→ Where is it

Visualize >> View >> Associated Screen

1.10.3 New Preview switch in Move/Copy

ZW3D added preview switch in “Move/Copy” command, which is placed in the red box as the figure shown below. “F” represents full preview and “O” represents no preview.





→ Where it is

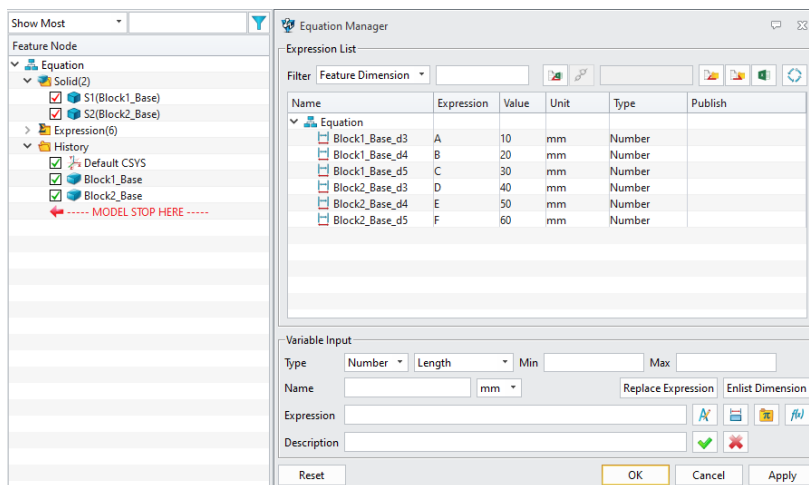
Part Environment >> Basic Shape >> Move/Copy

Assembly Environment >> Mold >> Library >> Move/Copy

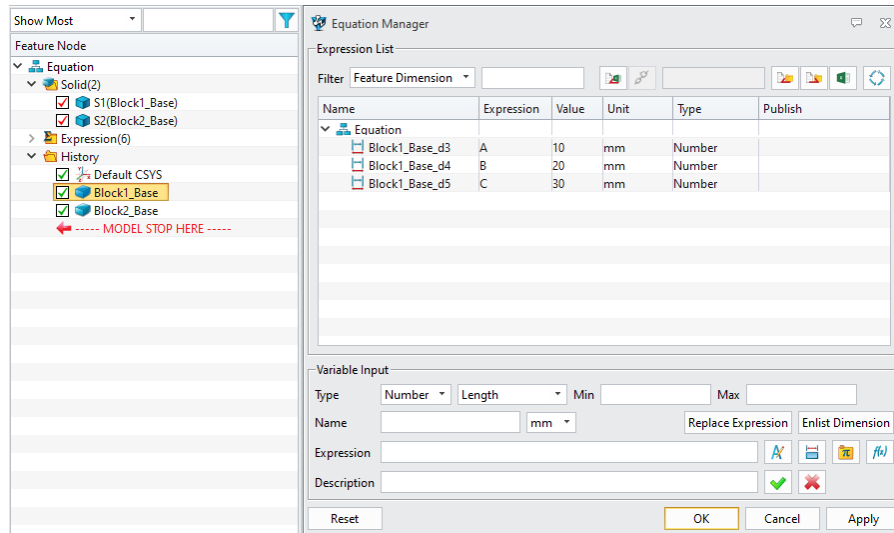
1.10.4 Manager Expression Filter Improvement

ZW3D optimized the filter function to easier identify the corresponding expressions of the features. After opening the equation manger and switching filter to Feature Dimension, the equation manger will filter other expressions after selecting the history features and only display the expressions that are referenced by the selected features.

Before selecting features:



After selecting features:



→ Where it is

Part Environment >> Tools >> Equation Manager

2 Translator

2.1 Support Switch Translator

The following table lists out the input formats supported in ZW3D 2024, updated formats are marked in red.

Import Product	File Extension Name	Default Translator	Alternative Translator
Catia V4	.model, .exp, .session	4.1.9 - 4.2.4	4.1.9 - 4.2.4
Catia V5	.CATPart, .CATProduct, .CGR	V5R10 - V5-6R2023	V5R8 - V5-6R2022
Catia_2D	.CATDrawing	V5R10 - V5-6R2023	V5R8 - V5-6R2022
3DEXperience (CATIA V6)	.CATPart, .CATProduct, .cgr	R2010x - R2023x Output	From V6 R2022x and previous versions output
NX(UG)	.prt	11 - NX 2212	11 - NX 2212
Creo(Pro/E)	.prt, .prt.*, .asm, .asm. *	16 - Creo 9.0	16 - Creo 9.0
Creo(Pro/E)_2D	.drw .drw.*	2000i - Creo 9.0	Do not support
SolidWorks	.sldprt, .sldasm	98 - 2023(Only 64-bit supported)	98 - 2023(Only 64-bit supported)
SolidWorks_2D	.slddrw	2004 - 2022(Only 64-bit supported)	2013 - 2023(Only 64-bit supported)
SolidEdge	.par, .asm, .psm	V10 - SE2022	V18 - SE2023
Inventor	.ipt, .iam	.ipt (V9 - V2023)	.ipt (V6 - V2023)

		.iam (V9 - V2023)	.iam (V11 - V2023)
ACIS	.sat, .sab, .asat, .asab	R1 - 2022 1.0	R1 - 2022 1.0
DWG	.dwg	R11 -	R11 -
DXF	.dxf	R11 -	R11 -
IGES	.ige, .iges	/	/
STEP	.stp, .step, .stpz	203, 214, 242	203, 214, 242
Parasolid	.x_t, .x_b, .xmt_txt, .xmt_bin	Up to 30.0	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	4.0 - 4.3
JT	.jt	6.4-10.4	6.4-10.4
OBJ	.obj	/	/
Rhino	.3dm	version 2 - 7	version 2 - 7

Do not support export Catia related formats by default.

Export Product	File Extension Name	Default Translator	Alternative Translator
Catia V4	.model	Do not support	4.1.9 - 4.2.4
Catia V5	.CATPart, .CATProduct	Do not support	V5R15---V5/V6R2018
3DXML	.3dxml	Do not support	4.3
CGR	.cgr	Do not support	V5R15---V5/V6R2018

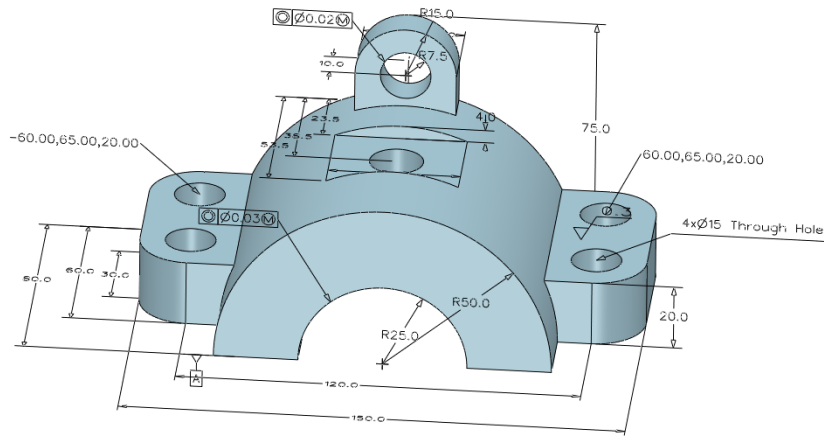
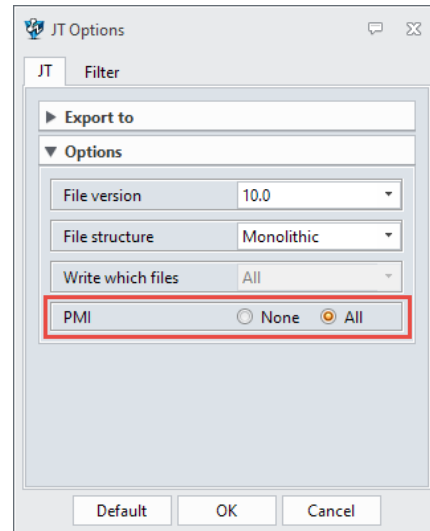
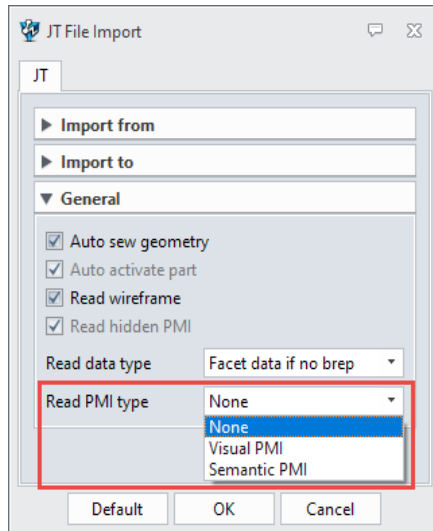
→ [Where is it](#)

[Configuration >> General >> Use default translator](#)

2.2 ★JT Format Support Exchanging PMI Data

Engineers can quickly understand the product manufacturing information such as model dimension and tolerance with the help of PMI. ZW3D have achieved PMI data in JT format, which can keep the product manufacturing information such as dimension, tolerance, and roughness in the model, providing engineers a clear product design purpose.

We added PMI option to JT format import and export setting. The JT file import supports reading PMI type including None, Visual PMI, and Semantic PMI. The JT export options supports None and All.



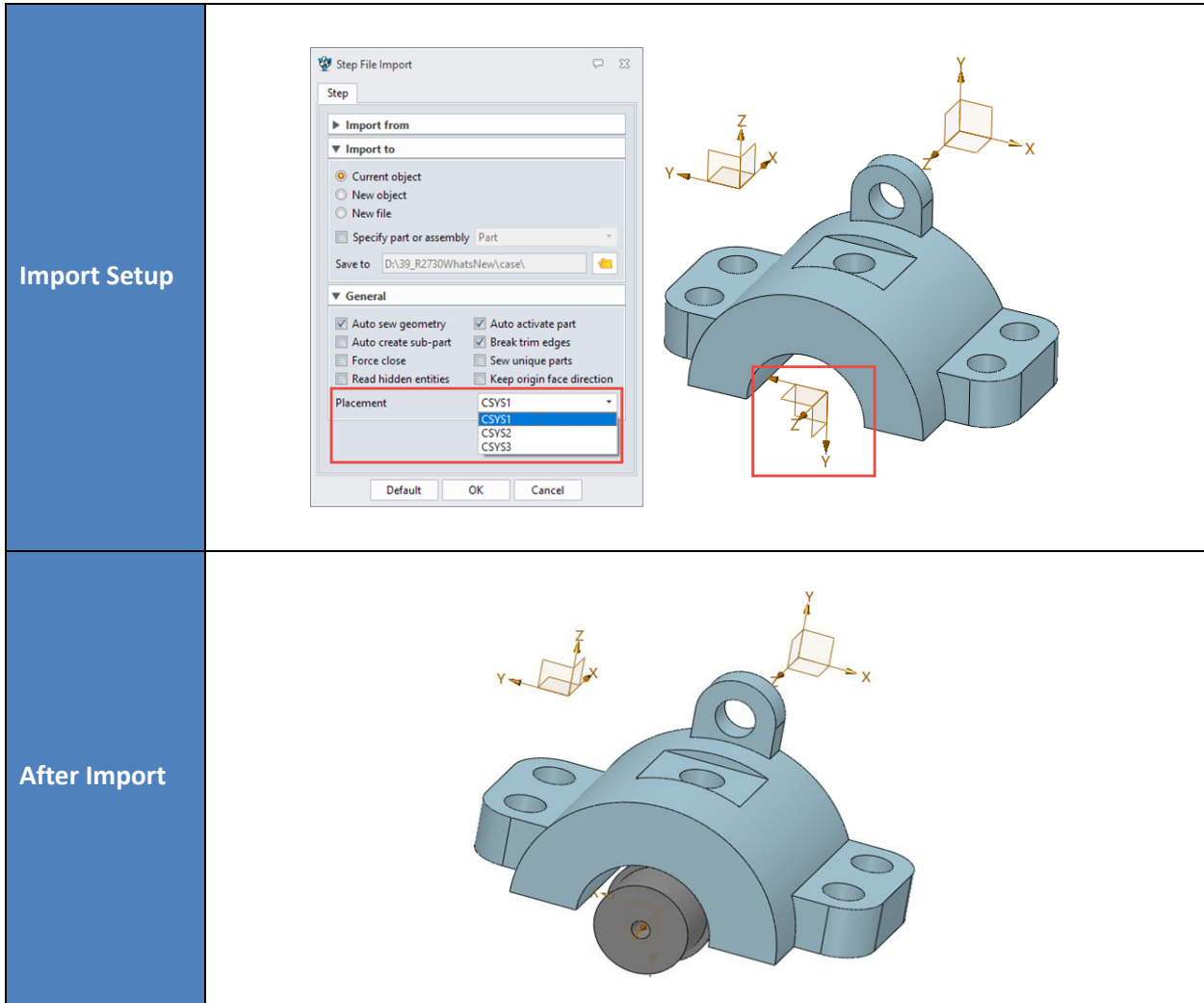
→ Where it is

File >> Import >> Select JT File >> JT File Import

File >> Export >> Select JT File >> JT Options

2.3 File Import Supports Selecting CSYS Placement

New “placement” option in step file import enable user to choose a specific CSYS to place the step file to the current object.



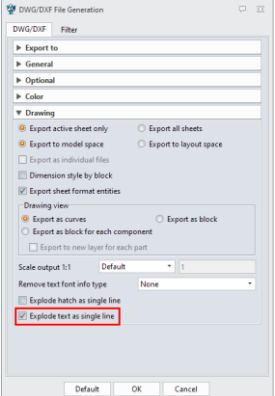

→ Where it is

File >> Import >> Select Step File Import >> Placement

2.4 DWG Export Supports Explode Text into Single Line

To avoid the exporting DWG file changed due to missing some characters, ZW3D supports exploding the text into single lines in DWG file.

You can check the “Explode text as single line” option in DWG export setting.

Export Setup	Text in Z3DRW	Text in Export DWG
	<p>Text to line</p>	

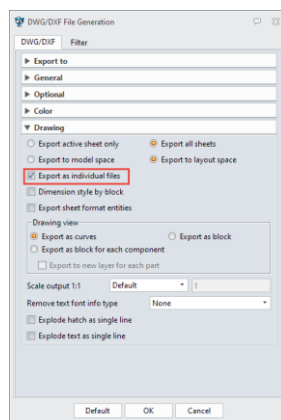
→ Where it is

File >> Export >> DWG/DXF File >> DWG/DXF File Generation

2.5 Drawing Sheet Supports Exporting Multiple DWG Files Individually

A Z3DRW file can have many drawing sheets. We added “Export as individual files” to allow user to export multiple drawing sheets into individual DWG files separately.

When “Export all sheets” is checked in DWG File Generation, you can check the option “Export as individual files”.

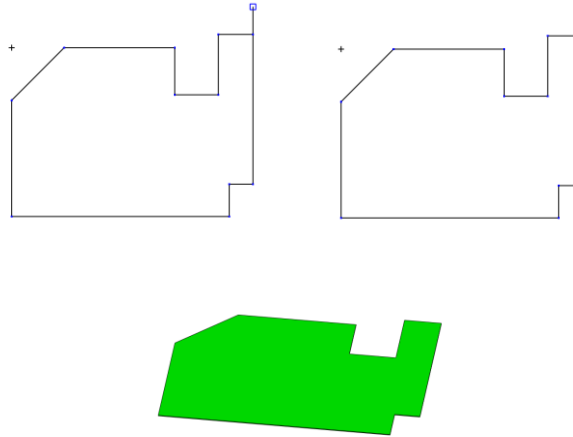


→ Where it is

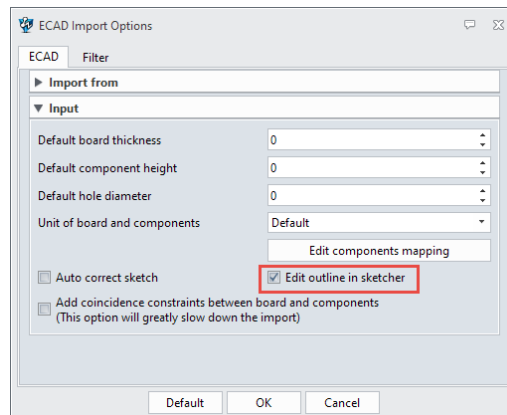
File >> Export >> DWG/DXF File >> DWG/DXF File Generation

2.6 New “Edit outline in sketcher” in ECAD Input Option Dialog

When users input an ECAD file, it will automatically detect whether the profile is a closed ring. If it isn't a closed ring, it will enter the sketch environment where allows user to directly edit profile. As shown in the figure below, importing the outline on the left will automatically enter the sketch environment. Refer to the outline on the right to edit the profile, and then import it normally.



In addition, we added “Edit outline in sketcher” in ECAD Import Options. If checked, it will automatically enter the sketch environment.



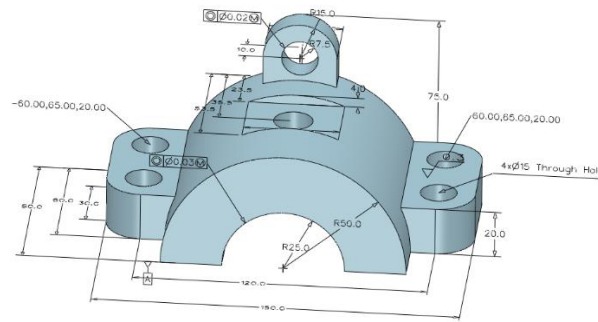
→ Where it is

File >> Open >> Select EMN File >> Open

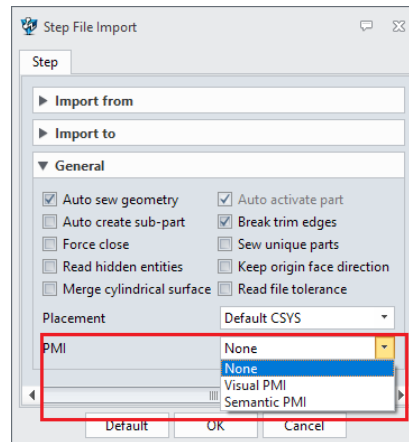
2.7 STEP Translation Enhancement

2.7.1 ★Support PMI Data Exchange

Engineers can quickly understand the product manufacturing information such as model dimension and tolerance with the help of PMI. In ZW3D 2024, we have achieved PMI data in STEP format, which can keep the product manufacturing information such as dimension, tolerance, baseline, and roughness in the model, providing engineers a clear product design purpose.



We improve PMI option to STEP format import and export setting. The STEP file import supports reading PMI type including None, Visual PMI, and Semantic PMI. The export option supports None and All.



→ Where is it

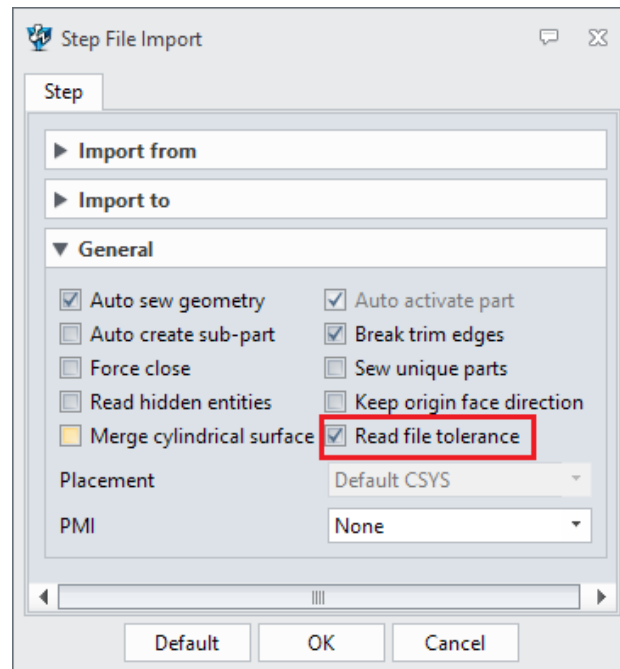
File >> Import >> Select Step File >> Step File Import

File >> Export >> Select Step File >> Step Options

2.7.2 Import Support Reading File Tolerance

When there is a huge difference between STEP file tolerance and system's modelling, import STEP file with the source of tolerance can improve the modelling quality.

If check "Read file tolerance" option in the STEP file import setting, ZW3D 2024 can use the source file tolerance to import model, which will not change the system's modelling tolerance.



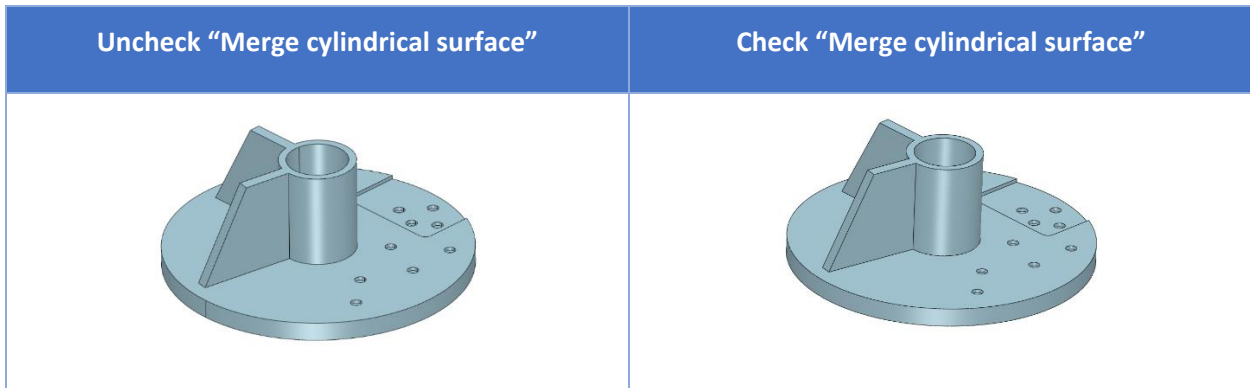
→ Where is it

File >> Import >> Select Step File >> Step File Import

2.7.3 Import Support Merge Cylindrical Surface

The cylinder in STEP files may be separated into two surfaces at the sewing place. Merging split cylindrical surface can reduce file size, which can be helpful in the follow-up process modelling.

Check "Merge cylindrical surface" in the STEP file setting which can merge the two split surfaces into the same cylindrical surface.



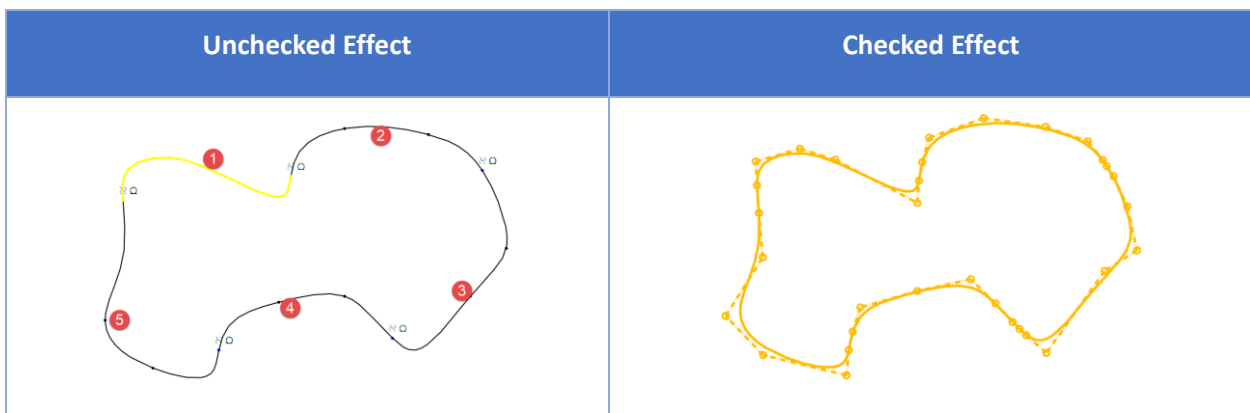
→ Where is it

File >> Import >> Select Step File >> Step File Import

2.8 New "Convert G2 continuous curves to one curve" in DWG/DXF Import

Add a new option "Convert G2 continuous curves to one curve" to DXF DWG import. After this option is checked, G2 or above continuous curves will be merged into one curve when importing.

The following figure shows a shape that formed by 5 G2 continuous curves. The left shows the effect of unchecked option, which keeps 5 curves. The right side shows the effect of checked option, which merges 5 curves into one curve.



→ Where is it

File >> Import >> DXF DWG Import Options

3 CAD

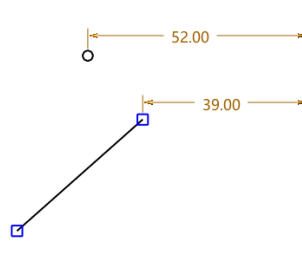
3.1 Sketch Design

3.1.1 Symmetry Dimension Improvement

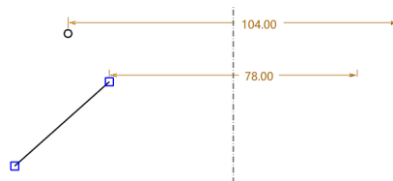
For the revolved part sketch, we added an annotation which can automatically switch radial to diameter to meet different dimensional needs. We also added the function in symmetrical sketch.

1. Use Quick Dimension command to pick axis objects, structure lines, reference lines and a line or a point for dimensioning. You can switch in diameter or radius dimension by dragging the mouse over both sides of axis, structure lines or reference.

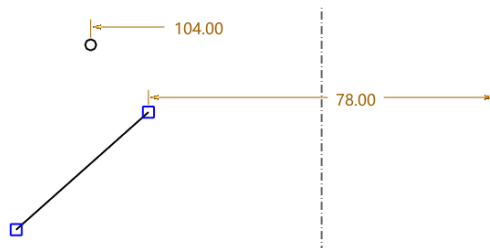
Radius dimension:



Diameter dimension:

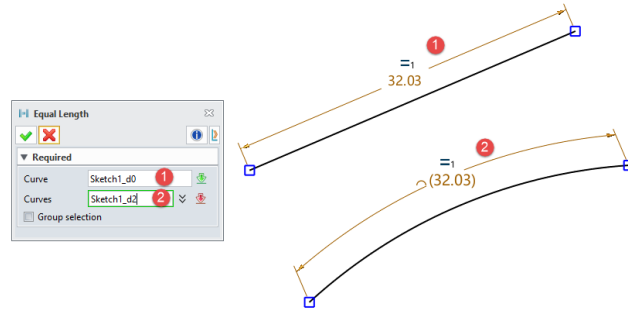


2. Use Symmetry command to pick axis objects, structure lines, reference lines and a line or a point for dimensioning. Switch the mouse on both sides of the axis, construction lines or reference lines to switch the dimension format.

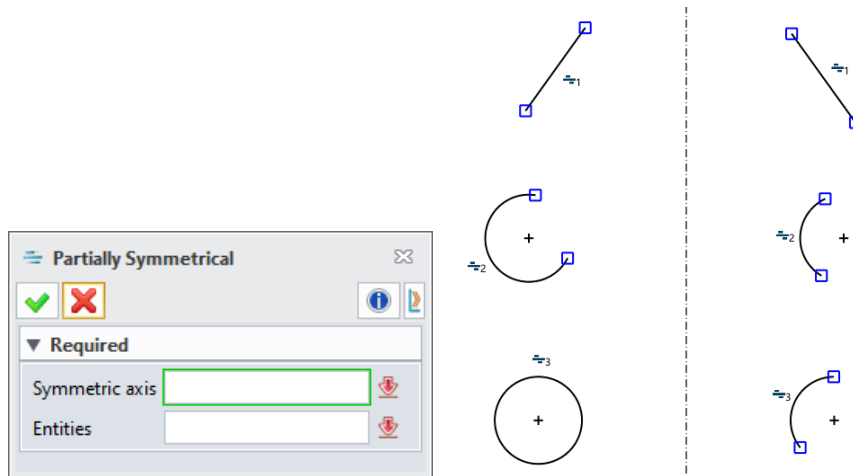


3.1.2 Sketch Constraint Improvement

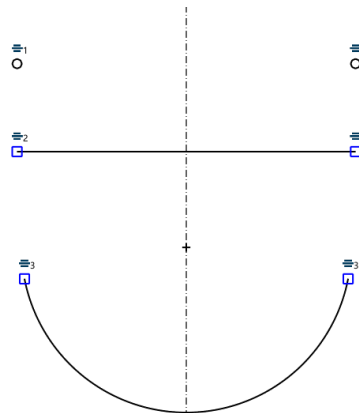
1. Add equal length constraint to linear dimensions and arc length dimensions.



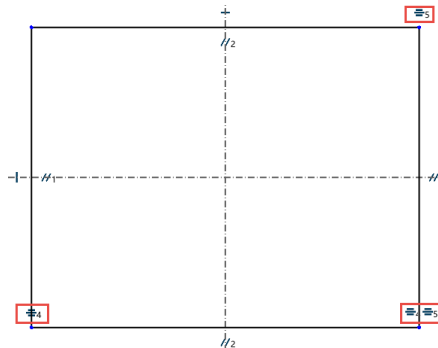
2. Add Partially Symmetrical command which can make lines, circles, and arcs partially symmetrical. For lines, partial symmetry only keeps line slope symmetric. For circles and arcs, the center and radius are symmetric.



3. Expand the scenes of symmetrical constraint capture. When creating points, lines, and arcs, it will automatically identify symmetrical axes and capture symmetrical constraint.

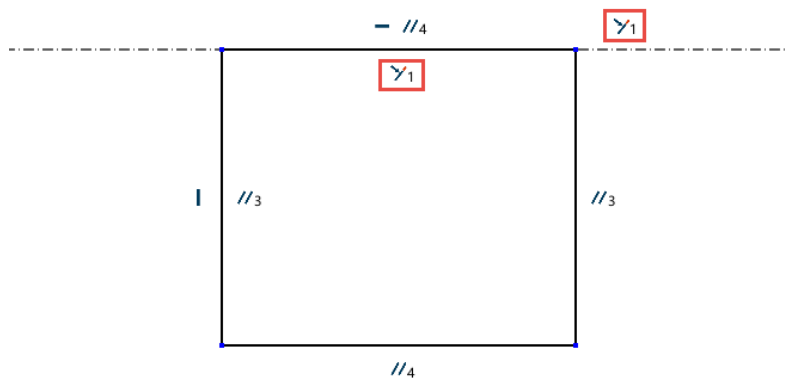


- Added a function that capture symmetrical constraint in two symmetry axes at the same time when create a corner rectangle.



- Optimized constraint addition

We optimized the scenario when capturing two points on a line to create axis, the system will automatically add colinear constraints to axis, as the figure shown below:



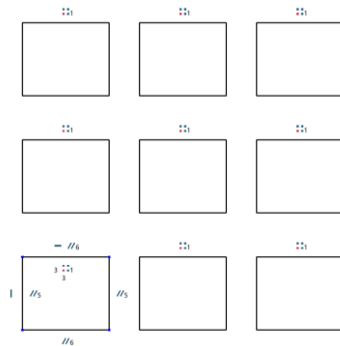
3.1.3 ★Constraint Group

ZW3D implemented sketch constraint grouping management, making it more efficient to manage constrains members in the group, reducing the number of constraint icons, giving a cleaner interface.

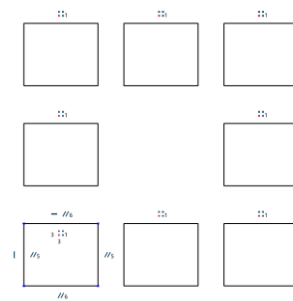
3.1.3.1 Pattern Constraint Group

We optimized sketch constraint group management for a better work efficiency and sketch maintaining in ZW3D. The improvement effect goes as below:

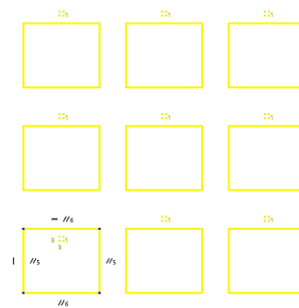
1. Simplified pattern constraint icons with displaying only one pattern icon in each group of pattern objects, which is more concise.



2. Grouped pattern objects. Delete grouped objects (excluding the original object) without effecting pattern constraint of other objects.



3. Group highlighted. Highlight all grouped geometries when hovering mouse over the constraint icons.



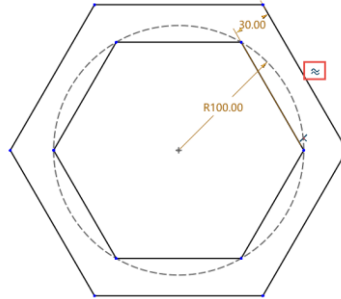
→ Where it is

Part Environment >> Basic Editing >> Pattern

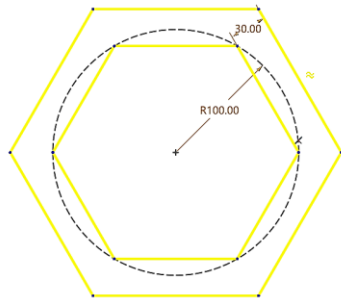
3.1.3.2 Offset Constraint Group

Simplified drawing expression in offset constraint.

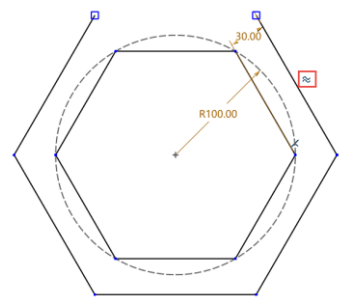
1. Use group management in offset constraint. Display only one offset icon in a grouped offset.



2. Highlight all grouped objects when hovering mouse over the constraint icons.



3. Delete one object in a grouped offset constraint without affecting offset constraint in other objects in the group.

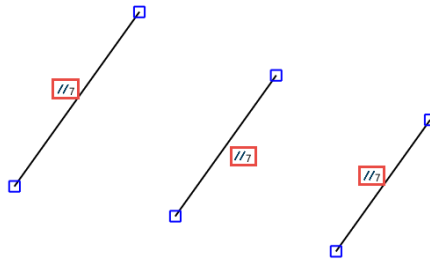


→ Where it is

[Sketch Environment >> Curve >> Offset](#)

3.1.3.3 Simplified Same Constraints Amongst N Geometries

To optimize constraint expression and achieve constraint transfer as well as to brief sketch surface, we simplified single constraint expression amongst N geometries, in placement of constraint displayed between every two objects before, to decrease constraint icon quantity and achieved the unified management of the same constraints.



→ Where it is

Sketch Environment >> Constraint

3.1.4 New “Auto Delete Unused Reference”

ZW3D added “Auto delete unused reference” function. The system will automatically delete the references that are not used for constraint or dimension while still existing in the sketch.

	New Sketch	Re-enter sketch
Auto-delete unused reference	<p>Used reference</p> <p>Unused reference</p>	<p>Reference kept</p> <p>Reference deleted</p>

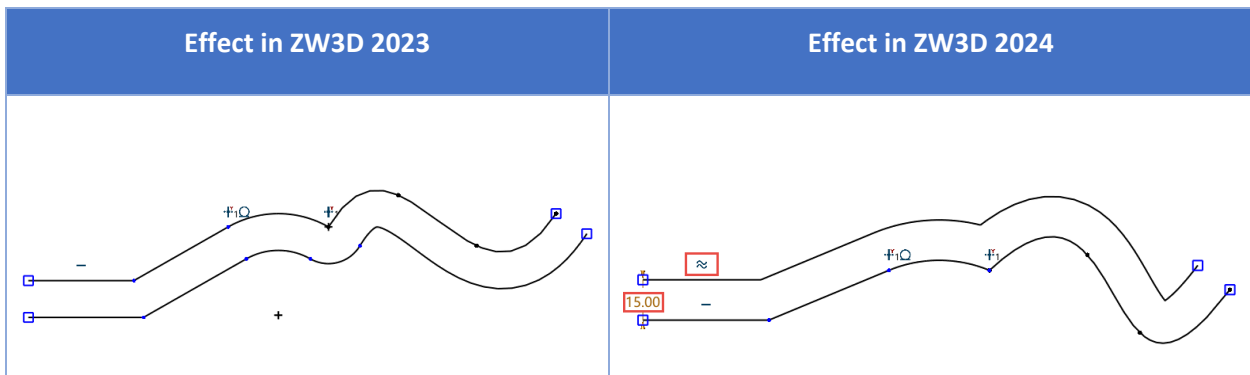
→ Where it is

Configuration >> 2D >> Sketch

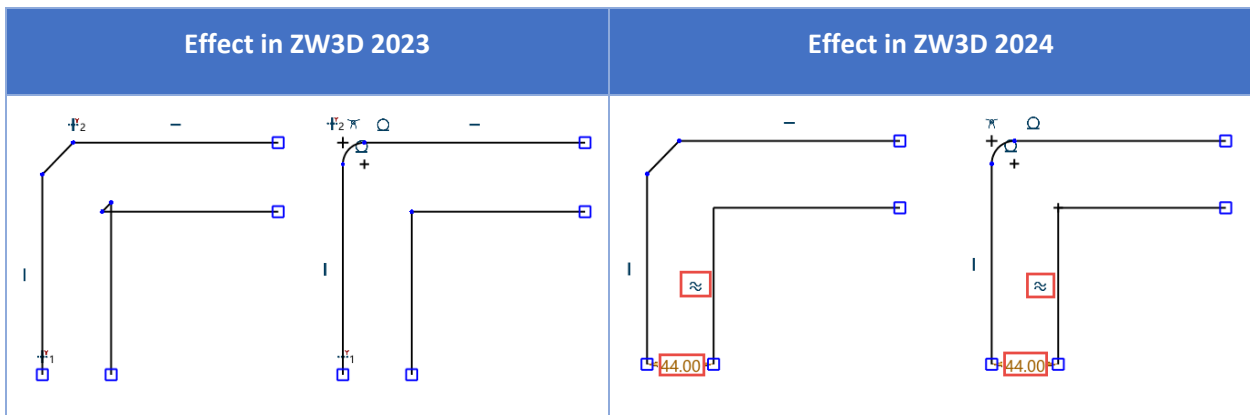
3.1.5 ★Offset Constraint Improvement

ZW3D 2024 optimizes the generation of sketch offset constraint, enriches the geometric types of offset constraints, reduces the behaviors to define sketch, and improves the stability of the sketch profile. The following are the main points:

- 1) Support parameterizing after offsetting curve chain that contains a spline.

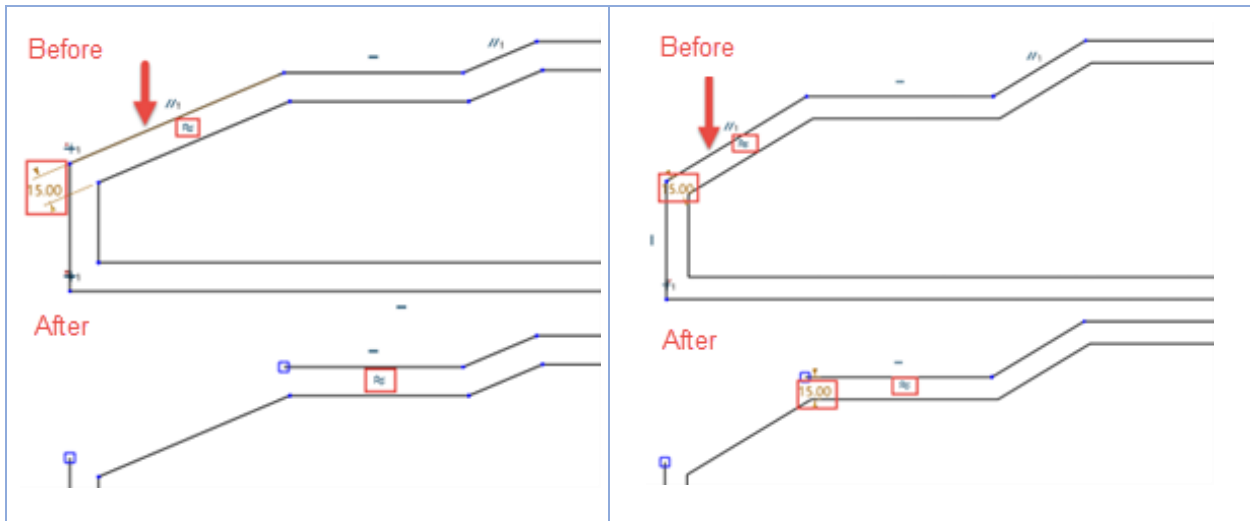


- 2) Support parameterizing after offsetting and removing bowties.



- 3) Improve offset constraint group.





→ Where is it

Sketch >> Constraint >> Offset Constraint

3.1.6 New Text Alignment in ReadySketch Text

ReadySketch Text supports defining the text's start point, so that to align the text.

	Left	Middle	R i g h t
Top	ZW3D 2024 NewText	ZW3D 2024 NewText	Z
Waist	ZW3D 2024 NewText	ZW3D 2024 NewText	Z
Bottom	ZW3D 2024 NewText	ZW3D 2024 NewText	Z

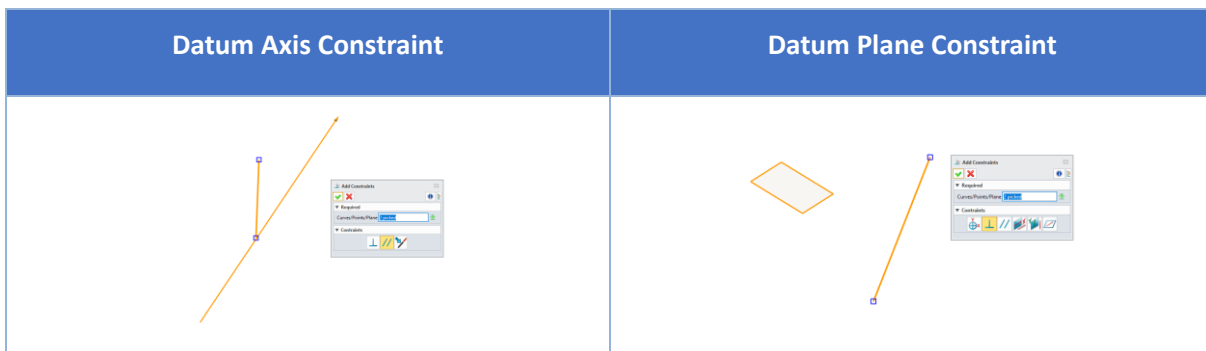
- 1) Support inserting special characters and stacked text and support setting color.

→ Where is it

Sketch >> Sketch >> Drawing >> ReadySketch Text

3.1.7 3D Sketch Constraint Improvement

3D sketch datum axis and datum plane are incorporated into the 3D sketch constraint solving system in ZW3D 2024, which support constraining the datum axis and the datum plane with other graphic primitives in the 3D sketch.



→ Where is it

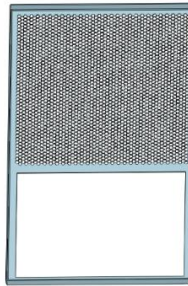
3D Sketch >> Add Constraint

3.2 Shape Design

3.2.1 Pattern Improvements

3.2.1.1 ★ “Pattern Feature” Efficiency Improvement

ZW3D 2024 raises the efficiency of the pattern feature in partial scenarios. In this version, if the instances of pattern do not interface with each other, the patten efficiency will be improved greatly by over 80%.

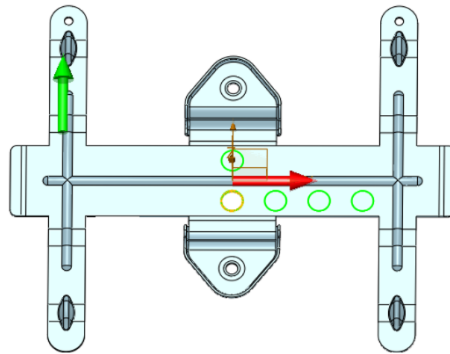
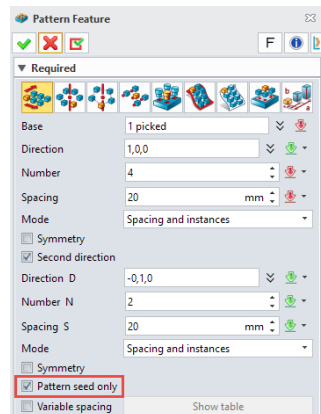


→ Where is it

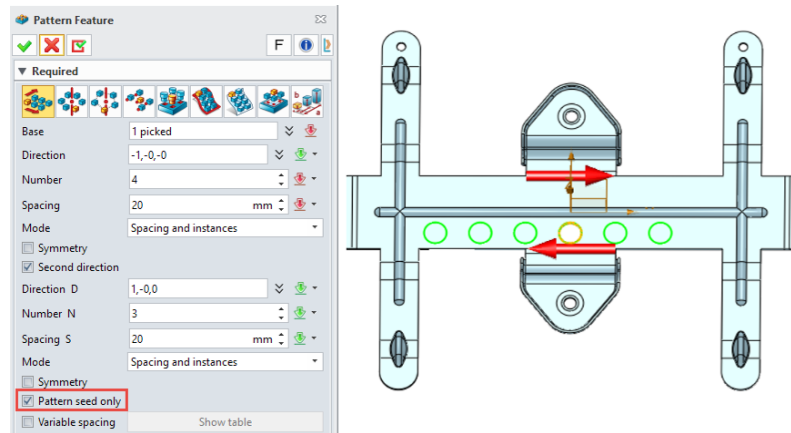
Part >> Shape >> Basic Editing >> Pattern Feature

3.2.1.2 Linear Pattern Feature/Pattern Geometry Improvement

When linear pattern part features in two directions, user can choose only pattern source features in the second direction. By checking the option "Pattern seed only" in the second direction, the pattern in the second direction will not contain the results of the pattern in the first direction.



Support using the opposite direction of the first direction as the second pattern direction.



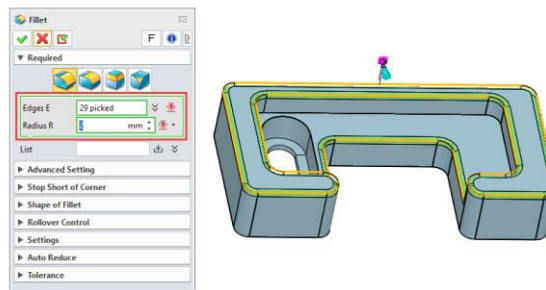
The improvement in linear pattern geometry is like the above-mentioned linear pattern feature.

→ Where it is

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

3.2.2 New Multiple Controls Activation Effect in Fillet/Chamfer

When the fillet/chamfer command is enabled, user can directly use the keyboard to input the radius parameter while selecting the input edge, without switching input fields, realizing the effect of activating multiple controls at the same time and improving work efficiency.



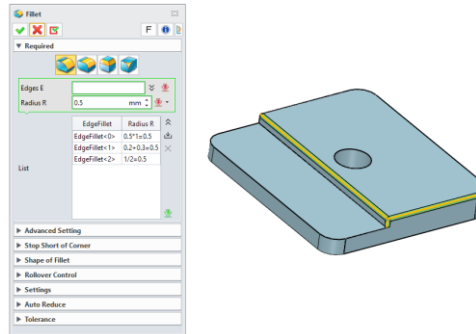
→ Where it is

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

3.2.3 Fillet Improvement

3.2.3.1 Fillet UI Enhancements

1. Add viewing fillet expressions function in fillet command. Save fillets to the list and view the parameter expressions of the listed fillets.

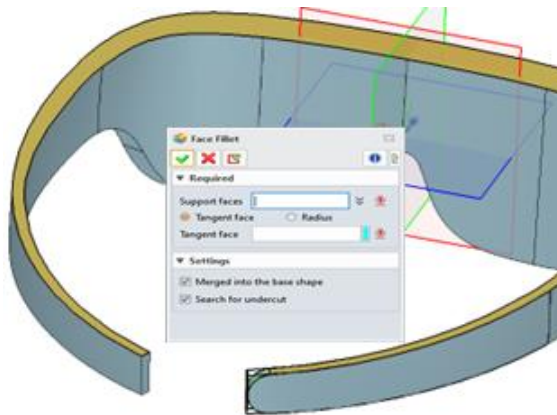


→ Where it is

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

3.2.3.2 New Face Fillet

The function of face fillet is based on selecting faces to create fillets. For instance, you can create a fillet based on selecting the fillet face and the specified radius or based on the tangent faces to create a full round fillet.

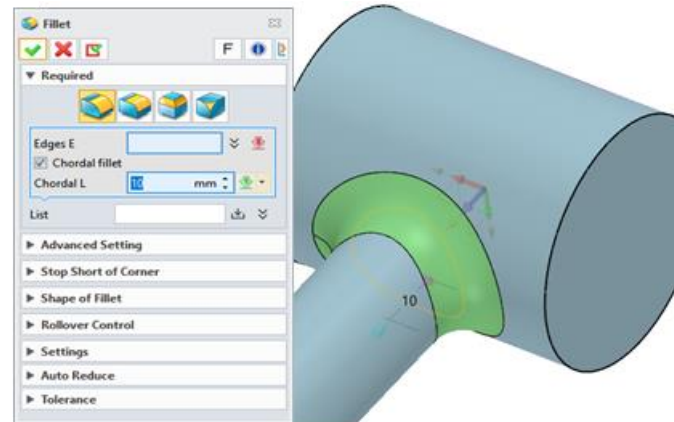


→ Where is it

Part >> Shape >> Engineering Feature >> Face Fillet

3.2.3.3 New Chordal Fillet

The function of chordal fillet can make the distance even between the two created fillet edges, to meet the requirement of aesthetic fillet related to shape and appearance.



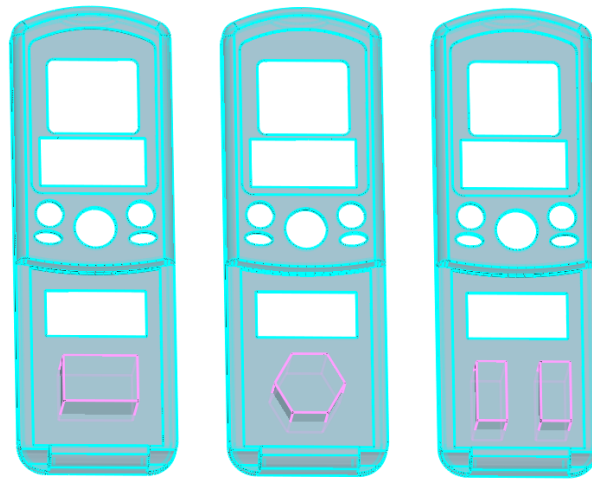
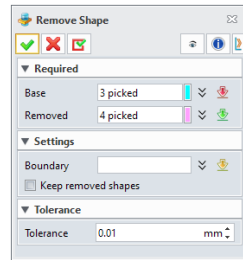
→ Where is it

Part >> Shape >> Engineering Feature >> Fillet

3.2.4 Boolean Improvements

3.2.4.1 Remove Shape and Intersect Shape Improvement

Remove Shape and Intersect Shape commands support applying to multiple bases in ZW3D 2024. Remove Shape and Intersect Shape improve their supporting ability. Combine the Remove Multiple Shapes command to the Remove Shape command; combine the Intersect Multiple Shapes command to the Intersect Shape command.

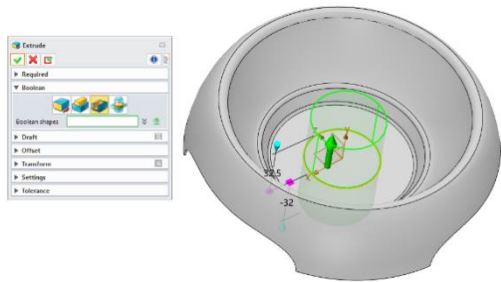
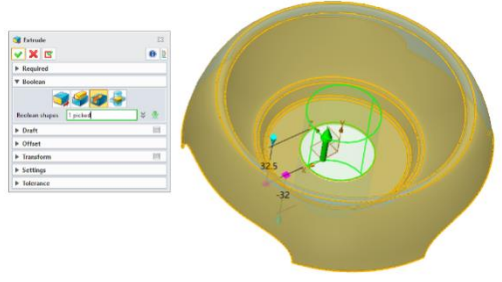


→ Where is it

Part/Assembly >> Shape >> Edit Shape >> Add Shape/Intersect Shape

3.2.4.2 Boolean Selection Adjustment

- 1) Auto select: when you are extruding shape and activating the Boolean function, if there is only one visible shape, the shape will automatically register the Boolean selection box.

ZW3D 2023	ZW3D 2024
	
<p>You need to select manually</p>	<p>Auto-register the sole visible shape</p>

- 2) Adjusted Boolean Add behavior in the basic shape modeling commands

Only one shape can be picked into Boolean field in the basic shape modeling command, like

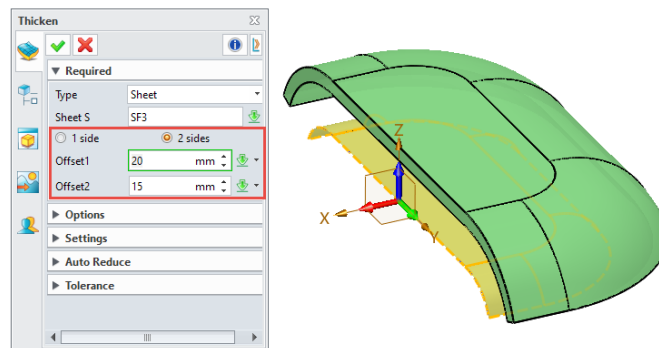
Extrude, Revolve, etc., to clarify its sole basic shape.

→ Where is it

Part/Assembly >> Shape >> Extrude >> Boolean

3.2.5 Thicken Supports 2-side Offset

ZW3D 2024 supports two-side offset to thicken, which combines thicken and offset into one function so to reduce feature usage. For the design panel and other commonly used thicken function industry, it can effectively shorten the history length and improve the usability of thicken.

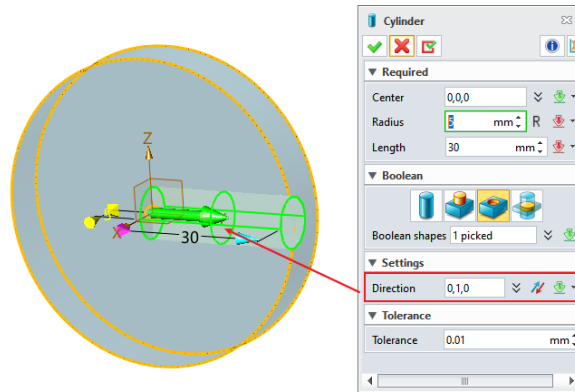


→ Where is it

Part >> Shape >> Edit Shape >> Thicken

3.2.6 Cylinder/Cone Creation

The cylinder/cone creation behavior has been improved. You can set directly according to the direction of axis, which can meet the more complex design environment to quickly create a cylinder or cone model.

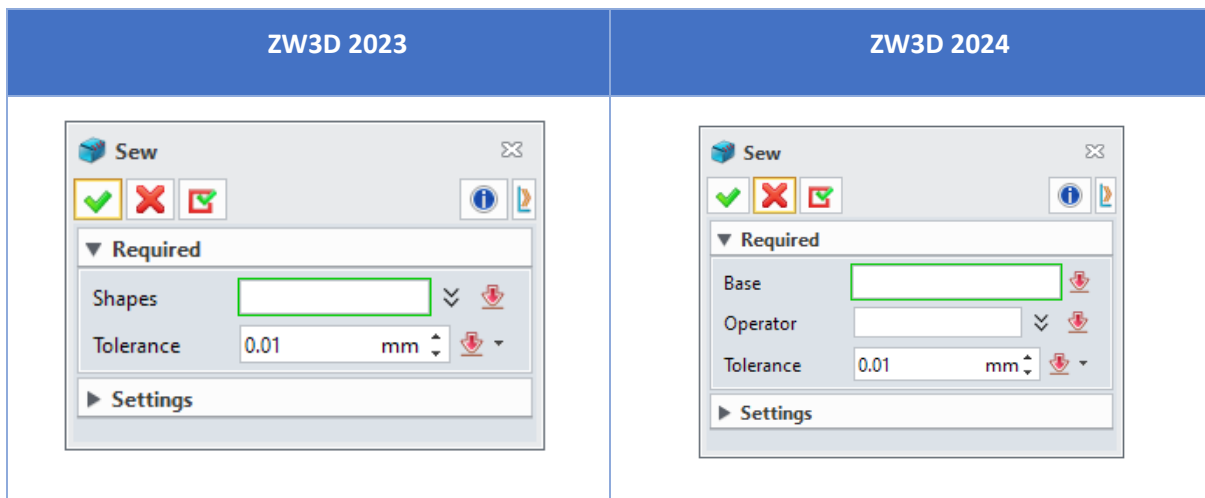


→ Where is it

Part >> Shape >> Basic Shape >> Cylinder/Cone

3.2.7 Sew Command Improvement

Newly add “Operator” and “Tool” which replace the original “Shapes” in Sew command panel. Make it clear that the sewing result will inherit the properties, improving the stability of history regeneration.

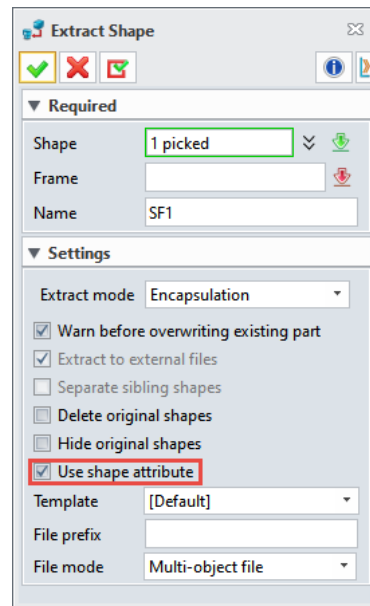


→ Where is it

Part/Assembly >> Heal >> Face >> Sew

3.2.8 Extract Shape Improvement

When extracting a shape, you can check the option “Use shape attribute” and save the user’s time in resetting time after extract shape.

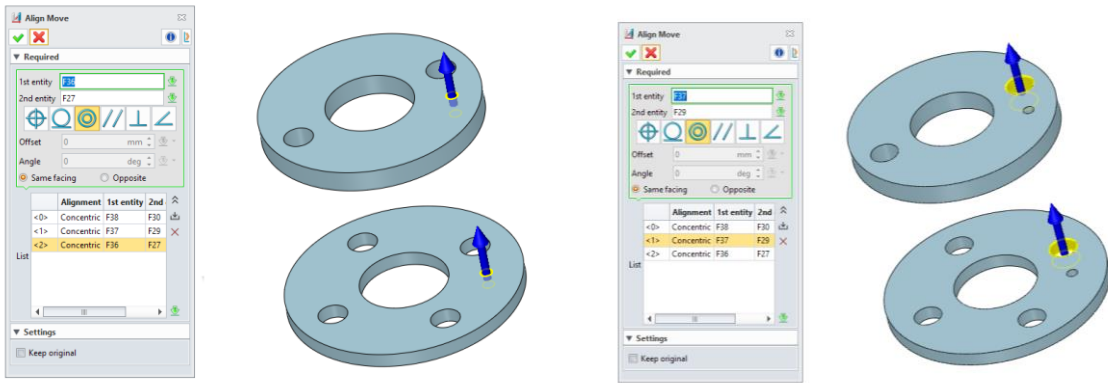


→ [Where is it](#)

[Part](#) >> [Data Exchange](#) >> [Export](#) >> [Extract Shape](#) >> [Settings](#)

3.2.9 “Align Move” Supports Multiple Groups of Constraints

Align Move command supports multiple groups of constraints. We added constraint list to Align Move. When a group of constraints is set, you can save the constraints to the list by clicking the “Add to list” button and add multiple groups of constraints to the list.

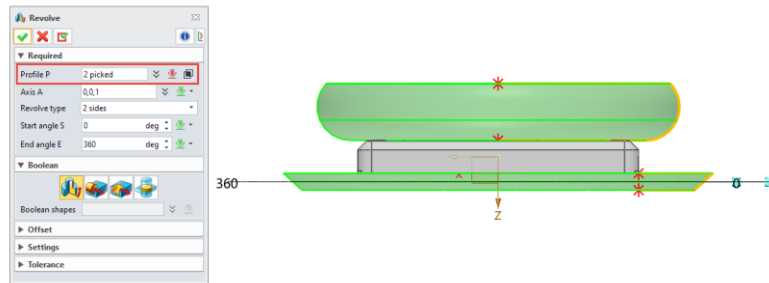


→ Where it is

Part Environment >> Shape >> Basic Shape >> Align Move

3.2.10 Revolve Command Supports One-time Selecting Multiple Objects

In the revolve command, the profile option box will not automatically jump to the next axis option box when input is made. The profile option input box supports the input of multiple profiles to revolve multiple objects at one time, avoiding using commands repeatedly to achieve modeling.

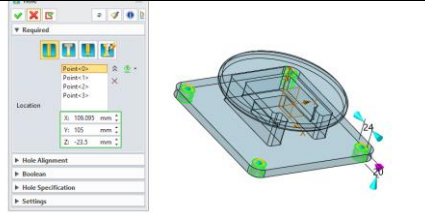
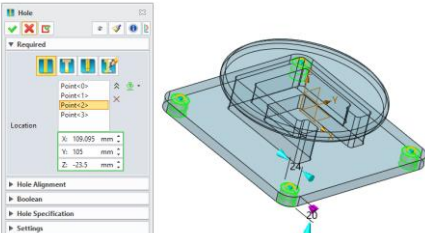


→ Where it is

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

3.2.11 New Hole Feature Selecting Effect

When there is multi-point editing in the hole feature, select one of the points to display the parameter definition preview of the point in the drawing area.

Choose	Result
Point 0	
Point 2	

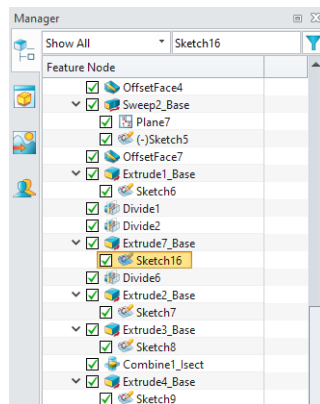
→ Where it is

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

3.3 History Feature

3.3.1 History Search Optimization

ZW3D supports searching built-in features in history manager. The built-in feature containing search text would be searched and highlighted.

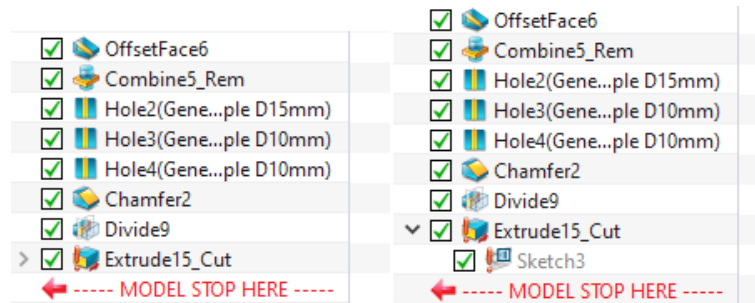


→ Where it is

Part Environment >> Manager >> History Manager

3.3.2 Feature Tip Optimization

When a warning tip occurs to built-in feature, the warning tip will respond to the attached feature, so that user can easily find that feature even it is folded.



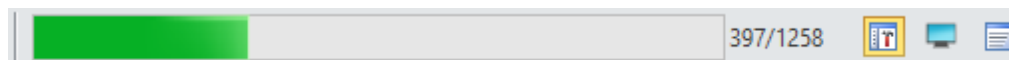
→ Where it is

Part Environment >> Manager >> History Manager

3.3.3 History Regeneration Improvement

We promoted history playback efficiency with 10%~30% increase comparing with ZW3D 2023. The longer history is, the higher efficiency will be.

We also change the regeneration progress bar from percentage to steps to show the regeneration progress more intuitively.

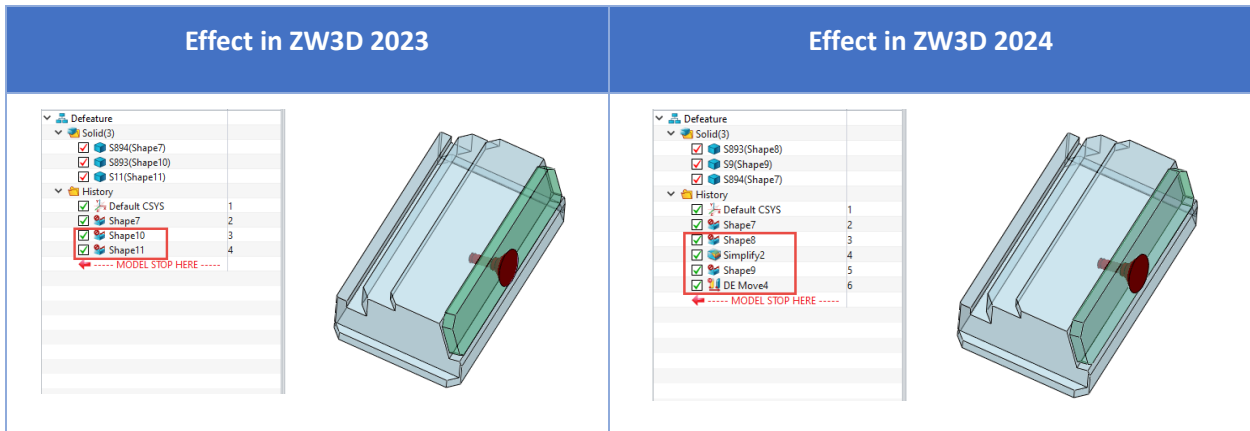


→ Where it is

Part Environment >> Manager >> History Manager

3.3.4 Defeature Improvement

The defeature mechanism is improved so that the parameters of the referenced body will not be removed. For instance, when shape1 top face is replaced to shape2 top face, the shape1 parameters will be removed and the shape2 parameter will be kept.

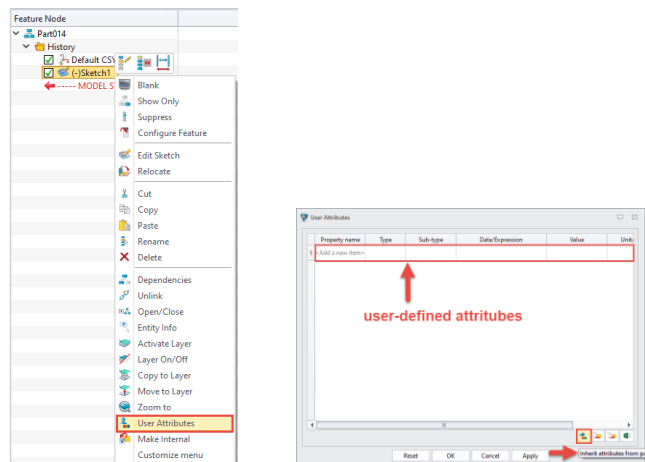


→ Where is it

Part/Assembly >> Tools >> Utilities >> Defeature

3.3.5 Support Adding User Attributes

We added user attributes option to the feature context menu in feature manager. You can select user-defined attributes or inherit attributes from parts.



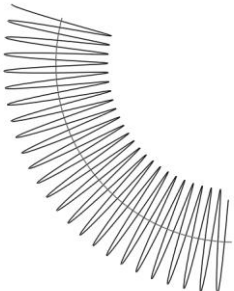
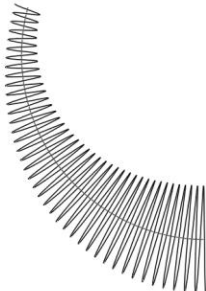
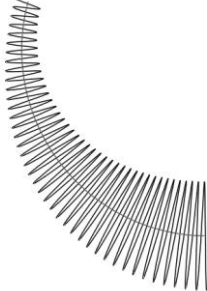
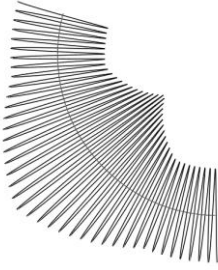
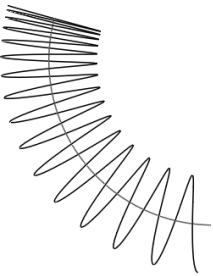
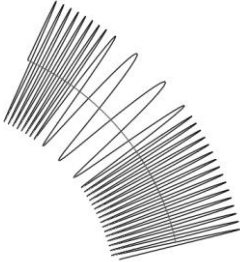
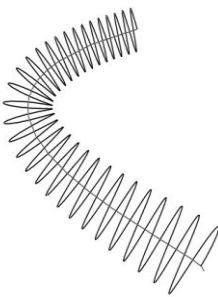
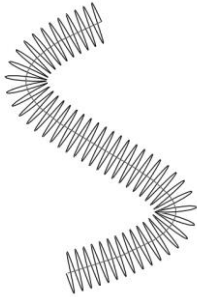
→ Where it is

Part Environment >> Manager >> Sketch >> Context Menu

3.4 Wireframe Design

3.4.1 ★Spiral Helix Improvement

We optimized the creation of spiral helix in ZW3D by adding a new method of creating spiral helix along curve path. User can adapt variable radius and variable pitch at the same time. Support user to control the shape of spiral helix with any two variables of revolution number, pitch distance or length.

Constant radius and pitch	Linear variable radius	Degree 3 radius	Customized radius
			
Linear variable pitch	Customized variable pitch	Both variable radius and variable pitch	
			

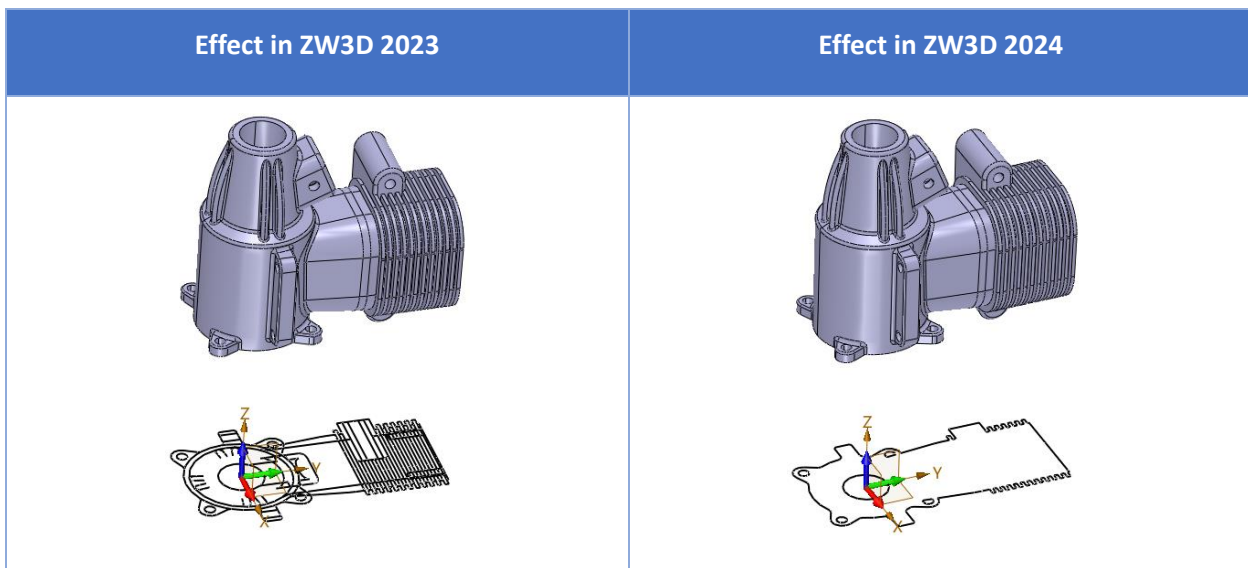
→ Where it is

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

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

3.4.2 ★Silhouette Curves Improvement

The internal mechanism optimizes the projection ability of silhouette curves, solves the issues of missing and multiple lines when projecting internal and external silhouette, greatly improves the correctness of the internal and external silhouette of the entity projection.

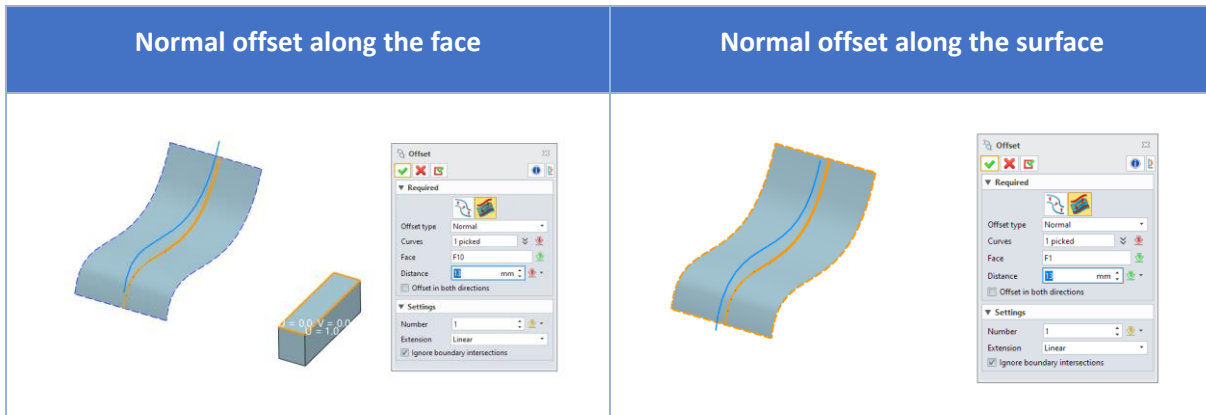


→ Where is it

Part/Assembly >> Wireframe >> Curve >> Silhouette Curves

3.4.3 Offset Improvement

ZW3D 2024 adds “Normal” as offset type when offsetting curves on plane. When the selected face is a plane, allow any curve offsetting along the selected plane normal. When the selected face is a surface, limit the offsetting curve as the curve on the selected surface, which can achieve the normal offset the selected curve along the selected surface.

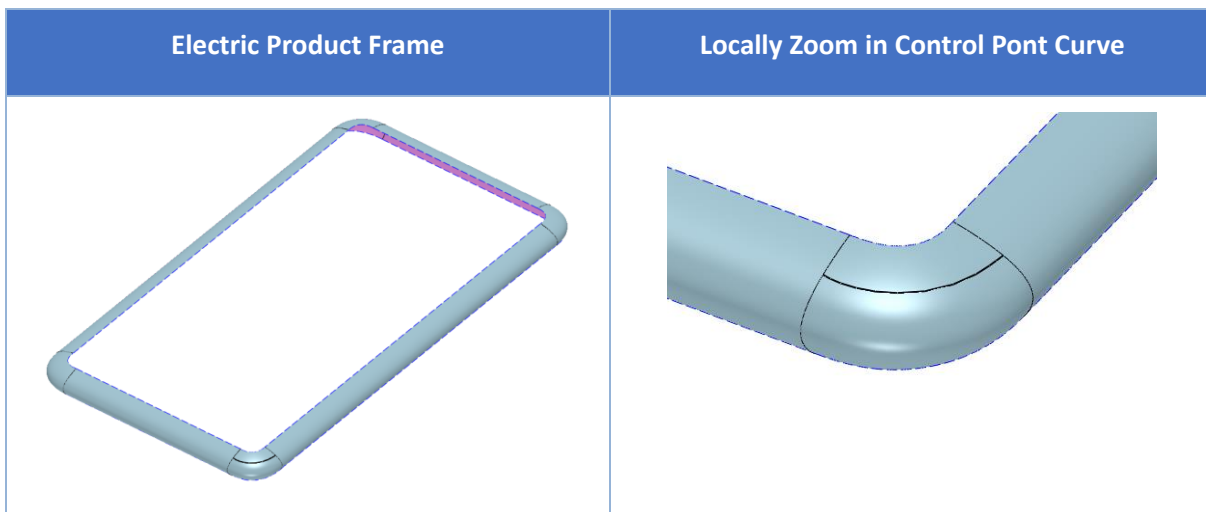


→ Where is it

Part/Assembly >> Wireframe >> Curve >> Offset >> On face

3.4.4 Curve on Face Thru Pts

We add “Control point” type to Curve on Face Thru Pts, which can help users adjust the shape of curve and ensure curve curvature changes smoothly and improve surface quality. It is widely used in the surface design of frame of electric products.



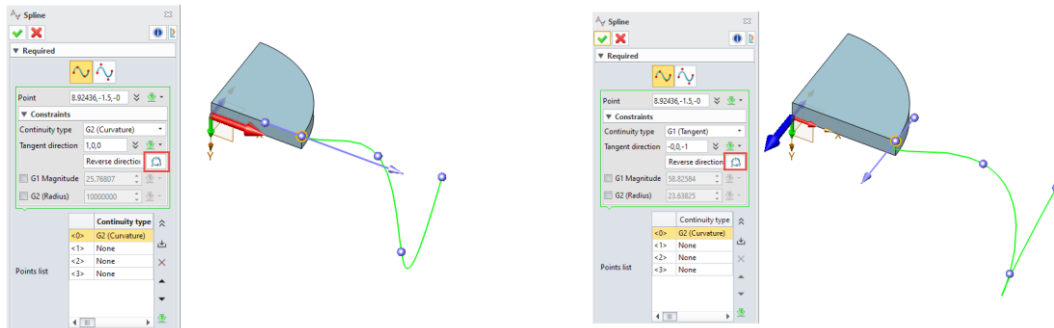
→ Where is it

Wireframe >> Curve >> Curve on Face Thru Pts

3.4.5 Spline and Curvature Continuity Constraint

When creating a spline curve in the part environment, if the spline node connects with “Edge” or

“Plane”, you can click the new control to switch the connect object after setting the G_n type.

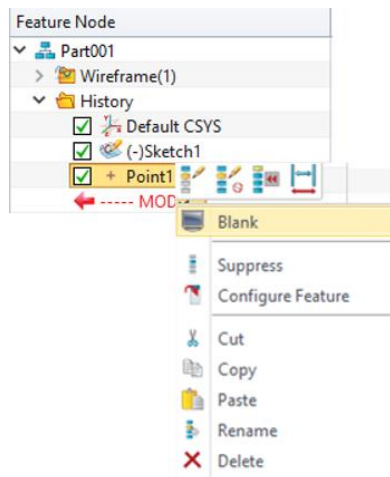


→ Where it is

Part Environment >> Wireframe >> Curve >> Spline

3.4.6 Support Points Blank or Unblank Display in Manager

Added Blank/Unblank option in points context menu. User can select to blank or unblank points through right-clicking points in the manager.

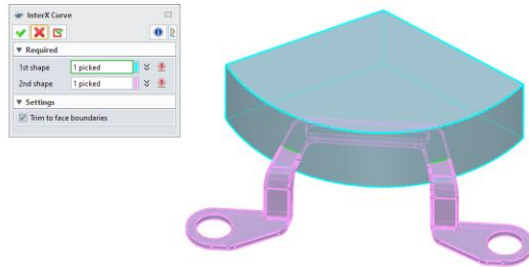


→ Where it is

Part/Assembly Environment >> Manager >> Point >> Context Menu

3.4.7 New Preview to InterX Curve

Added a preview effect of intersecting curves on the drawing area when using the intersecting curve command. When user selects the input object, the curve obtained by the intersection of the two object surfaces will be previewed on the drawing area.



→ Where it is

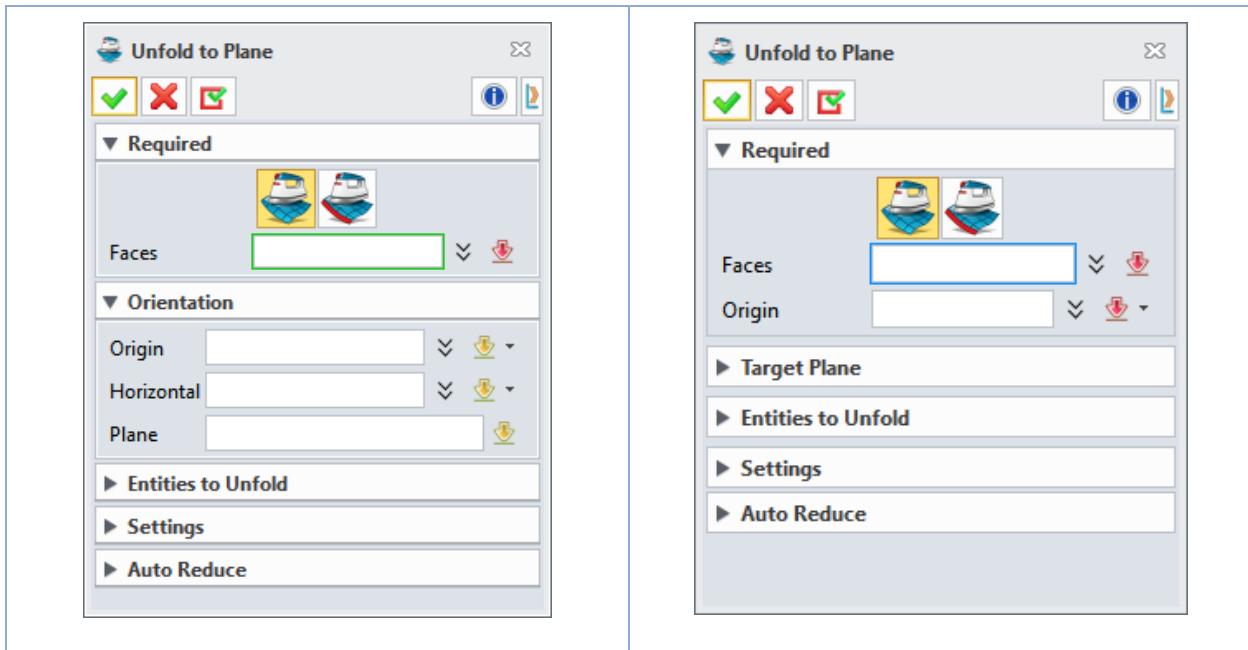
Part Environment >> Wireframe >> InterX Curve >> InterX Curve

3.5 Free Form Design

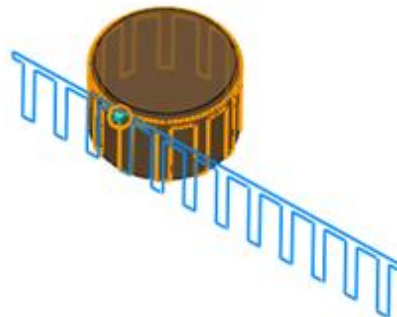
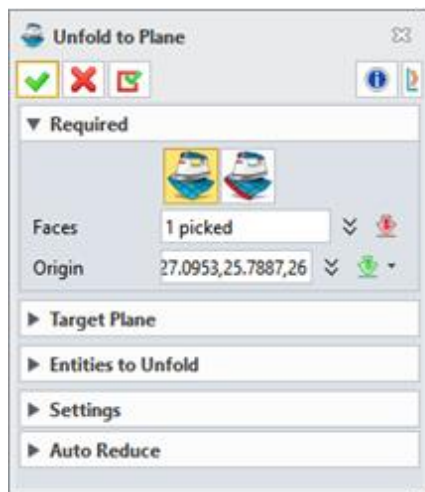
3.5.1 Unfold to Plane Improvement

ZW3D 2024 improves the design workflow and interface of Unfold to Plane command.

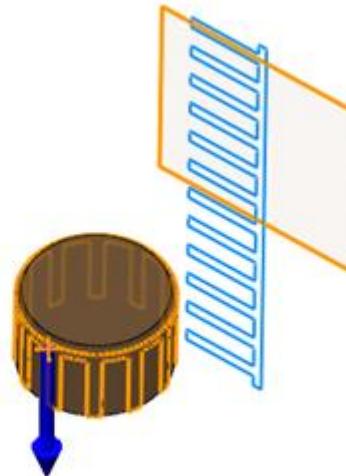
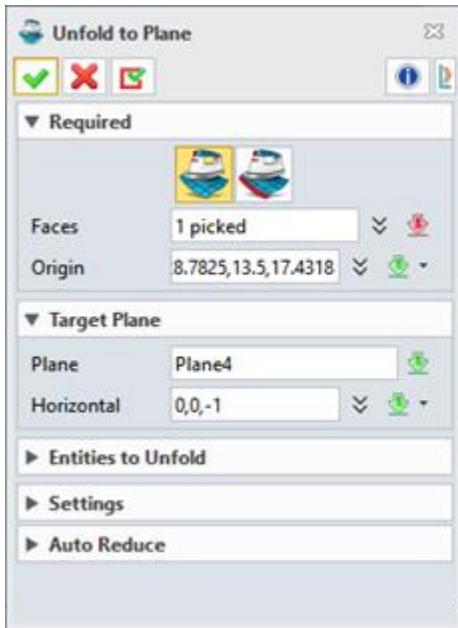
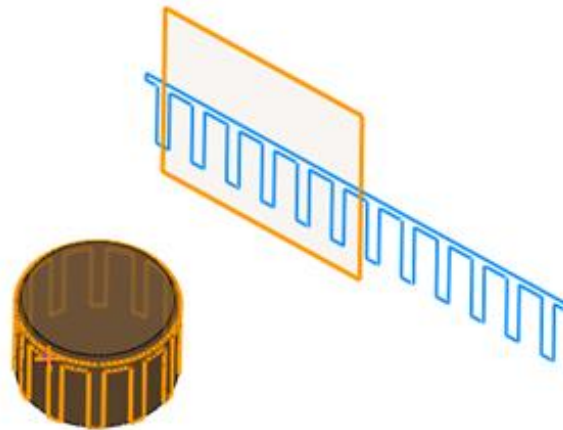
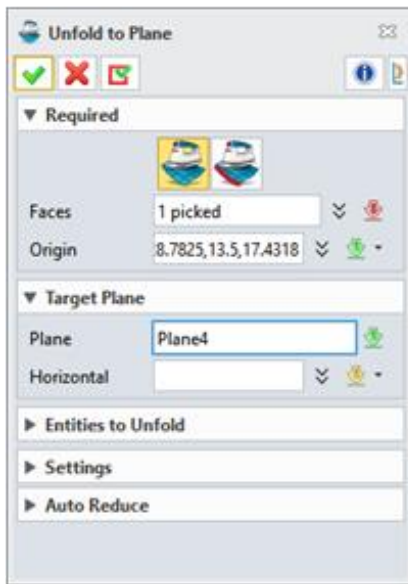
UI in ZW3D 2023	UI in ZW3D 2024
-----------------	-----------------



- 1) The “Origin” is changed as a required option. Select an unfolded face and specify a point as “Origin” on the unfolded face, which means a surface can be unfolded as a plane at the “Origin”.



- 2) The “Orientation” is changed to “Target Plane”, adjust the order of “Plane” and “Horizontal”. Firstly select “Plane”, and then the unfolded plane can be placed to the selected plane. At the same time, the “Horizontal” orientation can define the unfolding plane’s axis X, so that it’s easy to adjust angle for placement. This is the specified placement.



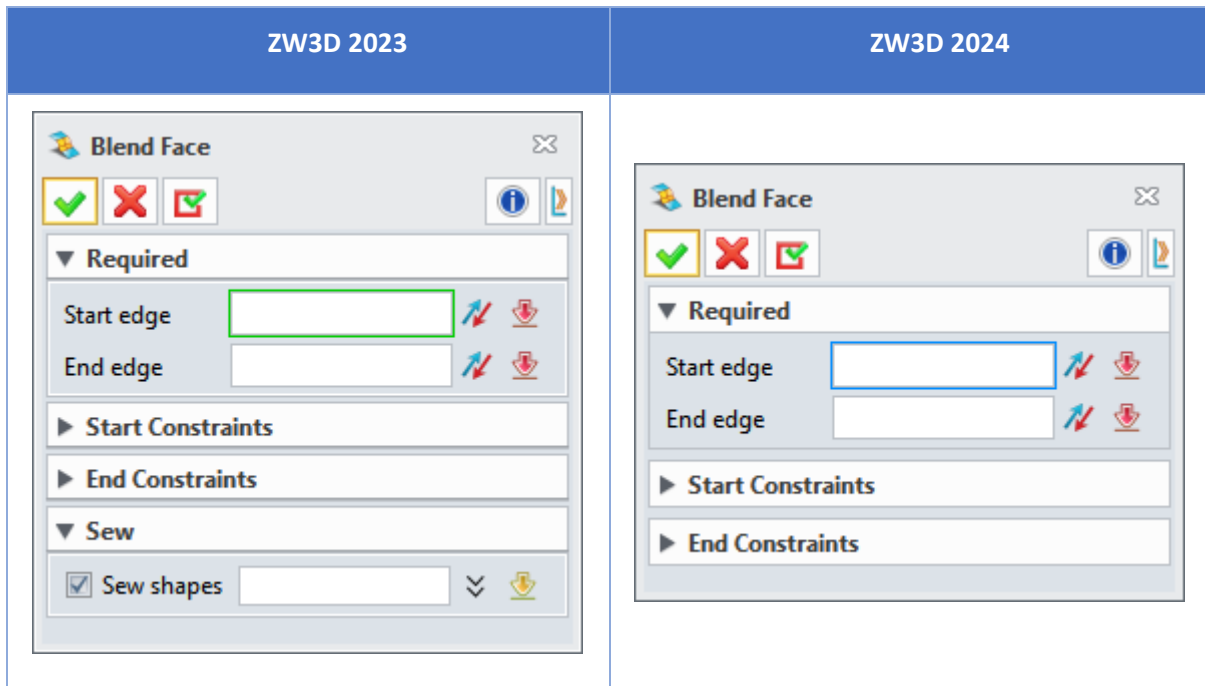
With the improvement, Unfold to Plane command is more suitable for user design flow and can enhance user experience with a faster and more effective surface design.

→ [Where is it](#)

[Part/Assembly >> Free Form >> Unfold to Plane](#)

3.5.2 Remove Sew Shapes that Built-in Free Form

To increase the stability of free form commands, the function of sew shapes in removed from the free form commands. Started from ZW3D 2024, user need to sew faces through Sew command, which can improve the stability of face feature regeneration after modifying parameters and increase the cost of recovering model and promote the design efficiency.



→ Where is it

Part>> Free Form >> Blend Face

3.6 Assembly Design

3.6.1 ★Improvement of Assembly Efficiency

The assembly efficiency had been significantly increased. In a large assembly, the efficiency in blank/unblank, suppress/unsuppress, add constraints, blank/unblank external datums, delete, etc. can be improved up to 80%.

Due to optimize the internal algorithm logic of the assembly tree, the assembly efficiency had been significantly increased. Comparing with ZW3D 2023, in a large assembly, the efficiency in blank/unblank, suppress/unsuppress, constraints, blank/unblank external datums, delete, etc. can be improved up to 77% on average.

Function	Blank/unblank	Suppress/unsuppress	Blank/unblank external datums	Delete	Fix	Add constraints	Redefine constraint
Speed Increase Rate	80%	87%	89%	90%	72%	74%	51%

→ Where it is

Assembly Environment

3.6.2 ★Top-down Design Based on Master Layout

To better support top-down design, ZW3D 2024 adds “Master Layout” model, by which the designers can use the layout model to define the design purpose and the overall product architecture information. At the same time, through the function of publishing geometry and copying geometry, the necessary design information in the layout can be transferred from the general assembly to each sub-assembly or sub-part.

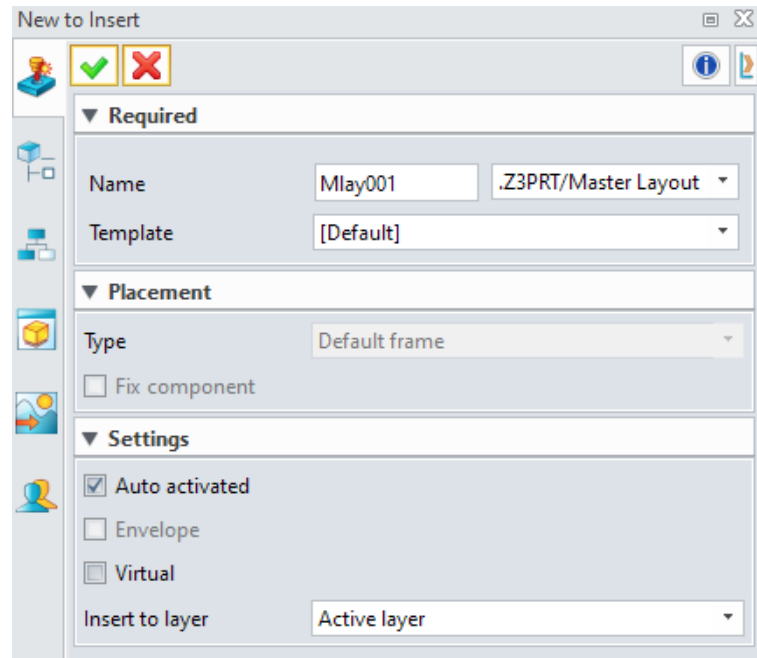
Top-down design based on layout model has two outstanding advantages:

- 1) Through layout model, you can decompose tasks more effectively and assign tasks to different co-designers.
- 2) Through modifying layout model, the modification will be quickly transferred to the corresponding sub-assembly or sub-part, so that to achieve the effect of quick design change or rapid variant design.

3.6.2.1 Master Layout Introduction

- 1) Master layout model can only be created in the assembly module when creating a new model

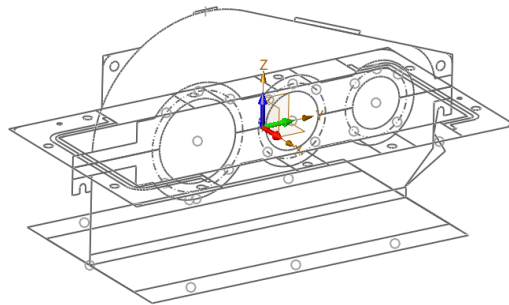
and selecting “Master Layout” from the drop-down list. Partial industry functions are removed in the created master layout models and only keep corresponding part’s modelling function.



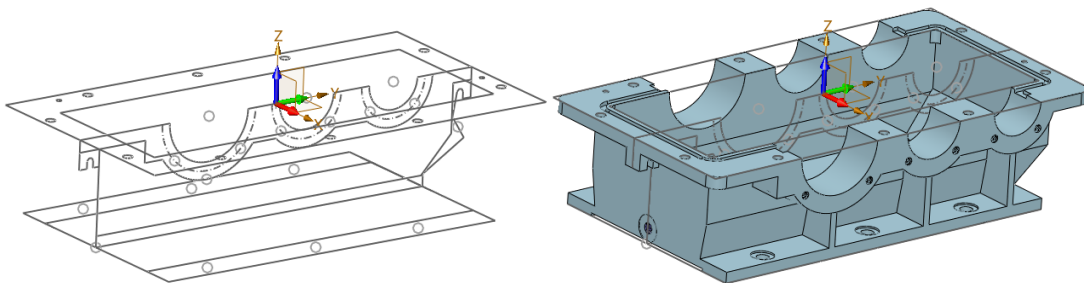
- 2) To highlight master layout model as “design control intention” model, whenever newly created master layout model will be in the first node of layout folder on the assembly tree, and do not allow to drag layout model on the assembly tree.
- 3) As the purpose of master layout model is “design control intention” which does not participate in the actual producing assembly, therefore the quality of master layout model will be not counted to the assembly model. 3D BOM will not record the master layout model, either.
- 4) To ensure the correctness and stability of master layout design information transfer, the following settings are done to the master layout model:
 - A. The crated master layout model coordinate keeps the same as the assembly of creating level.
 - B. Do not support the master layout model being replaced by the general part. The commands of Merge, Pattern, Mirror, Cut, Hole, Chamfer, and Fillet do not work in the master layout model.
 - C. Do not support master layout model reference part model or assembly model.
 - D. Do not support upper master layout model reference to lower part model or assembly model.

3.6.2.2 Recommended Top-down Design

Take the design of the lower box of a gear box as an example. First, create the general assembly of the gear box, and then create the layout model. In the layout model, the overall size, the sketch of the lower box and the upper box are outlined clearly, as follows:



Use the functionality of Publish Set in Data Exchange to publish the design information of lower box. Create a corresponding part and select master layout model to copy the corresponding information to the part through the functionality of Geom to Copy. That is, modeling can be done based on reference design datum information, as follows:



If modification is in need, you only need adjust the sketch dimension of the master layout model, and then refresh the lower box model.

After confirming and finishing the low box design, you need only to insert it into the general assembly through the default coordinate.

3.6.3 ★Link Manager Improvement

Optimize the link manager processing logic to pinpoint the link Ref record object to a specific reference object. For example, whereas previous versions of sketches referred to record objects as entities, the new version records to specific edges. This further reduces the problem of component obsolescence caused by changes to unrelated items.

Old

New

Three types are added to the link relation type: assembly feature reference, history feature reference, and component reference.

- Assembly feature reference: it refers to the selection of external geometry or variables as reference inputs to complete the design when the assembly pattern and assembly mirror are used in the assembly design environment.

- History feature reference: it refers to the selection of external geometry or variables as reference inputs to complete the design when the part features are created in the part design environment.
- Component reference: it refers to the information completing design of reference assembly components in the assembly environment. For instance, the reference occurs in relation to the location of the component in the assembly environment.

→ [Where is it](#)

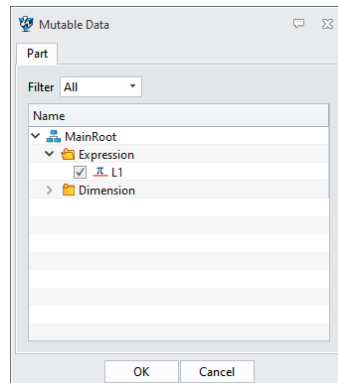
[Tools >> Link Manager](#)

3.6.4 ★Flexible Part

ZW3D2024 supports creating flexible parts which can stimulate the flexible change process of parts such as springs in the assembly environment. The mutable design includes two main steps:

1) Mutable data predefinition

Predefine the mutable data in the part design environment. If you've predefined mutable data, it will pop-up flexible design panel when you're assembling components. The specific case is as follows:



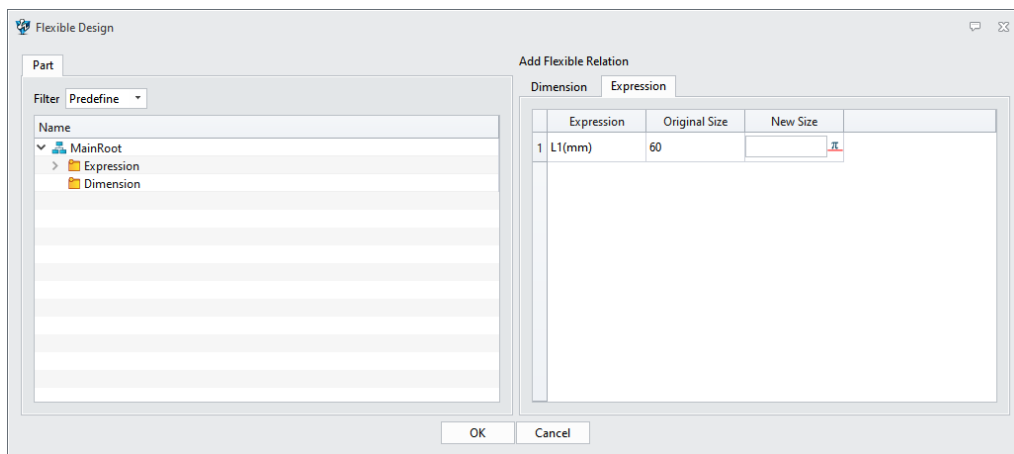
Note: You can also predefine mutable data in the assembly design environment.

2) Add flexible driven relation.

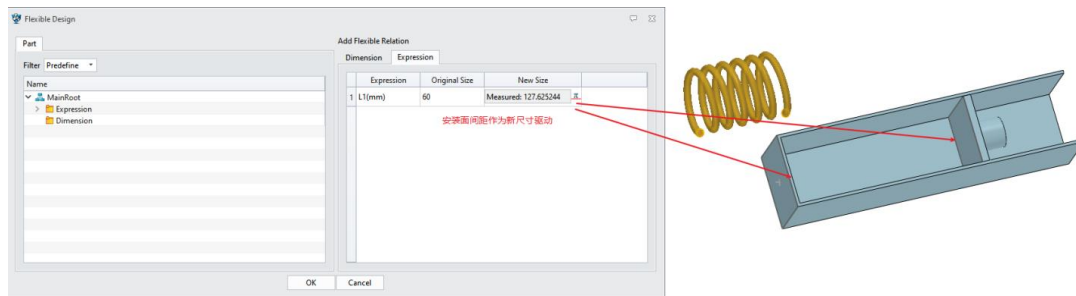
Insert parts to the assembly file.



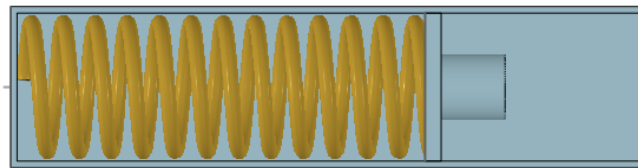
Insert predefined parts of mutable data, click OK, the system will pop-up a flexible design panel where you can continue to add mutable data.




Add driven relations to mutable data. According to different data type, you can add different relations to mutable data. As the below figure shows, add the distance between the two-installing surface for the spring length as a drive.

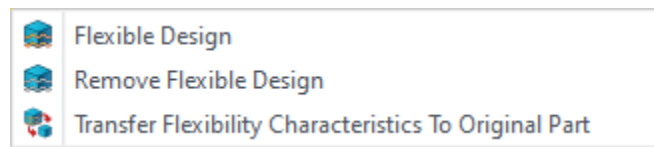


After flexible design, there will be a special mark on the assembly component node.



 (-)Spring

In addition to carry out flexible design when inserting assembly's components, you can select Flexible Design command by right clicking menu to create flexible parts. There also add Remove Flexible Design and Transfer Flexible to Source Part



Notes:

- 1) Flexible parts do not support adding assembly context features (assembly fillet, assembly chamfer, assembly hole, and assembly cut).
- 2) After assembly context features are added, the flexible parts cannot be set up anymore.
- 3) When components are referenced externally by other parts, flexible parts cannot be set up.

→ [Where is it](#)

[Assembly >> Component's Context Menu >> Flexible Design](#)







3.6.5 Interference Check Interactive Selection Improvement

ZW3D 2024 optimizes interactive selection logics of assembly interference check result. The new interactive logic is close to the Windows interactive behavior, which decreases user's learning cost and enhances the convenience of the operation.

- 1) Optimize interference check result single option logic: cancel the checkbox of interactive mode and update click-to-select interactive mode. The interference check that user clicks is displayed.
- 2) Optimize interference result multiple options logic: press Ctrl key to click interference result and multiple select one by one; press Shift key to batch select all results between the two

interference check results.

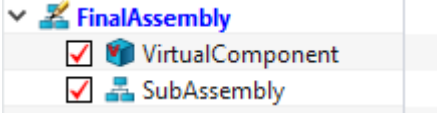
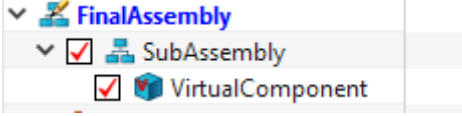
- 3) Optimize ignored interference check result interaction: when user ignores the current selected interference check result, the next interference result will display automatically in the list and highlight in the list.

Effect in ZW3D 2023	Effect in ZW3D 2024
<ul style="list-style-type: none"> > <input checked="" type="checkbox"/>  Interference 1 - 375.000000 mm³ > <input type="checkbox"/>  Interference 2 - 375.000000 mm³ > <input type="checkbox"/>  Interference 3 - 125.000000 mm³ 	<ul style="list-style-type: none"> >  Interference 1 - 375.000000 mm³ >  Interference 2 - 375.000000 mm³ >  Interference 3 - 125.000000 mm³

→ Where is it

Assembly >> Assembly >> Inquire >> Interference Check

3.6.6 Virtual Component Support Moving to Other Sub-assembly

Effect in ZW3D 2023	Effect in ZW3D 2024
	
Virtual component cannot be moved to sub-assembly	Virtual component can be moved to sub-assembly

ZW3D 2024 adds the function of virtual component being moved to other sub-assembly, which improves the ability to manage virtual components and makes the users easier to adjust the assembly level of virtual components after assembly.

- 1) Support common virtual components being moved to sub-assembly and virtual components created by assembly features (cam feature, gear feature) cannot be moved.
- 2) Virtual component can be moved to sub-assembly: This is equivalent to deleting the original virtual components and creating an identical virtual component in the target sub-assembly. All

the original associations are removed.

- 3) Virtual component copy to sub-assembly: This is equivalent to creating an identical virtual component in the target sub-assembly, but it has no relation with the original virtual component. All the original associations are removed.

Before moving	After moving

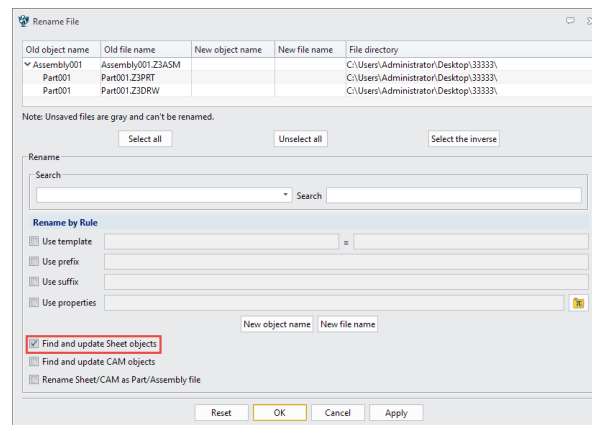
- 4) After virtual component is moved to sub-assembly, all the original association relations are removed.

→ Where is it

Assembly >> Assembly Manager

3.6.7 Rename File

The option “Find and update Sheet objects” is checked by default in Rename File command. In a real scenario, in-use drawing sheet is associated with 3D model. Almost all drawing sheets will need to update along with 3D models.

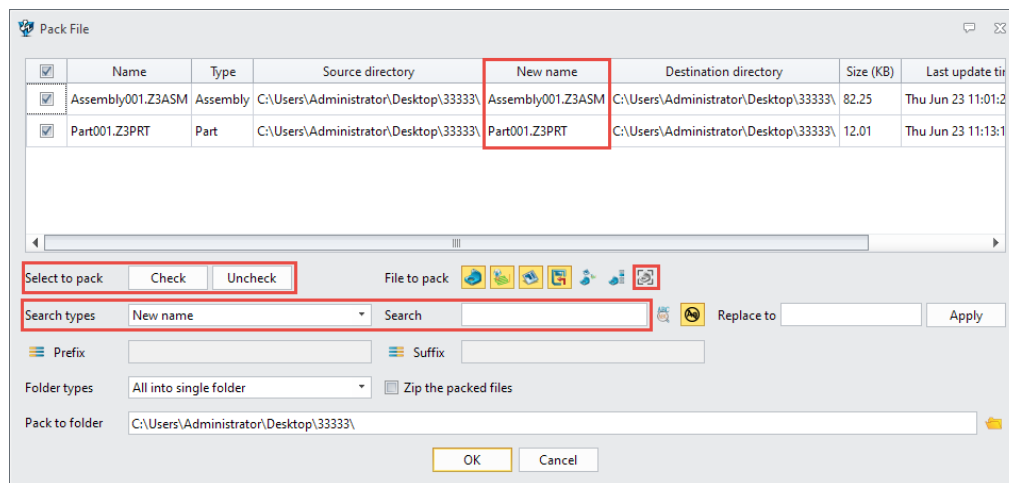


→ Where it is

3.6.8 New Search and Replace to in Pack File

UI improvements of Pack File mainly include:

- New UI layout and simplified Pack File interface.
- New selection type “unplaced components”.
- New support in user-defined name.
- New search function including exact search and fuzzy search.
- Search object supports checking or unchecking file browsing and batch text replacing.

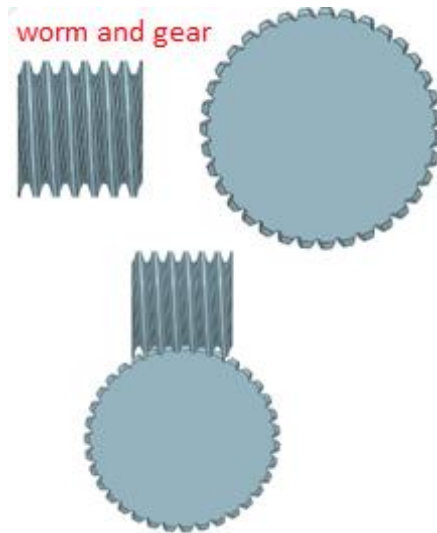


→ Where it is

File >> Pack

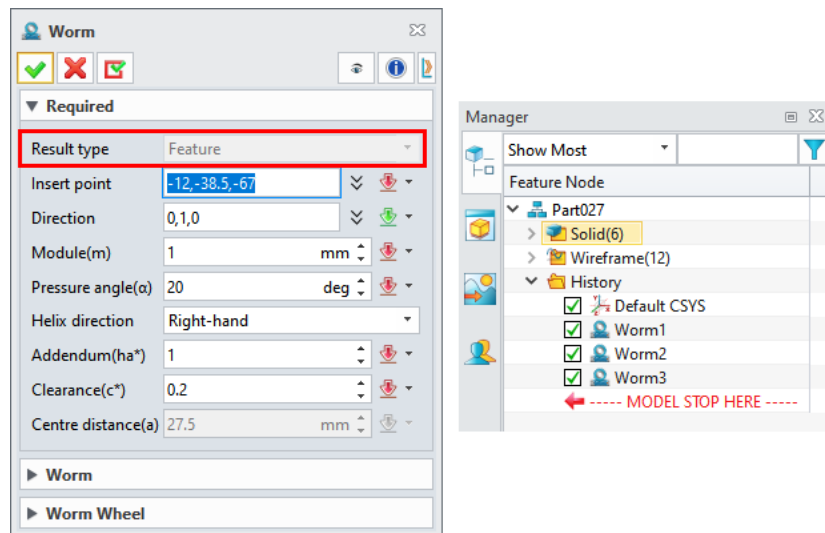
3.6.9 New Worm

ZW3D 2024 adds new worm and gear library, which can quickly generate worm and gear models including Archimedes cylindrical worm (ZA) and hollow flank worm (ZC) and worm gear mechanism.

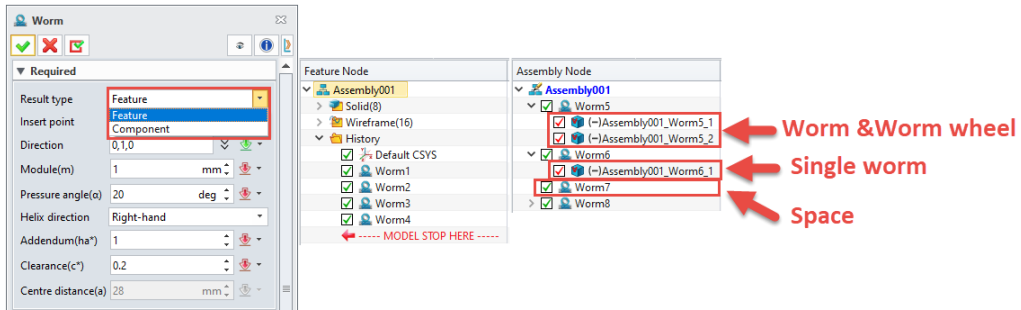


The supported methods to generate features and components:

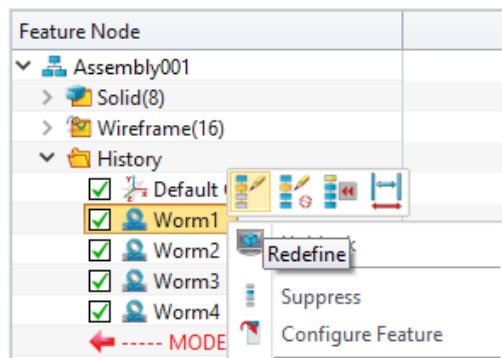
- 1) In the part environment, generate worm and gear features by default. An empty gear feature is not supported to generate on the history tree record feature node.



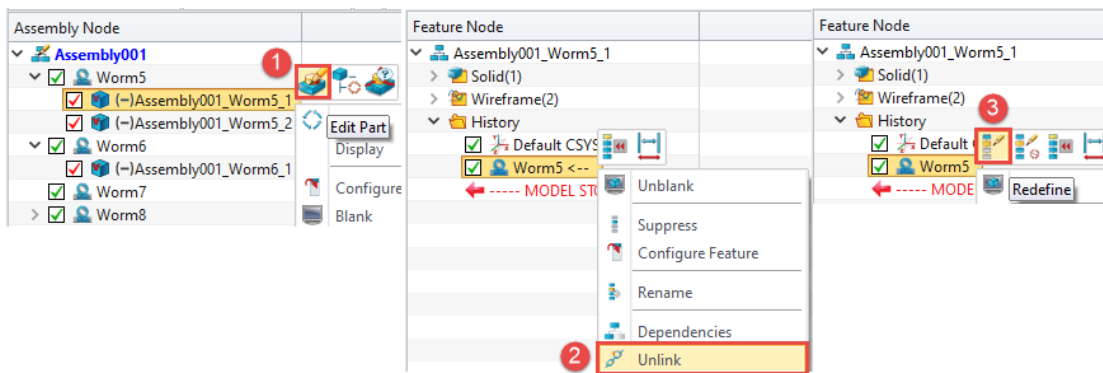
- 2) In the assembly environment, support two methods to generate features and components. A feature is recorded on the assembly history tree as an assembly node, and a component is recorded on the assembly history tree as a part. When generating a component, a worm or a gear is placed in a part. When a pair of worm gear is generated, its location meets the meshing condition.



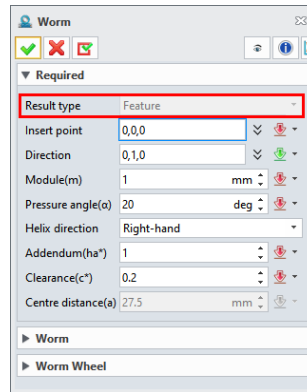
- 3) Under the feature node, you can double click or right click a worm gear feature to redefine the feature.



- 4) The worm gear can be redefined only after double-clicking under the assembly node to enter the part environment and disassociate under the feature node of the part.



- 5) When the worm gear is generated, the same interface will pop-up when creating it, and the gear is not supported for editing.



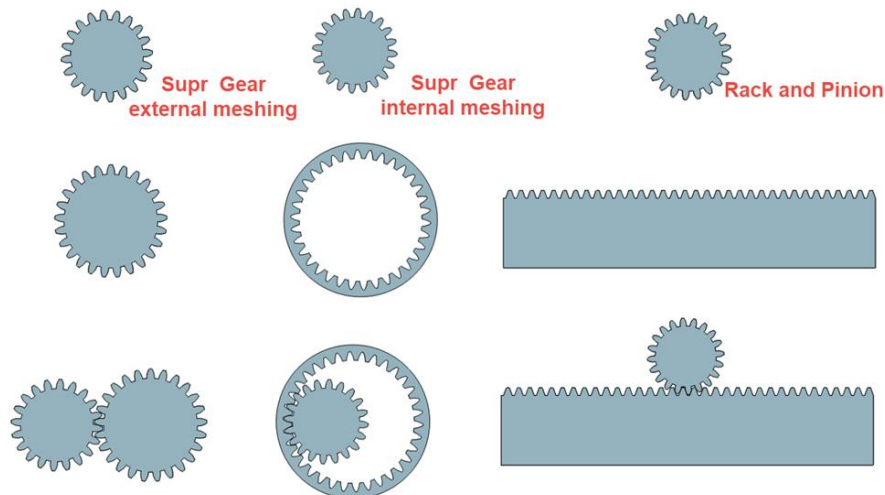
→ Where is it

Part/Assembly >> Library >> Worm

3.6.10 ★ New Spur Gear

New gear standard library enables user to quickly generate various of gears including cylindrical external meshing gear, internal meshing gear, pinion rack, and bevel gear mechanism:

1. Cylindrical meshing gear mechanism of external meshing gear and internal meshing gear can create standard gear, equivarient gear and unequal variant gear. Rack and pinion meshing mechanism can create standard gear and equal gear.



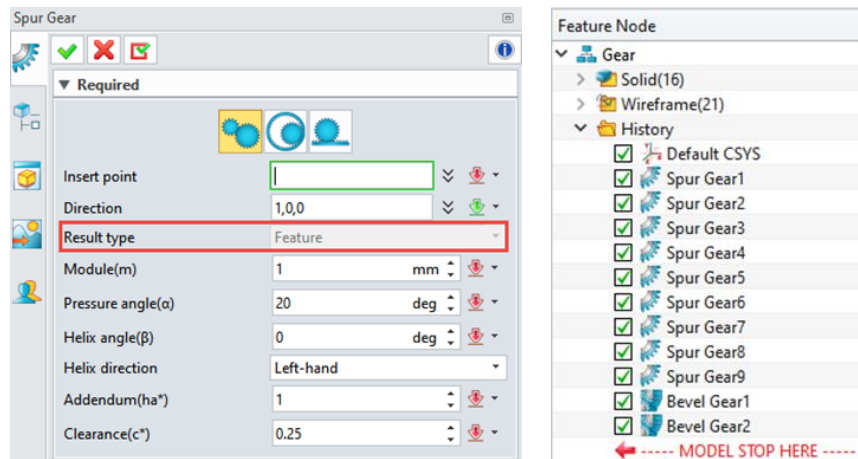
2. Bevel gear command can quickly generate bevel gears that support both spur and helical tooth shapes.



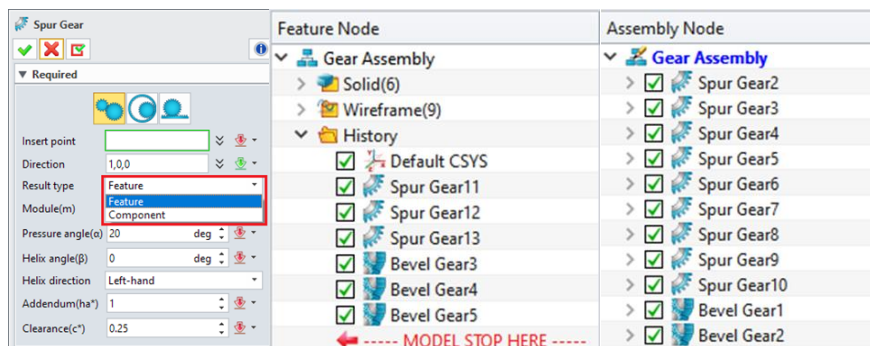
3. Support generating gear as features or components:

3.1 In the part environment, by default generate spur gear as features and record it in the history tree.

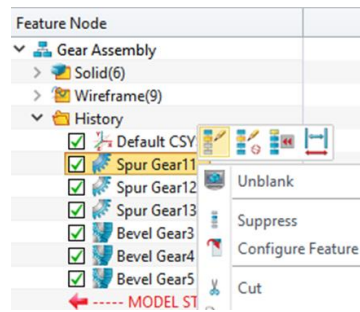
Do not support generating an empty spur gear feature.



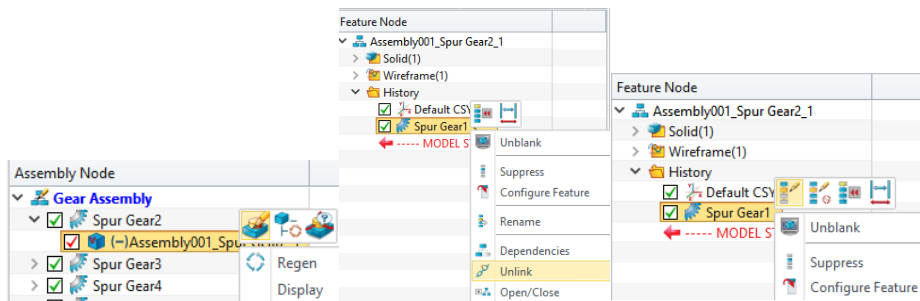
3.2 In the assembly environment, support generating both in features and components; features are recorded in the assembly history tree as assembly features; components are recorded in the assembly history tree as the form of part. When generating components, one spur gear stores in a part; when generating two spur gears, the position of the two spur gears meet the meshing conditions.



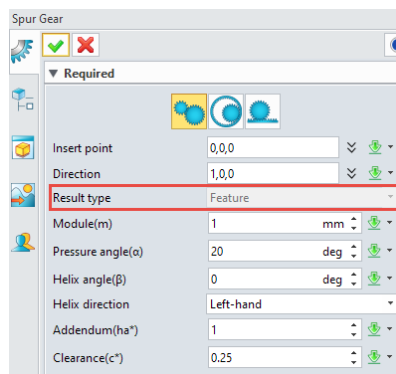
3.3 In the feature node, double-click or right-click spur gears features to redefine spur gears.



3.4 In assembly nodes, you must double click to enter the part environment and disassociate feature nodes in the part to redefine spur gears.



3.5 When editing spur gears, the same pop-up interface as creation does not support editing the generation form of spur gears.



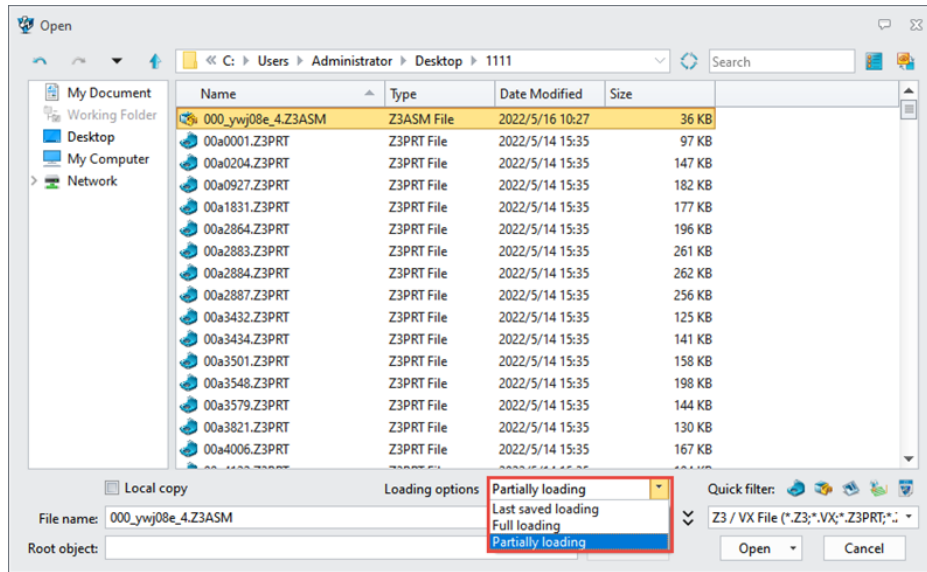
→ Where it is

Part/Assembly Environment >> Library >> Cylindrical Spur Gear

Part/Assembly Environment >> Library >> Bevel Gear

3.6.11 ★ Assembly Partially Loading

When users select an assembly file in Open interface, it supports to load the file in last saved loading, full loading and partially loading options.



Last saved loading: It's convenient to load file according to partially loading that set by user last time. If user had never opened and saved assembly file in partial loading mode before, it would open the current assembly file in full loading mode.

Full loading: Ignore partial loading states that are already saved and open the selected assembly file with ZW3D existing file open mode. With this mode, it will load the current structures, constraints, and associated relations of assembly parts to the memory excluding suppressed components and unplaced components. At this moment, the suppressed components and unplaced components are still not being loaded to the memory.

Partially loading: Only the information of parts and components selected by user on the partial loading interface can be loaded into the memory.

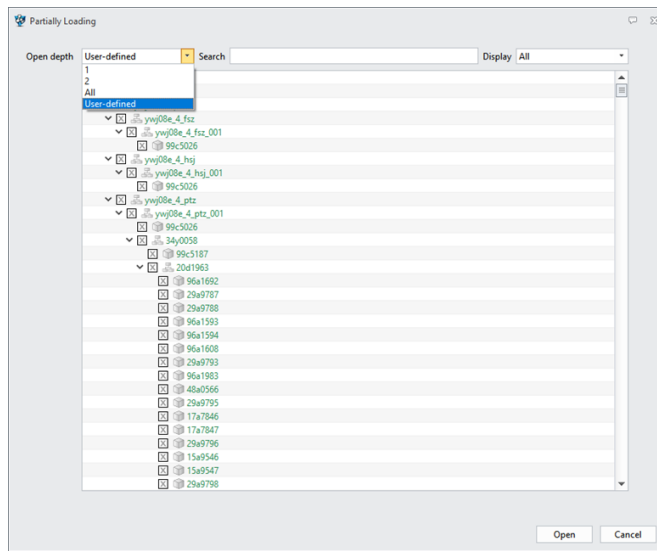
When user selects "Partially loading" and clicks Open command, pop up partially loading interface where user can perform the following operations:

- 1) Open depth column support user to select loading the first-level sub-assembly only, loading the second-level sub-assembly only, full loading and user-defined (default not loading any sub-components)

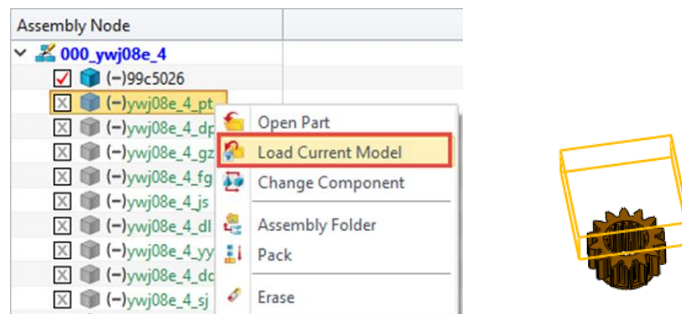
as well as support user to adjust the components' loading status by selecting checkbox before structure tree.

2) Support user to search related parts according to the part name to easily locate the part and specified the loading status.

3) Support user to switch the display style of the current page structure tree through "All" and "Only loading" option in the display bar.



When opening the assembly with the partial loading mode, user is supported to pick the unloaded component by right-clicking "Load current model" to load the component into the memory. When the mouse hovers on the unloaded components, it should display the component bounding in the drawing area.

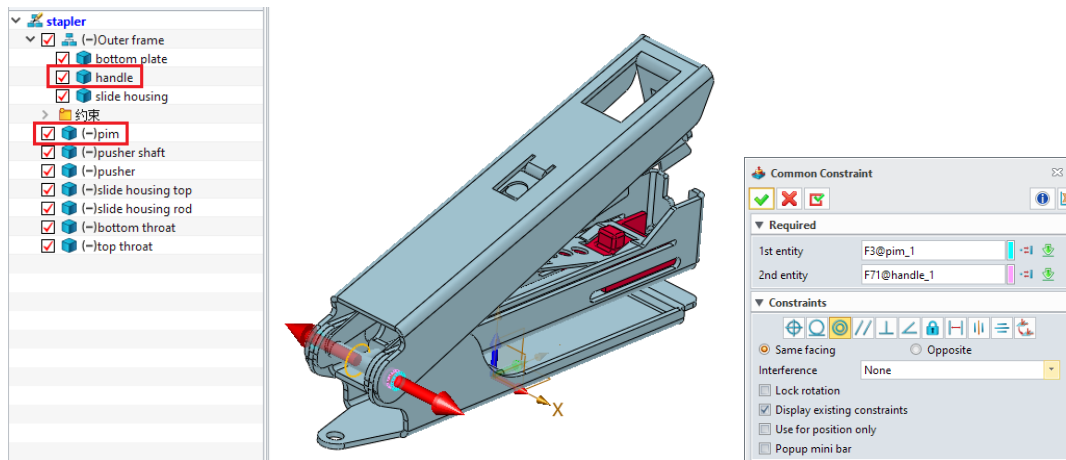


→ Where it is

File >> Open >> Loading Option

3.6.12 New Cross-level Assembly Constraint Picking Function

For assembly design, sometimes the assembly tree structure of design flow does not completely conform to the assembly tree structure in technological process, which leads to crossing level picking on other components in non-current assembly as constraint reference object. To improve the efficiency of creating constraint in such scenario, ZW3D added the function of cross-level assembly constraint object picking function.

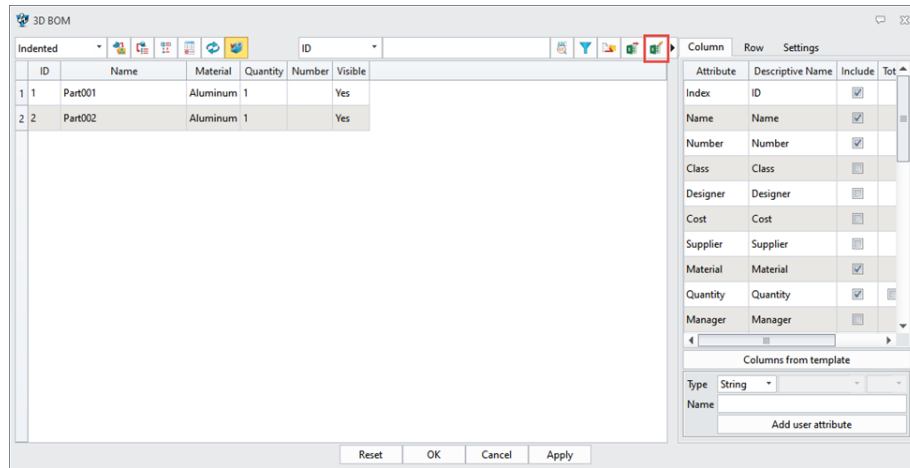


→ Where it is

Configuration >> Part >> Cross-level assembly constraint

3.6.13 New “Edit in excel” in “3D BOM”

Sometimes user may need to use data processing tool or import Excel file to edit part attributes. ZW3D supports editing part attributes in Excel as shown below. The function expands the application range of 3D BOM and enhances the data processing ability in part attributes.



	A	B	C	D	E	F
1	ID	Name	Material	Quantity	Number	Visible
2	1	Part001	Aluminum	1		Yes
3	2	Part002	Aluminum	1		Yes
4						

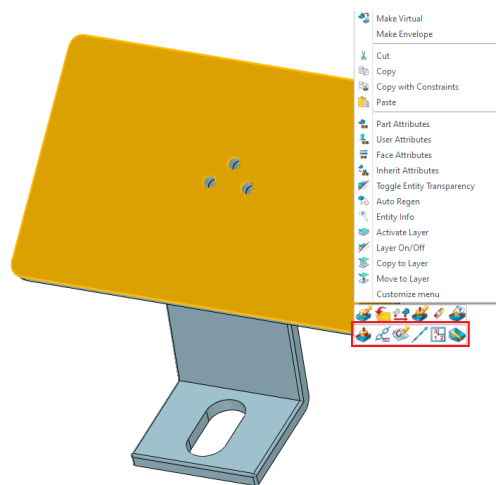
→ Where it is

Assembly >> 3D BOM >> Edit in excel

3.6.14 ★Support Part Design Behaviors in Assembly Environment

ZW3D supports part design behaviors in the assembly environment. It supports directly selecting geometric objects of component in the assembly environment (such as points, edges, surfaces, bodies, etc.) and features performing by right-clicking menu.

When selecting component geometries, it will pop up context menu by right clicking the mouse. This menu adds common geometry operations (such as creating constraints, creating reference, assembly fillets, and assembly holes) in the assembly environment, and add several modification operations regarding to the design history (such as redefining, extruding and rotating), in addition to functionalities for the component itself in previous version. If you select a command to modify design history of a component itself, the system will automatically switch the scenario between part environment and assembly environment.



New function menu may vary according to the different selected geometric types. The specific difference goes as below:

Element type	Display menu	Element type	Display menu
Sketch		Plane	
Shape		Surface	
Edge		Curve	
Point		Text (text, balloon text)	
Datum plane		Datum axis	
Datum CSYS			

3.6.15 ★Support In-place Editing Component Properties on Assembly Tree

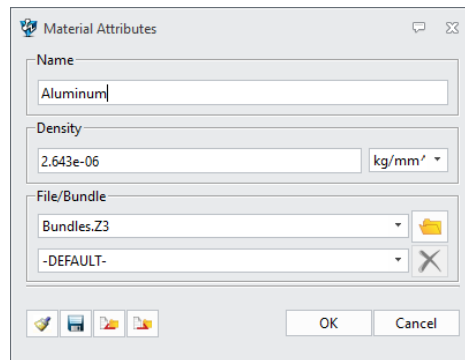
In the assembly environment, you can directly select part properties to perform in-place editing on the assembly tree. The supported properties include standard properties, user properties and physical

properties such as material, density, stock.

Double-click the properties on the assembly tree directly to edit the standard properties.

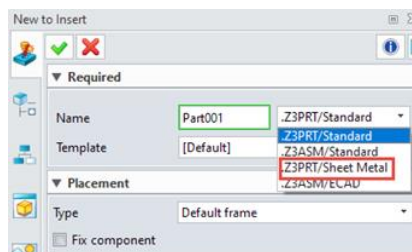
Assembly Node	Designer	Manager	Supplier
Assembly001			
Part001	ZWsoft1	ZWsoft2	ZWsoft3
Part002			
Constraints			

Double-click the properties on the assembly tree to open the corresponding property panel for editing user, physical properties such as material, density, stock size.



3.6.16 New File Type in “New to Insert” Command

We added “Sheet Metal” to part type to meet user’s need of inserting sheet metal when inserting a new part with “Sheet Metal” type.

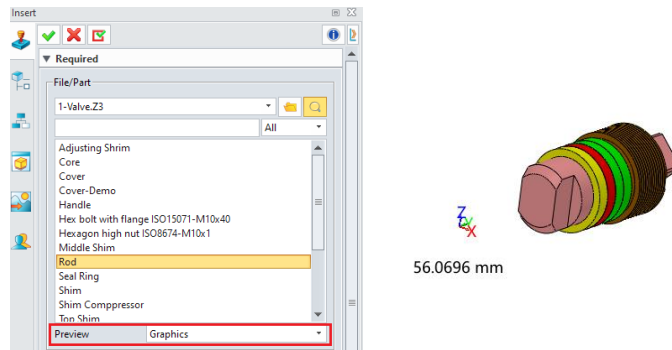


→ Where it is

Assembly Environment >>New to Insert >> Name

3.6.17 Optimized Function of Inserting in multi-object File

We added a preview function to the multi-object assembly file insert option, as shown in the following figure



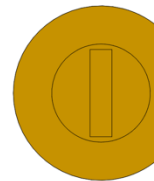
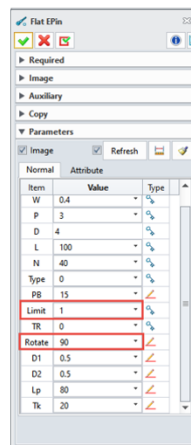
→ Where it is

Assembly Environment >> Insert >> Preview

3.7 Mold Design

3.7.1 Flat EPin Improvement

We optimized the alignment method of the flat EPin with headcutting: after removal, the limit edge will be aligned with the head of flat EPin; when rotation value is set, the limit edges will change along with the head of flat EPin.

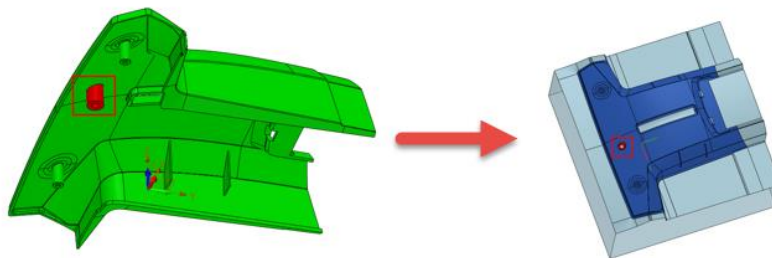


→ Where it is

Mold Environment >> Library >> Flat EPin

3.7.2 ★ Support Parting Update after Product Model Changed

After parting product model, if any change occurred to the product model, ZW3D supports updating relative core and cavity model. But noted that the change cannot involve parting faces or generate inner ring (hole). As the figure shown below, we added a cylinder to the product model and then update all components on the top level of assembly. At this point, the relative component models will be automatically updated.

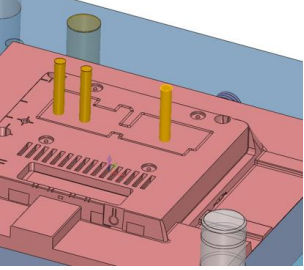
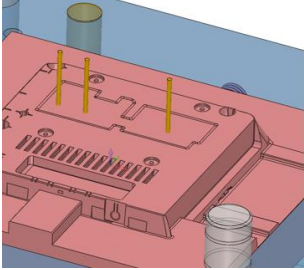
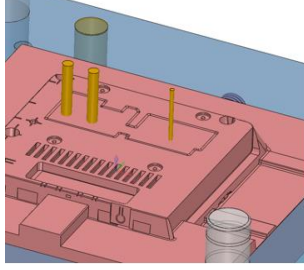


→ Where it is

Mold Environment >> Parting Design

3.7.3 “Modify” Command Optimization

We add a “Modified as new components” option to “Modify” command. (1) When the option is unchecked, the command will directly modify the standard part parameters according to the new user-defined parameters and regenerate the standard part. (2) When the option is checked, the command will generate new components according to the new user-defined parameter and replace the original components (only replace user-selected components). If there were components with the same names, add “_1”, “_2” etc. at the end of the name.

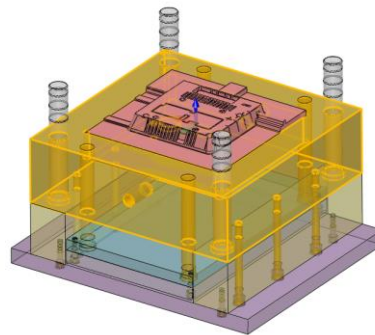
The original assembly	Didn't modify as new components	Modified as new components
		

→ Where it is

[Assembly Environment](#) >> [Mold Environment](#) >> [Library](#) >> [Modify](#)

3.7.4 ★ “MoldBase Modify” Command Optimization

In previous versions it would renew a mold base when using “MoldBase Modify” command to redefine a mold base. In new version, the command will read the configuration parameters of the current mold base file and display them in “Modify MoldBase” command interface when redefining picked mold bases. When users modify relative parameters and confirm, the command directly rewrites the expression values in the mold base assembly file and regenerates the mold base. As the updating is performed on the original model, the created features can be preserved. As shown in the below figure, redefining this mold base with the thickness of Plate B from 130mm to 150mm, the created Groove feature is preserved.

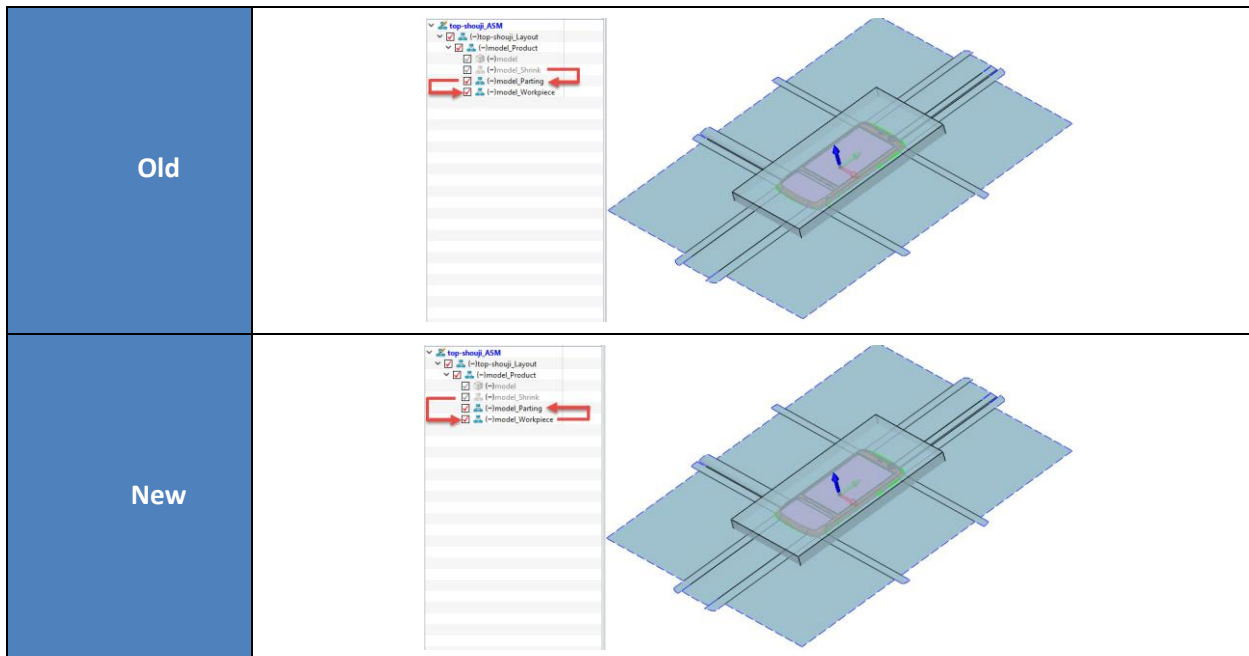


→ Where it is

[Mold Environment](#) >> [Library](#) >> [MoldBase Modify](#)

3.7.5 Workpiece Model Creation Node Adjustment

To make the process clearer and improve the stability of regeneration, we adjusted the creation nodes of workpiece model as shown in the below figure. According to the model in the Shrink component, a workpiece model is created at first in the Workpiece component, then its copy is sent to Parting to avoid workpiece model influenced by Parting component, improving the stability of regeneration.

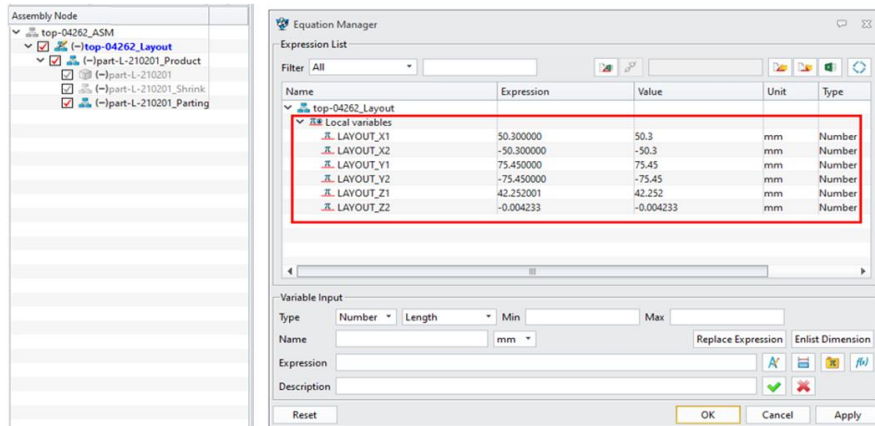


→ Where it is

[Mold Environment >> Parting Design >> Workpiece](#)

3.7.6 Adjust Layout Parameter Generation Time

ZW3D optimized generation timing of the Layout parameter. When product model is loaded, it will generate relative Layout parameters in Layout components. At this moment, the following two cases will cause update in Layout parameter. (1) When using “Workpiece” command to create workpieces, Layout command will envelope the workpiece model and shrink model to recalculate x, y, and z values. (2) After rearranging the model with “Move” or “quick workpiece” function, use the “Centering” command and the Layout command will envelope all layout product models and recalculate x, y, and z values.

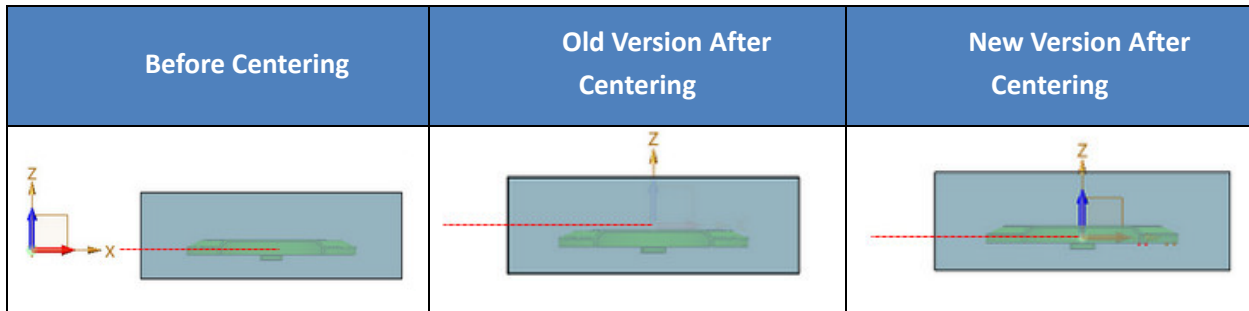


→ Where it is

Mold Environment >> Product >> Layout

3.7.7 Centering Optimization

During the product model layout, the “Centering” functionality is often used to reset the layout center to the coordinate origin. ZW3D optimized the logics of centering to ensure no offset occurring to the Z direction when using “Centering” command.



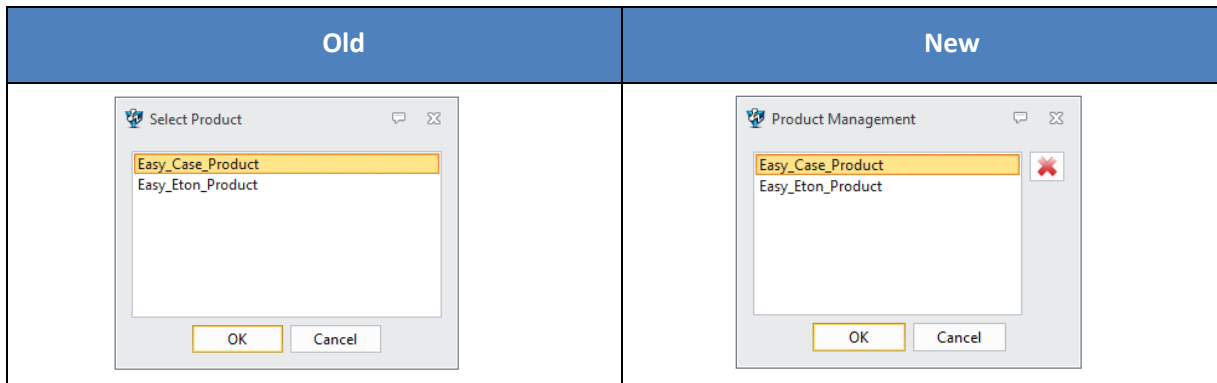
→ Where it is

Mold Environment >> Product >> Layout

3.7.8 Select Product Optimization

“Select Product” command had renamed as “Product Management” and added deletion to it. The

workflow goes as below right figure: select a product in the list and click the deletion button, the product will be removed from the list as well as its relative components and constraints in assembly tree.

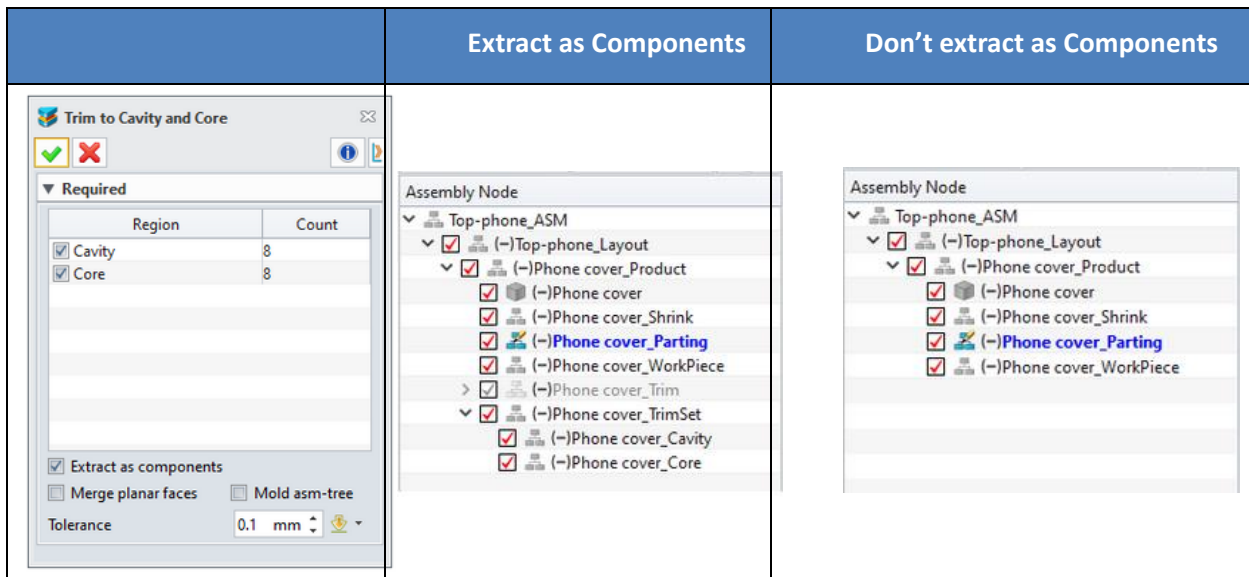


→ Where it is

Mold Environment >> Product >> Product Management

3.7.9 Trim Command Optimization

We added “Extract as component” option to Trim command. The option is checked by default. With this option, TrimSet components and their sub-components can be created after trimming.

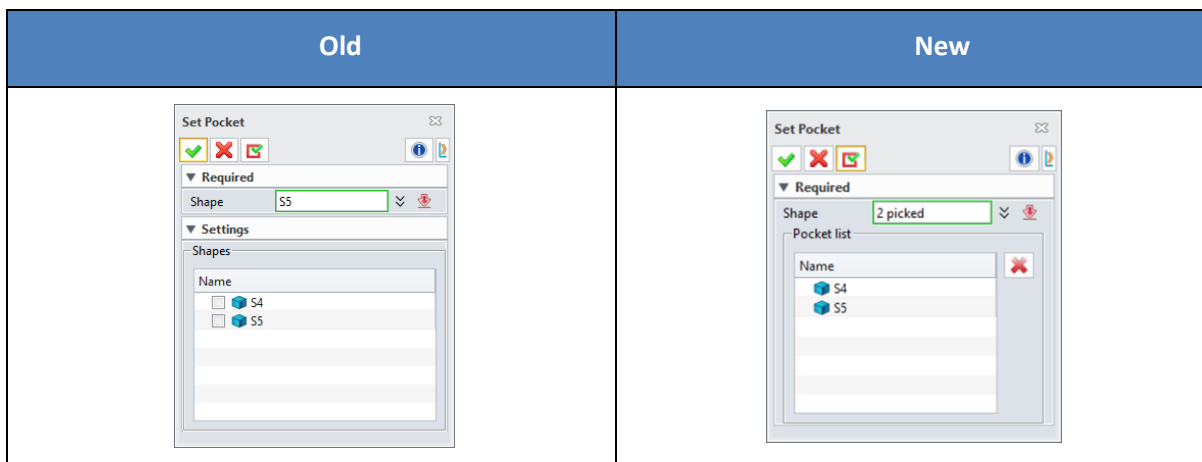


→ Where it is

Mold Environment >> Parting Design >> Trim

3.7.10 Set Pocket Command Improvement

To quickly define pocket name and solve the problem of one standard part corresponding to multiple pockets, we optimized the command in the following aspects: (1) Improve command interface; (2) Add removing button. When users pick a shape, it will be loaded into list automatically. You can remove the pocket shapes from the list by delete command. (3) For a shape defined as a pocket, the word "pocket" will be added after the name in the history tree.



→ Where it is

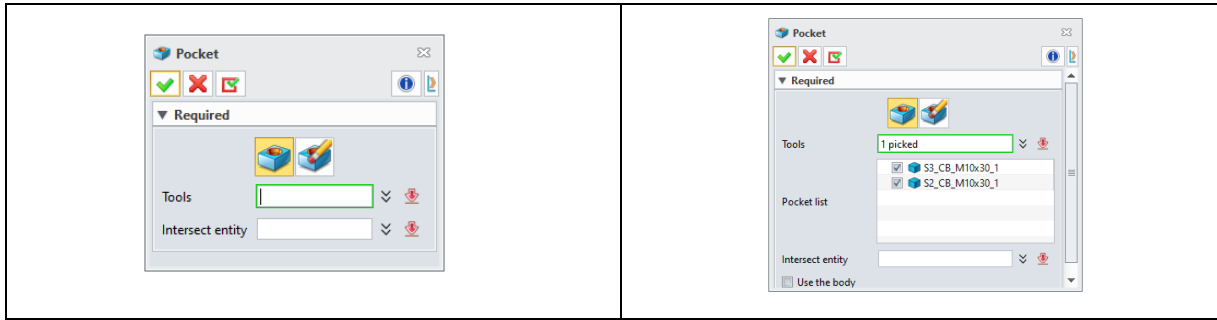
Part Environment >> Tools >> Library >> Set Pocket

3.7.11 Pocket Command Improvement

To improve the identification and accessibility in pocket, Pocket command optimized in following aspects:

- (1) Add pocket list next to the selection box. All pockets' shapes are listed out here, and the pocket name ends with the part name. For example, S3_CB_M10x30_1.
- (2) Add a checkbox before each pocket and check it by default. When checked, the pocket will be used to pocket the intersect entity.
- (3) Add "Use the body" function which is unchecked by default. When checked, it will use the tool body to pocket the target body.



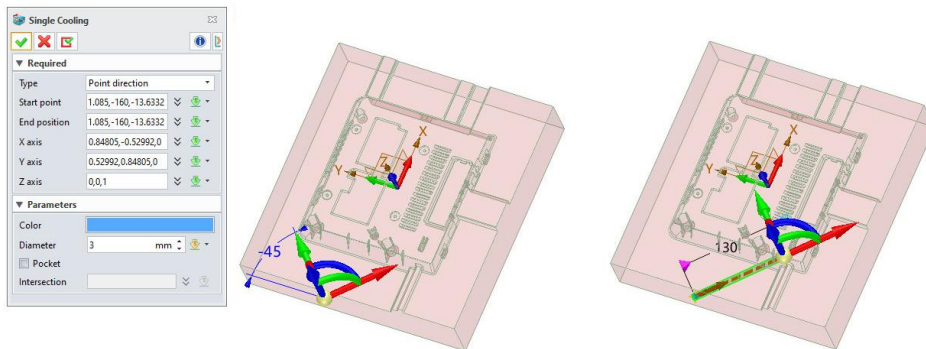


→ Where it is

Part Environment >> Library >> Pocket

3.7.12 New Point Direction to Create Single Cooling

We added a new method in Single Cooling command to meet the requirements of creating oblique waterway in mold design. As the below figure shown, start the command, rotate the coordinate system angle to confirm the direction, and then click the mouse and drag the coordinate to create a waterway.



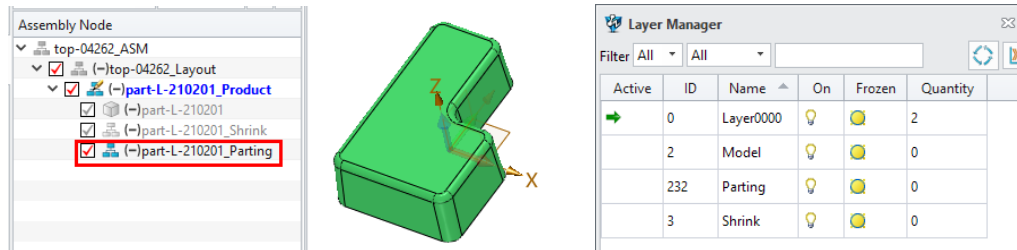
→ Where it is

Mold Environment >> Detail Design >> Single Cooling

3.7.13 Configured Layers to Mold Assembly Tree Components

To increase parting operation efficiency, after loading product model in mold project, the system will distribute configured layers to the original model components, the components with Shrink and the components with Parting. Only display the parting components by default and other two types of

components can be blanked by closing their layers. User can activate product component and manage these three types of components in layer manager.

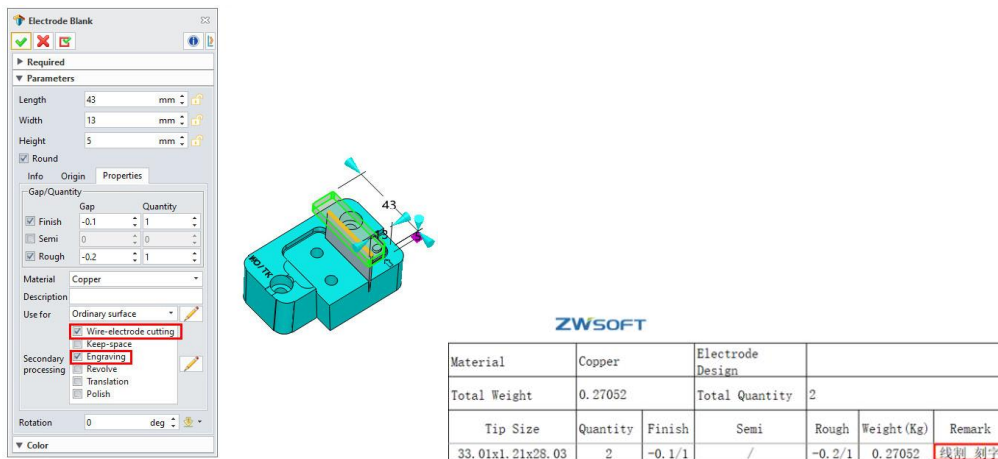


→ Where it is

Mold Environment >> Product

3.7.14 Electrode Blank Secondary Processing Properties in Bill of Electrodes

When creating an electrode blank, if corresponding secondary processing properties are checked, the corresponding properties will be displayed in the remark column inside the bill of electrodes.

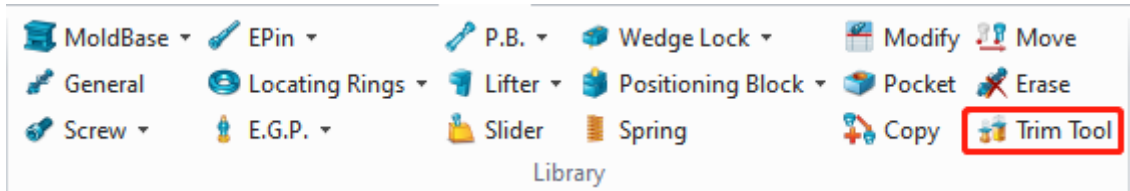


→ Where it is

Part/Assembly Environment >> Electrode >> Bill of Electrodes

3.7.15 ★New Trim Tool

New command Trim Tool includes three sub-commands: Trim epin, Tim component, and Delete trim.

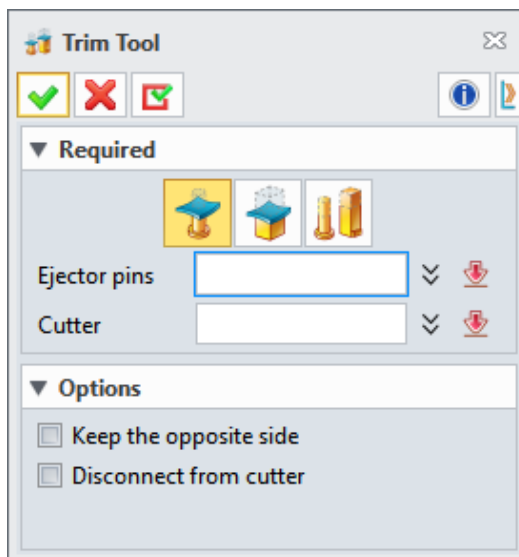


3.7.15.1 Trim Epin Improvement

Trim epin is merged to Trim Tool as a sub-command and added to “Keep the opposite site” and “Disconnect from cutter” options.

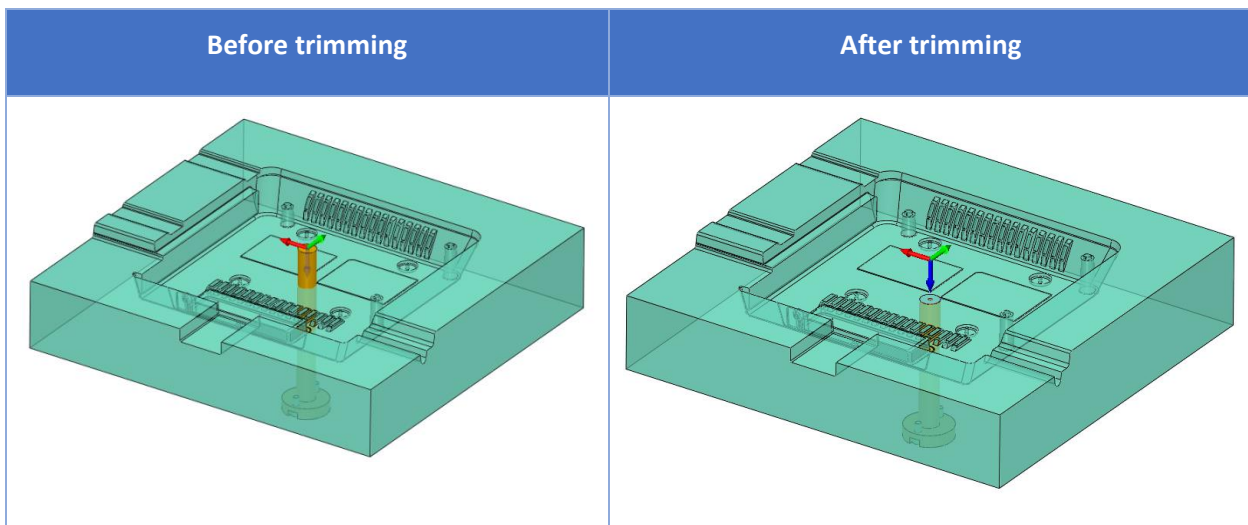
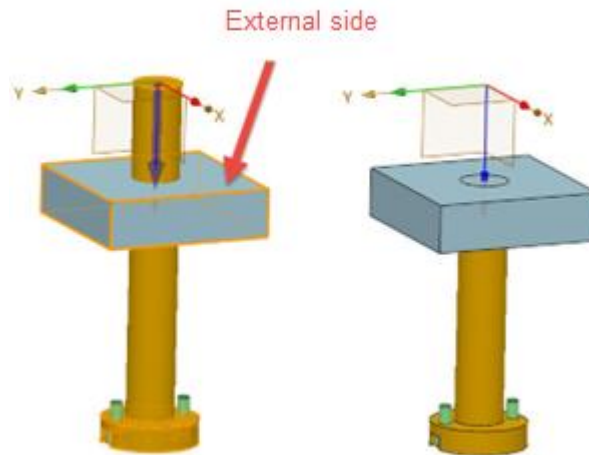
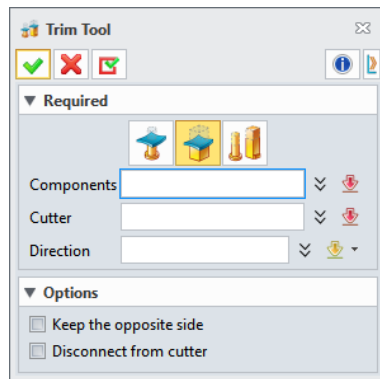
Keep the opposite side: uncheck it by default, after trimming, the cutting body is treated as boundary and keeps shape in -Z direction; check it, keep shape in +Z direction.

Disconnect from cutter: after trimming, disconnect the reference relation between an epin and a cutter.



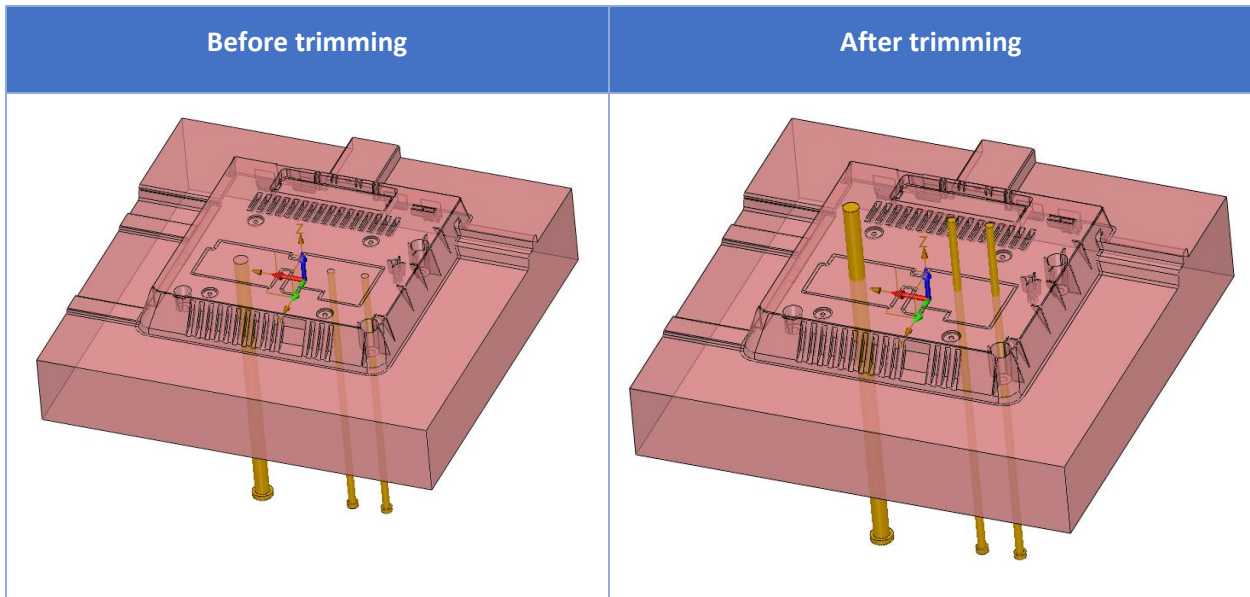
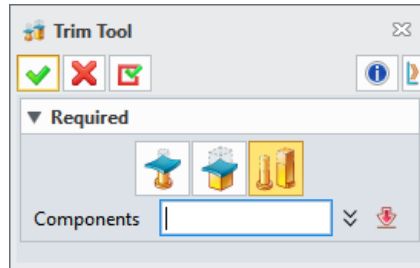
3.7.15.2 New Trim Component

To quickly trim mold components other than epin (e.g., insert, lifter, and gate, etc.), a new command “Trim component” has been added. Same as Trim Epin command, you can select components as target components for directly trimming. The functions of “Keep the opposite side” and “Disconnect from cutter” are the same as Trim Epin command. Add a direction function that specifies the kept direction of target component after it is trimmed. As the following right picture shows, use a block as cutter to trim a gate and select +Z as direction. After trimming, take the block’s external side as boundary, and keep shape +Z direction.



3.7.15.3 New Delete Trim

Add “Delete Trim” command, which supports quickly recover the trimmed epic or component to its pre-trimmed appearance.

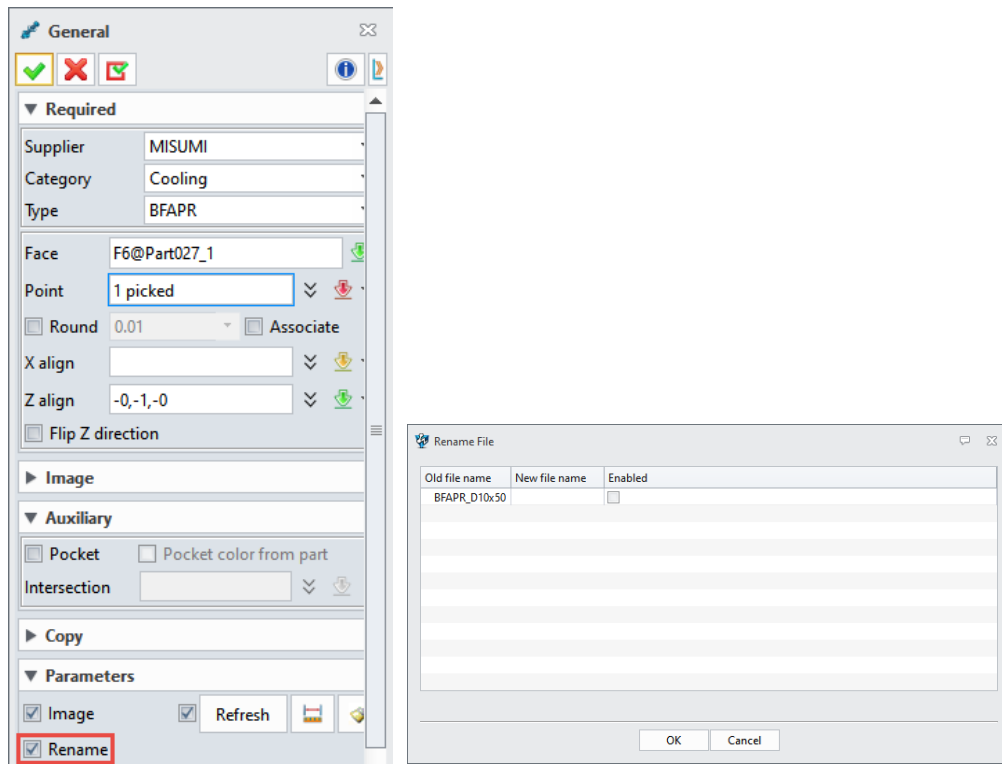


→ Where is it

Mold >> Library >> Trim Tool

3.7.16 General Rename

Add “Rename” checkbox in the General interface. Uncheck it by default. If check it, pop-up a “Rename File” interface where user can rename the stand part to be created.



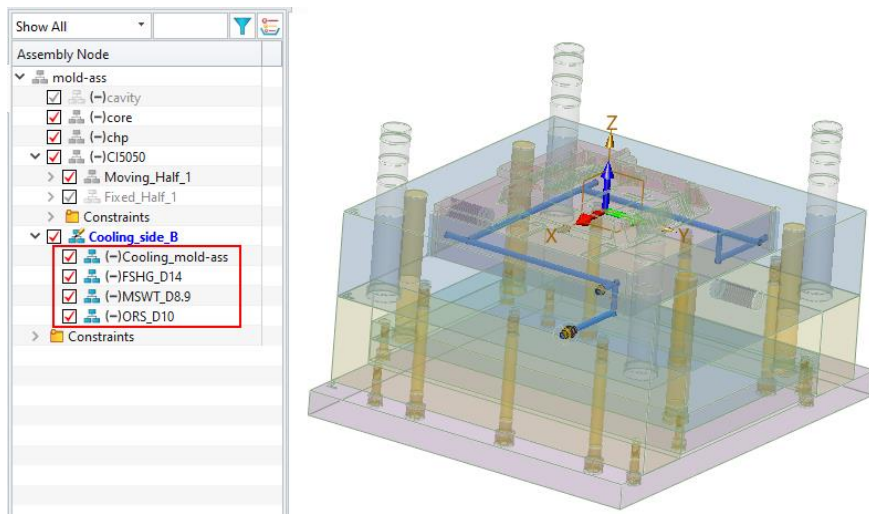
→ Where is it

Mold >> Library >> General/Other Specified Commands

3.7.17 Insert Cooling Component In-place Editing

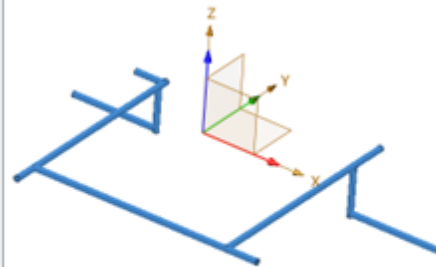
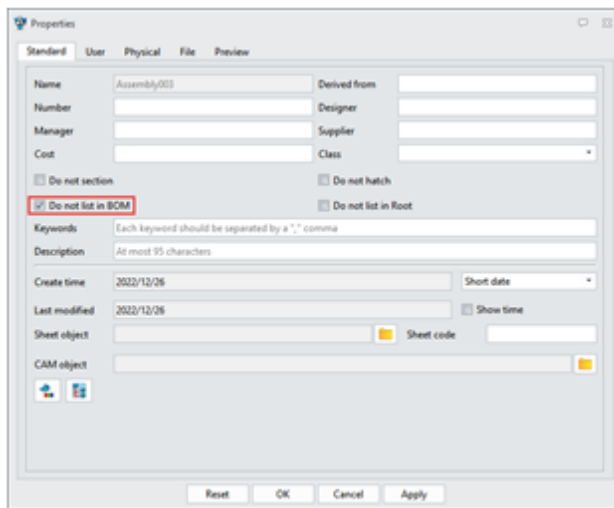
3.7.17.1 Create Cooling Standards in Activating Node

When the assembly node under in-place editing status utilizes the cooling component related commands (Cooling, Single Cooling, Well, Pipe Joint, Pipe Ring), support picking up surfaces and points on other non-active components and creating the corresponding cooling components which are placed under active assembly nodes by default.



3.7.17.2 Cooling Feature Component Set “Do not list in BOM”

The files created by cooling, single cooling, and well can only be used in mold pocket. Thus, they do not need to list in BOM. These standard commands are optimized. For the standard parts created by these commands, check the option “Do not list in BOM” by default in Properties to ensure that they are not displayed in BOM.

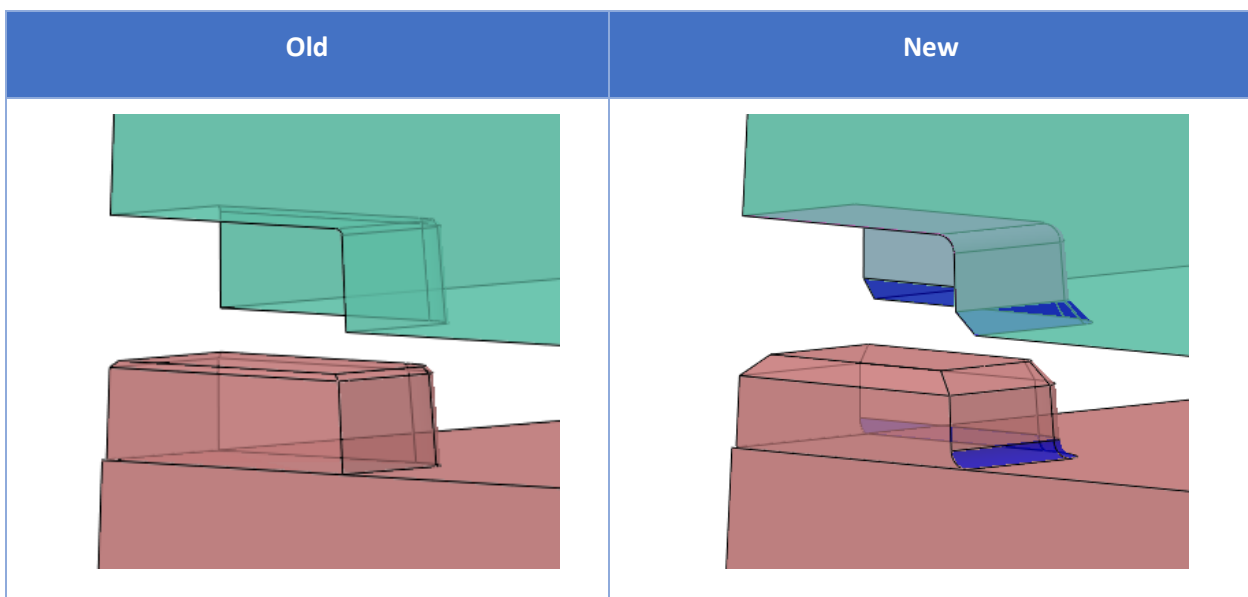
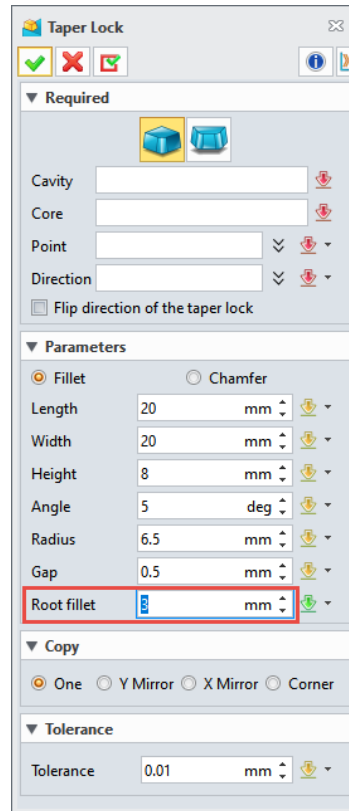


→ Where is it

Assembly >> Tools >> Properties >> Standard

3.7.18 Taper Lock Improvement

The function of root fillet is added to Taper Lock. When a taper lock is created, you can add fillet to boss and chamfer to groove. (The value of chamfer value is 0.5mm larger than the fillet).

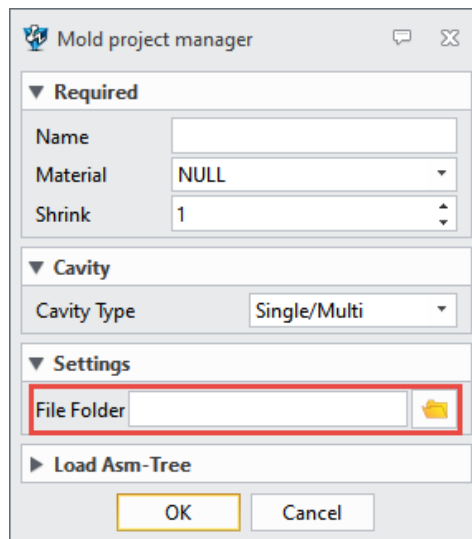


→ Where is it

Mold >> Detail Design >> Taper Lock

3.7.19 Mold Project Default Created Path Improvement

When creating a mold project, instead of creating a new folder in the selected path of the command interface, save the project related files directly to the selected path, as the following figure shows:



Old	New
<ul style="list-style-type: none"> 📁 top-0919 🔧 cover2 .Z3PRT 	<ul style="list-style-type: none"> 🔧 cover3 .Z3PRT 🔧 top_0919_ASM.Z3ASM 🔧 top_0919_Layout.Z3ASM

The improvements also include:

- 1) In the premise of opening product file, you save a project directly to the folder that the current file locates when creating a project.
- 2) In the case of first creating project, the files related to the mold project will be created directly in the folder selected by the project.

3) When loading a product file, default open the selected path when creating the project.

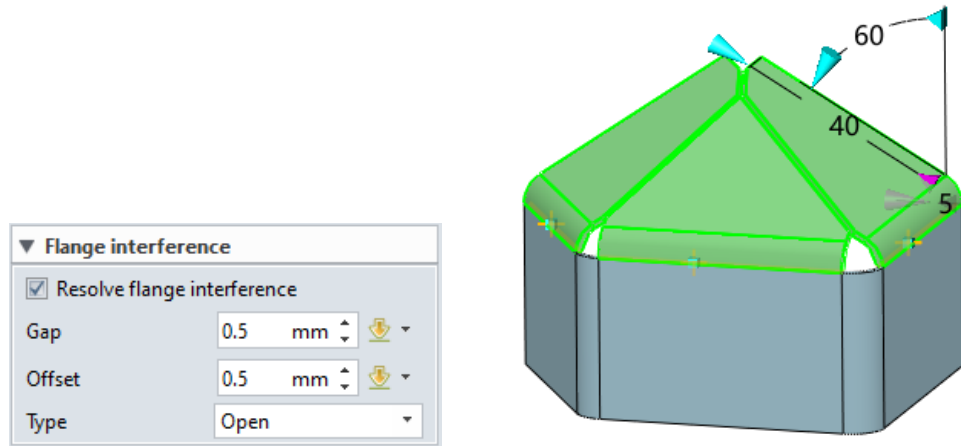
→ Where is it

Mold >> Product >> Mold Project

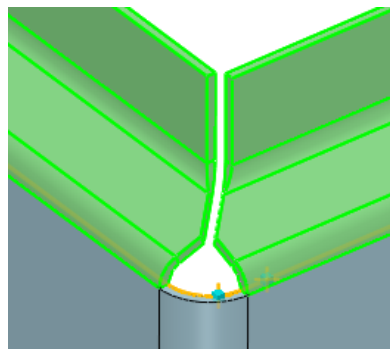
3.8 Sheet Metal Design

3.8.1 ★New Miter Cut Functionality

Sheet metal flanges can correctly eliminate the self-intersection between multiple flanges. User can adjust miter cut by modifying “Gap” and “Offset”.



The miter cut supports complicated geometries such as S bend.

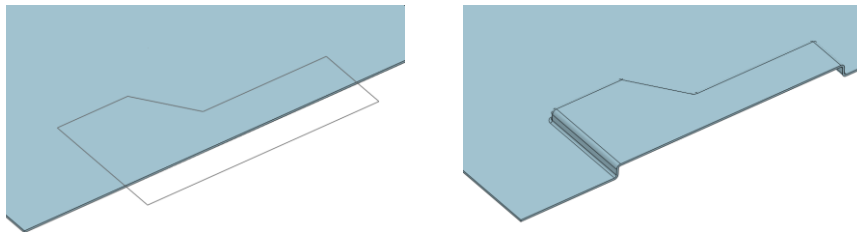


→ Where it is

Sheet Metal Environment >> Flange >> Full Flange >> Flange interference

3.8.2 Enhanced Dimple Supports Complex Sketch

Flange dimple supports more complicated sketch and efficiently generates more complicated dimple features to provide better user experience in flange part modelling.

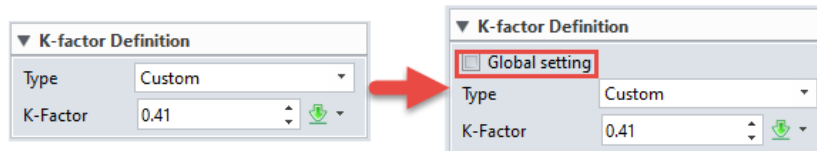


→ Where it is

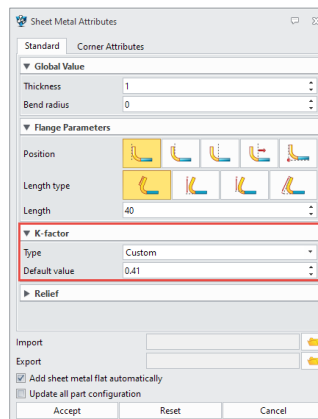
Sheet Metal Environment >> Form >> Dimple

3.8.3 K-factor Setting Improvement

All “K-factor Definition” in flange has been adjusted to global setting. When user checks “Global setting”, the flange command is applied to global K-factor setting and keeps the same type and value of K-factor as set in the sheet metal attributes.



Notes: Global K-factor can be set in the sheet metal attributes.

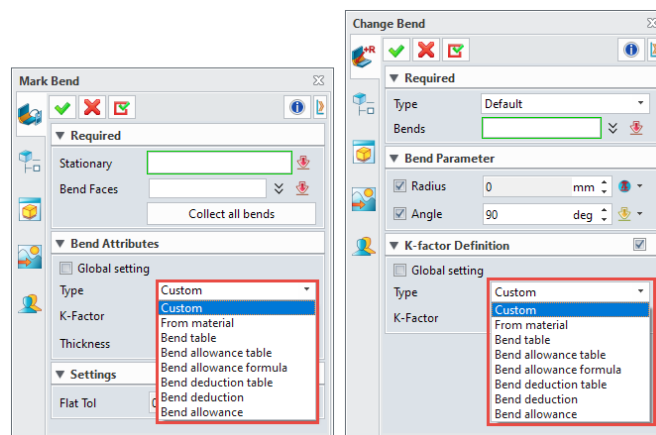


→ Where it is

Sheet Metal Environment >> Flange/Bend/Convert >> K-factor Definition

1.1.1 Mark Bend/Change Bend Improvement

The Bend Attributes of Mark Bend and K-factor Definition of Change Bend have been uniformly adjusted into Custom, From material, Bend table, Bend allowance table, Bend allowance formula, Bend deduction table, Bend deduction and Bend allowance, keeping same as that in K-factor setting types of each flange command.



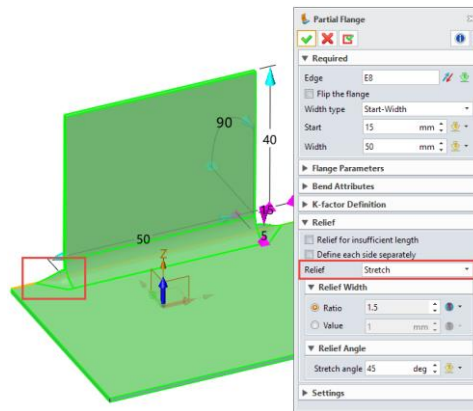
→ Where it is

Sheet Metal Environment >> Convert >> Mark Bend

Sheet Metal Environment >> Bend >> Change Bend

3.8.4 New Relief Type in Flange Sketch

In the electrical industry, to reduce processing steps in partial flange bend, the relief does not apply with the blanking technology but with the punching at the same process of bending. ZW3D added the relief of sheet metal stretch type to support such scenario.



→ Where it is

Sheet Metal Environment >> Flange >> Full Flange / Flange with Profile / Partial Flange / Hem Flange >> Relief

3.8.5 Flange Command Integration Improvement

1. Support all bend types.

New flange command combines bend types in other flanges in the system and support user using sketch to self-define. In the design process, user can modify the created flange features, which can quickly switch bend type and raise work efficiency.

Support bend types include Simple, S bend, Closed, Open, Joggle, Curl, Z bend (S bend in ham flange), Closed loop, Open loop, Centered loop, and user defined.

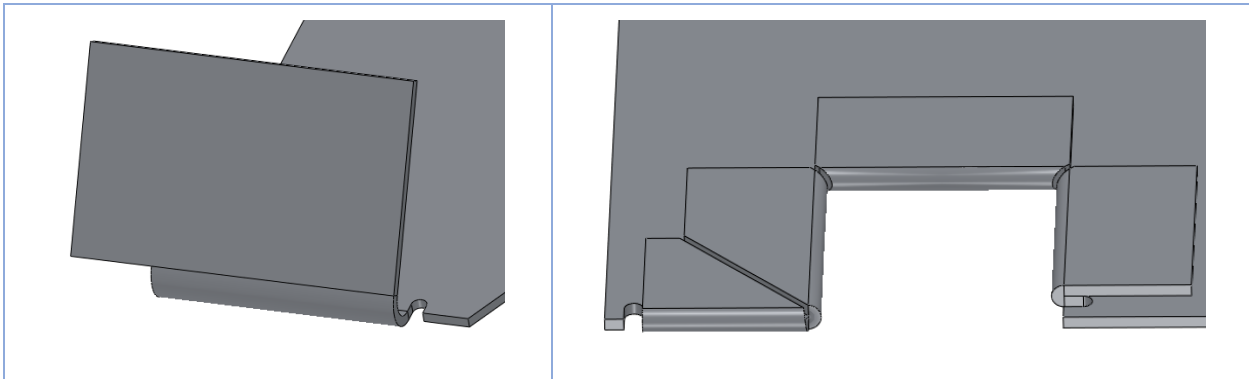
2. Partial flange upgrade

New flange command has merged the function of Partial Flange with improvements including support for inputting G0 continuous edge chain and creating extending flange that exceeds the edge chain.

Support inputting G0 continuous edge chain and directly eliminate the interference of non G1 continuity, and do not need extra processing.

Extend flange

Eliminate G1 continuity interference

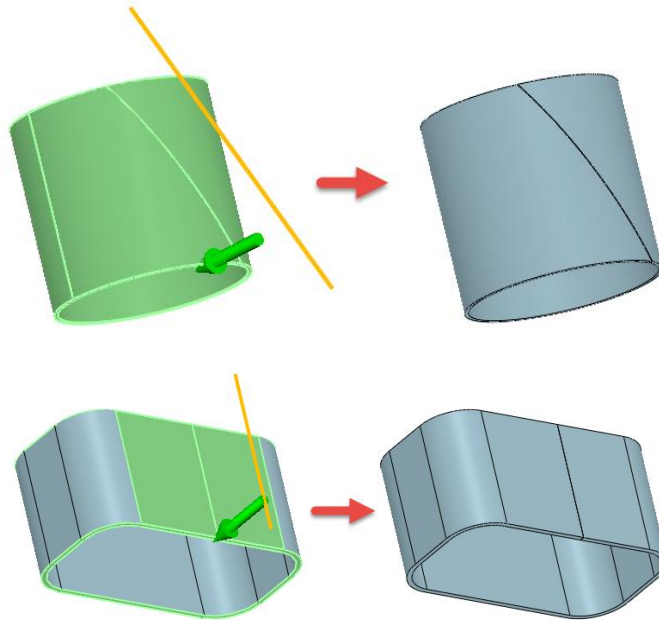


→ Where is it

Part/Assembly >> Sheet Metal >> Flange

3.8.6 ★New Sketched Rip

ZW3D 2024 adds the function of Sketched Rip which supports ripping the sheet metal part with 2D sketch. This function can generate a rip to sheet metal shape effectively.

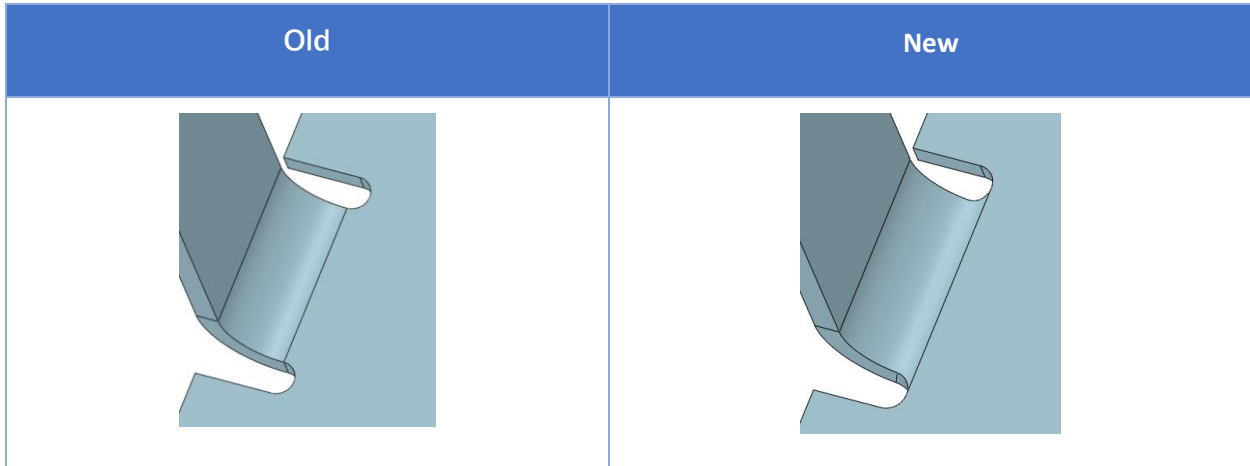


→ Where is it

Sheet Metal >> Convert >> Sketched Rip

3.8.7 Relief Support Bend Alignment

Add “Up to bend” type of the obround relief slot, which supports aligning obround relief slot with bend.



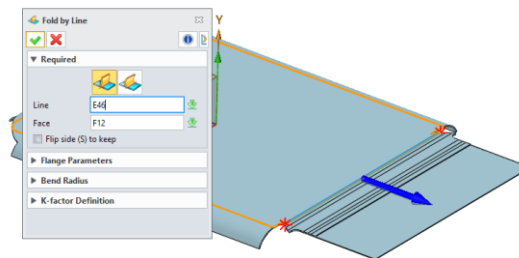
→ Where is it

Part/Assembly >> Sheet Metal >> Flange >> Relief >> Relief Length >>. Up to bend

Part/Assembly >> Sheet Metal >> Flange with Profile >> Relief >> Relief Length >>. Up to bend

3.8.8 Fold by Line

To increase the design efficiency of folding by line, ZW3D 2024 supports directly selecting edge as the fold line.



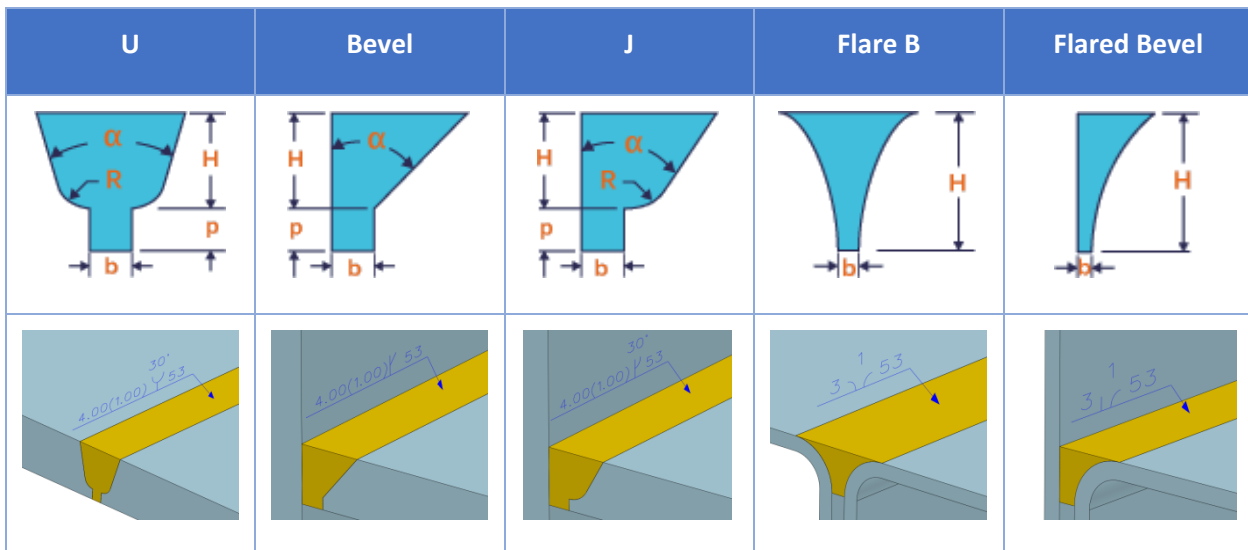
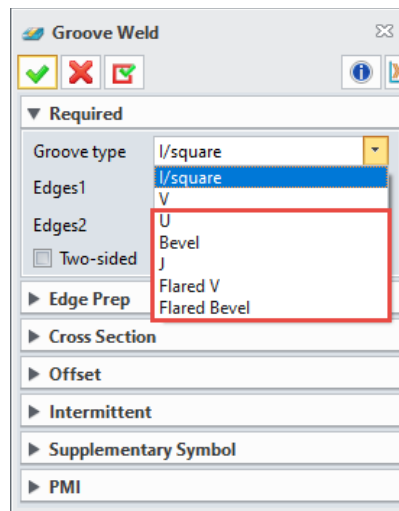
→ Where is it

Part/Assembly >> Sheet Metal >> Flange >> Fold by Line

3.9 Weldments

3.9.1 New 5 Groove Weld Types

ZW3D 2024 add 5 new groove types: V, U, Bevel, J, Flared B, and Flared Bevel. In the process of weldment design, you can create different groove weld types and create to generate surface weld, light weld, and solid weld and its corresponding PMI dimension which can clearly deliver designer's processing requirements to weldment.

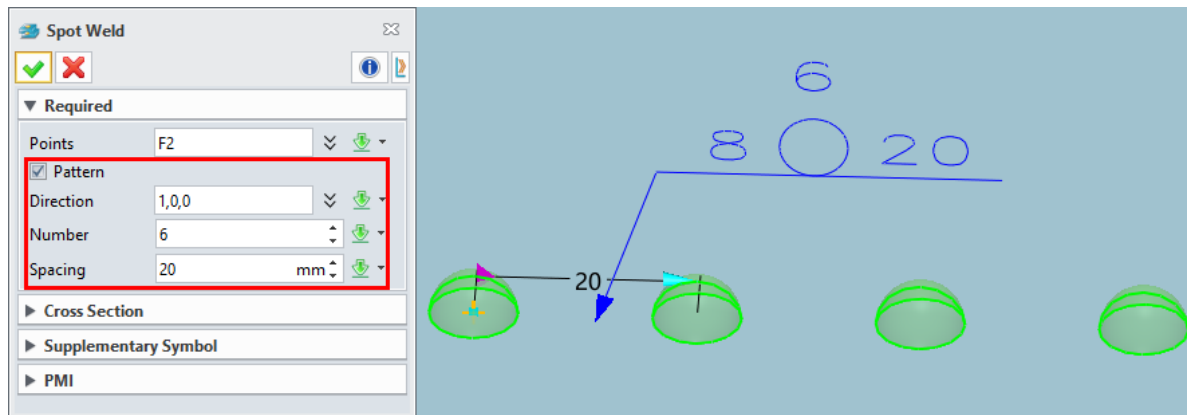


→ Where is it

Part/Assembly >> Weldments >> Frame >> Groove Weld

3.9.2 New Pattern in Spot Weld

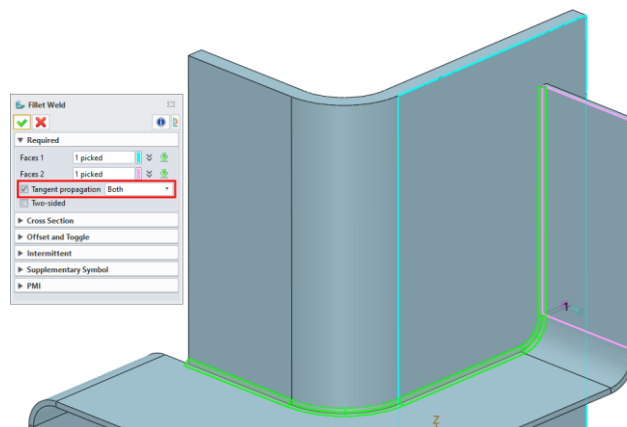
ZW3D 2024 adds “Pattern” option to Spot Weld. After the option is checked, you can create spot weld feature by setting the direction, number, and spacing. After the feature is generate, PMI dimension of spot weld is automatically generated. User can quickly generate multiple spot weld of drawing and dimension. The dimension can be synchronously updated when modifying feature parameter to improve the drawing and dimension efficiency of spot weld.



→ Where is it

Part/Assembly >> Weldments >> Frame >> Spot Weld

3.9.3 Add Tangent Propagation to Fillet Weld





ZW3D 2024 adds “Tangent propagation” option. If checked it, the generating scope of fillet weld will externally propagate from the selected surface according to the surface tangent condition. From the drop-down list, you can choose to

→ Where is it

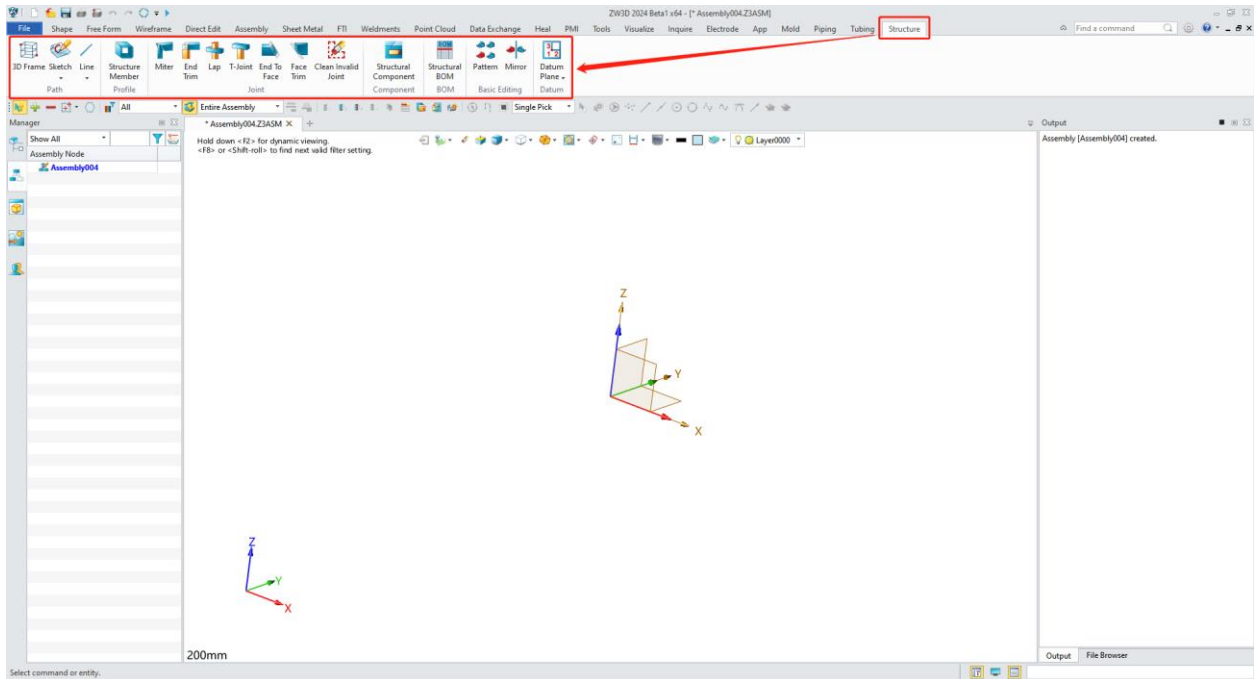
[Part/Assembly >> Weldments >> Frame >> Fillet Weld](#)

3.10 ★Structure

In ZW3D 2024, the original weldment module that based on multiple entity is upgraded to component based structural assembly module, which comprehensively improves the ability to construct structures scenarios and better meet the actual needs of engineering and customers.

3.10.1 Structural Module

ZW3D 2024 adds a structure module to the assembly environment. The structure module includes the functions of profile line drawing, structure member creation and editing, joint creation and editing, component creation and editing and structural BOM, etc.

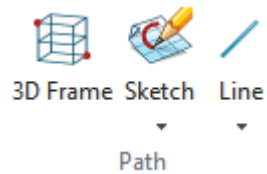


→Where is it

Assembly Environment >>Structure

3.10.2 Profile Line Drawing

ZW3D 2024 structure module provides 3D frame, sketch, 3D sketch, point, line and arc functions for drawing profile lines.

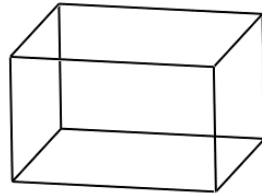


→Where is it

Assembly Environment >>Structure

3.10.2.1 3D Frame

ZW3D 2024 structure module provides 3D frame function for drawing 3D frame lines with cuboid outline.



→Where is it

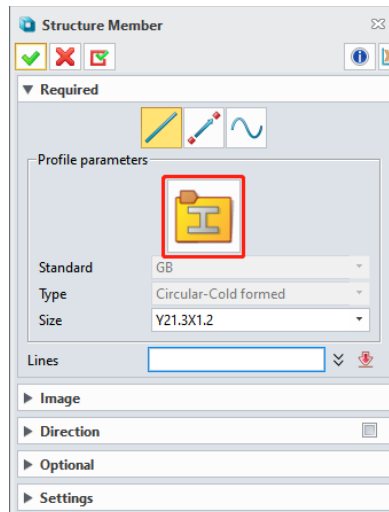
Assembly Environment >>Structure>>Path>>3D Frame

3.10.3 Structural Member Creation and Editing

ZW3D 2024 structure module provides structural member functionality for the creation of structural members.

3.10.3.1 New Structure Member Library

In ZW3D 2024, the structure member interface has been fully optimized to provide users with library files for creating structure members.

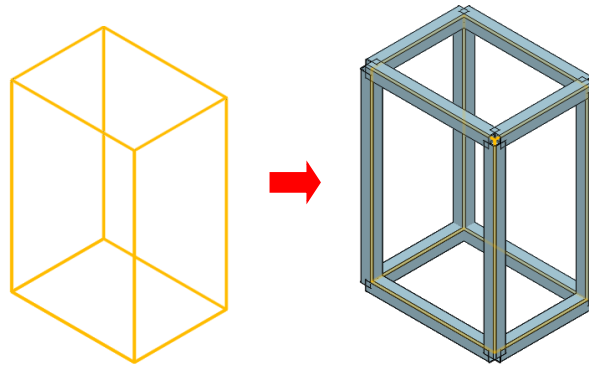


→Where is it

Assembly Environment >>Structure>>Profile>>Structure Member

3.10.3.2 Batch Create Structure Member

In ZW3D 2024, users can batch create all the required structure members in one batch by single, multiple or box selection based on the constructed profile lines.

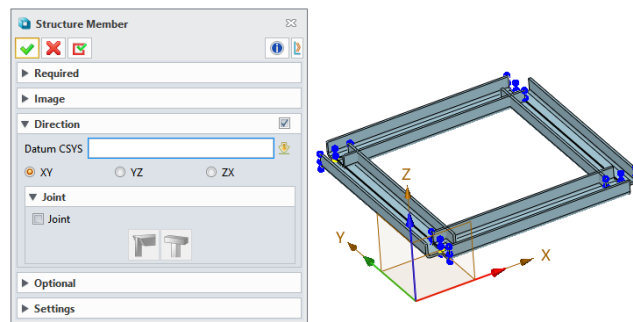


→Where is it

Assembly Environment >>Structure>>Profile>>Structure Member

3. 10. 3. 2. 1 Unified Orientation to Create Structure Member

In ZW3D 2024, users can unify the orientation of structure members when batch creating structure members.

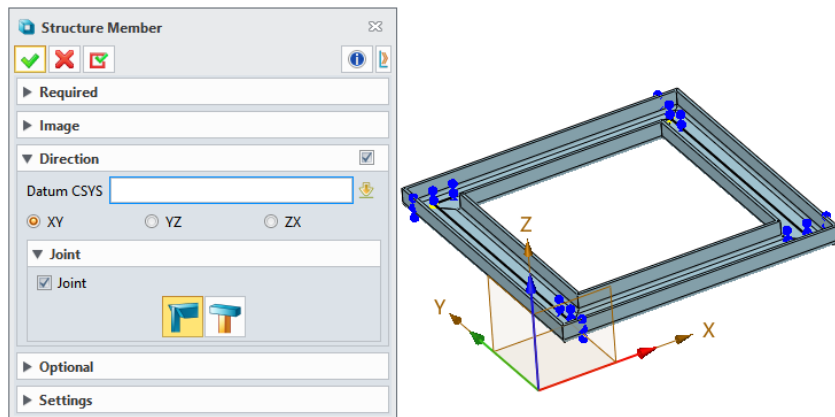


→Where is it

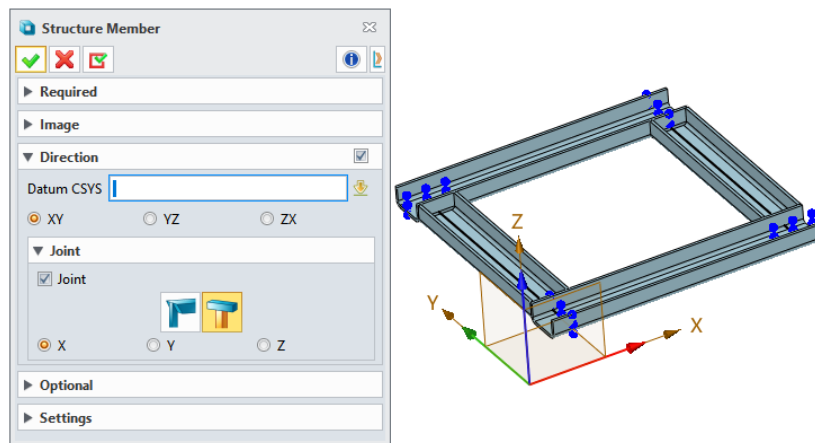
Assembly Environment >>Structure>>Profile>>Structure Member

3. 10. 3. 2. 2 Auto-Create Joint

In ZW3D 2024, users can set automatic joints creation based on the uniform structure members orientation when batch creating structure members. The joint types that can be created are miter or T-joint.



When you create a T-joint, you can select the direction in which the structure member of the longest edge is located.

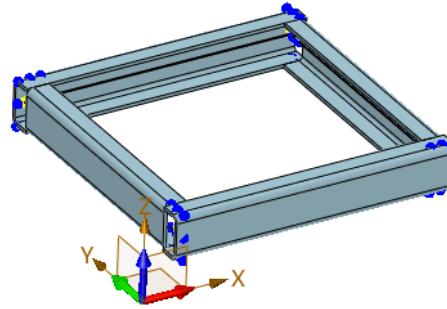
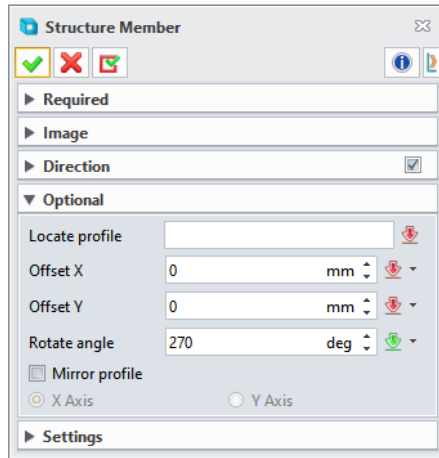


→Where is it

Assembly Environment >>Structure>>Profile>>Structure Member

3.10.3.3 Adjust Structure Member Orientation

In ZW3D 2024, users can flexibly adjust the orientation of structure members when creating structure members.



→Where is it

Assembly Environment >>Structure>>Profile>>Structure Member

3.10.3.4 Edit Structure Member

In ZW3D 2024, after creating a structure member, users can select the structure member and click the right mouse button to edit the type, size, direction, name, and other contents of the structure member through the function of editing structure member.



→Where is it

Assembly Environment >>Structure>>Edit Structure Member

3.10.3.5 Delete Structure Member

In ZW3D 2024, after creating the structure member, the user can select the structure member and click the right mouse button to delete the structure member by delete structure component function.

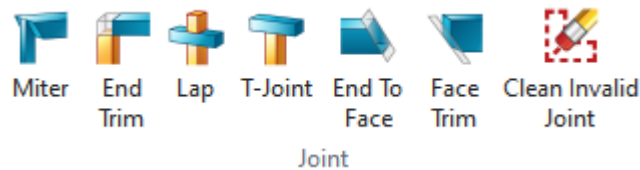


→Where is it

Assembly Environment >>Structure>>Delete Structure Component

3.10.4 Joint Creation and Editing

ZW3D 2024 structure module provides miter, end trim, lap, T-joint, end to surface, face trim, clean invalid joint, edit joint and other functions for joint creation and editing.

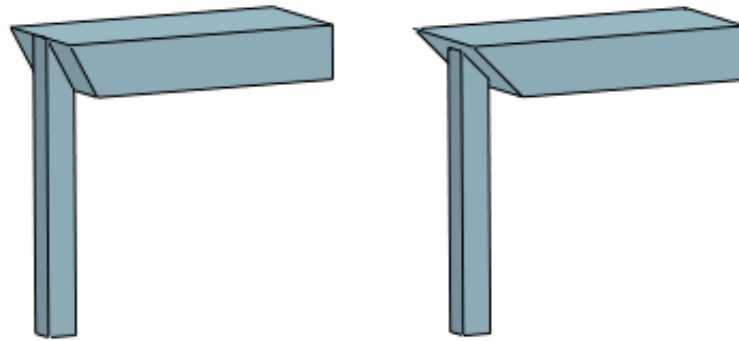


→Where is it

Assembly Environment >>Structure>>Joint

3.10.4.1 Miter

The miter function of the ZW3D 2024 structure module supports the creation of both holonomic miter and even miter results. Among them, the holonomic miter can make the projection of two structure members on the intersection direction of the plane where the section is in the mitering position completely coincide; Even miter allows the miter surfaces of two structure members to be located on the angular bisector of the plane where their sections are located.

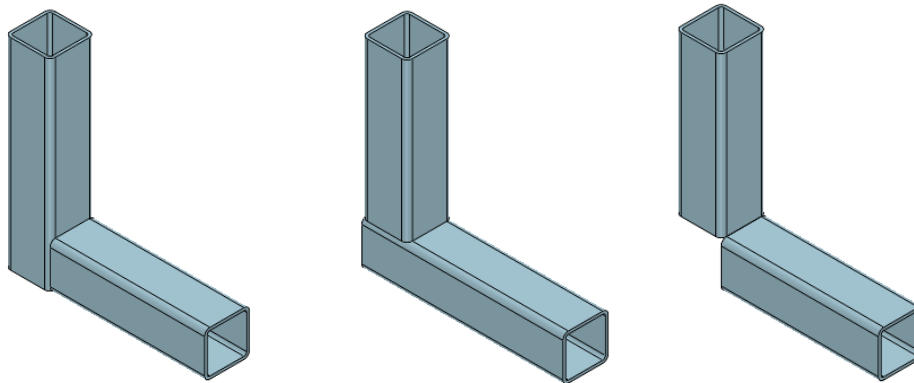


→Where is it

Assembly Environment >>Structure>>Joint>>Miter

3.10.4.2 End Trim

The end trim function of the ZW3D 2024 structure module supports the creation of end trim 1, end trim 2, and end trim 3.

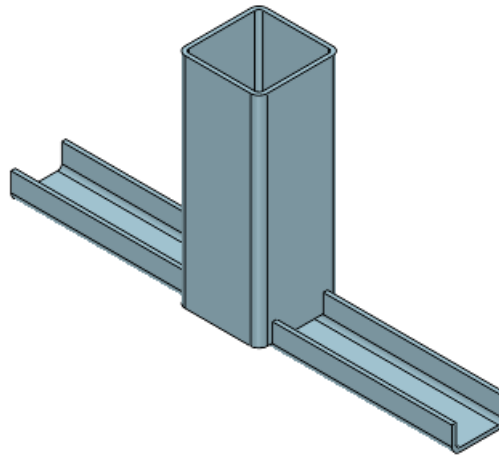


→Where is it

Assembly Environment >>Structure>>Joint>>End Trim

3.10.4.3 Lap

The lap function of the ZW3D 2024 structure module enables one structural member to be the matrix and the other structural member to be the trimming body, which can be used as a tool to trim the matrix.

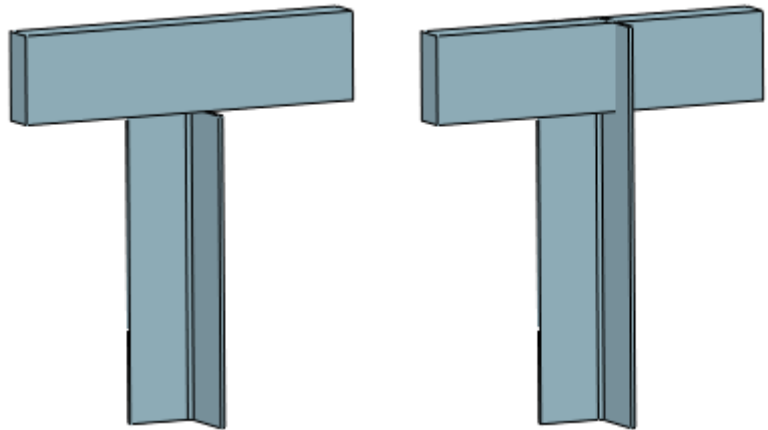


→Where is it

[Assembly Environment >>Structure>>Joint>>Lap](#)

3.10.4.4 T-Joint

The ZW3D 2024 structure module's T-joint function supports the creation of both "no overlap" and "overlap" results. Where, "no overlap" supports trimming or extending the matrix structure member to the plane close to the matrix structure member in the rectangular outer profile plane of the trim body structure member, and "overlap" supports trimming or extending the matrix structure member to the plane far away from the matrix structure member in the rectangular outer profile plane of the trim body structure member.

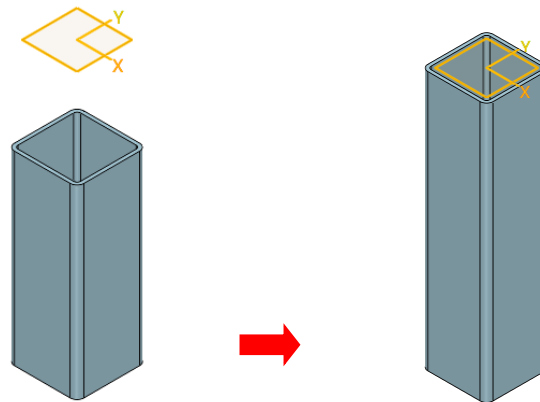


→Where is it

Assembly Environment >>Structure>>Joint>>T-Joint

3.10.4.5 End to Face

The ZW3D 2024 structure module's end to face function supports trimming or extending the end of the structure member to a certain face.

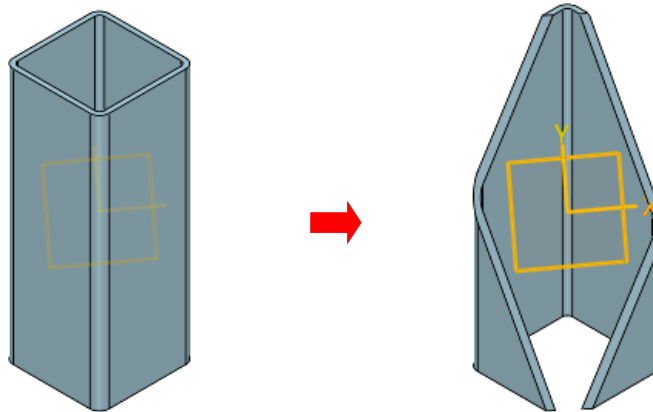


→Where is it

Assembly Environment >>Structure>>Joint>>End To Face

3.10.4.6 Face Trim

The face trim function of the ZW3D 2024 structure module supports trimming the existing structure member with the selected surface. During trimming, it is required that the selected surface and the structure member must have an intersection before trimming. Otherwise, the trimming operation cannot be completed.

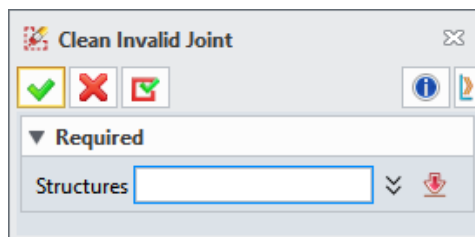


→Where is it

Assembly Environment >>Structure>>Joint>>Face Trim

3.10.4.7 Clean Invalid Joint

The ZW3D 2024 structure module's clean invalid joint function supports the removal of invalid joints in structure member.



→Where is it

Assembly Environment >>Structure>>Joint>>Clean Invalid Joint

3.10.4.8 Edit Joint

In ZW3D 2024, after creating structure members and joints, users can select structure member and click the right mouse button to edit joints through the function of edit joint.

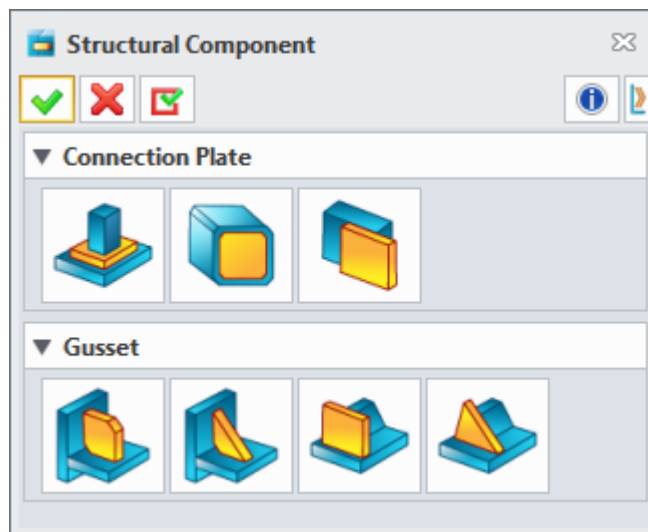


→Where is it

Assembly Environment >>Structure>>Edit Joint

3.10.5 Component Creation and Editing

The ZW3D 2024 structure module provides the ability to create and edit structural components. Two broad categories of structural components can be created, connection plate and gusset. The connection plate comprises end plate, end cap and end bracket; gusset include rectangular gusset, triangular gusset, side rectangular gusset and side triangular gusset.



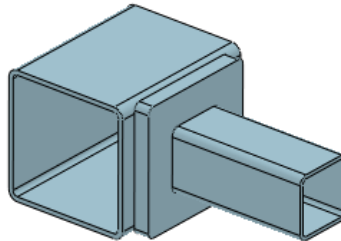
→Where is it

Assembly Environment >>Structure>>Component>>Structure Component

3.10.5.1 Connection Plate

3. 10. 5. 1. 1 End Plate

The ZW3D 2024 structure module provides the end plate function. Users can customize the size of the end plate and whether to perform operations such as fillet or chamfer.



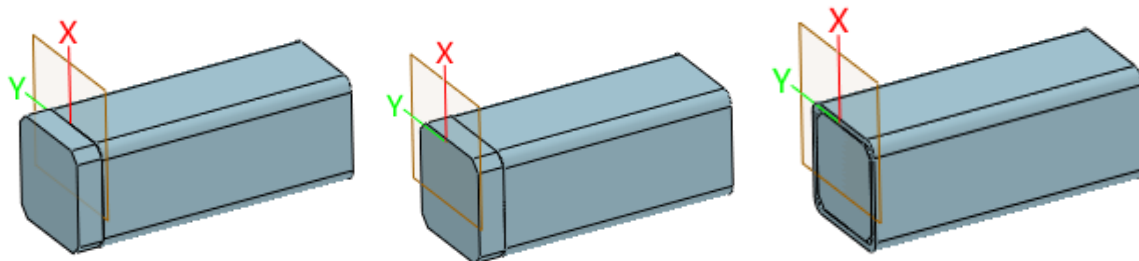
→Where is it

Assembly Environment >>Structure>>Component>>Structure Component>>Connection Plate>>End Plate

3. 10. 5. 1. 2 End Cap

The ZW3D 2024 structure module provides the end cap function, allowing the user to customize the size of the end cap by adjusting the thickness ratio (or constant distance) and thickness, and can define whether to fillet or chamfer the end cap.

There are three ways to create an end cap: create a non-trimming end cap outside the structure member, create a trimming end cap on the structure member, and create a non-trimming end cap inside the structure member.

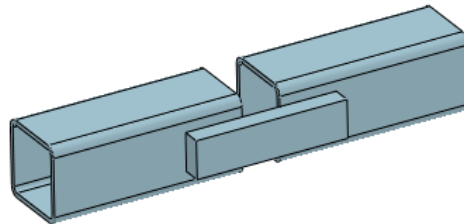


→Where is it

Assembly Environment >>Structure>>Component>>Structure Component>>Connection Plate>>End Cap

3. 10. 5. 1. 3 End Bracket

The ZW3D 2024 structure module provides the end bracket function. Users can customize the size of the end bracket.



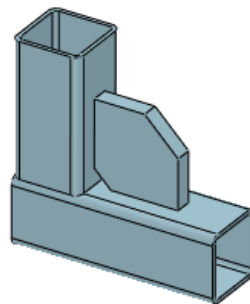
→Where is it

Assembly Environment >>Structure>>Component>>Structure Component>>Connection Plate>>End Bracket

3.10.5.2 Gusset

3. 10. 5. 2. 1 Rectangular Gusset

The ZW3D 2024 structure module provides rectangular gusset function, which allows users to customize the size of rectangular gusset.

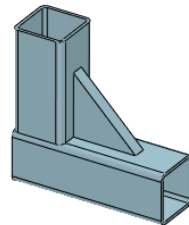


→Where is it

Assembly Environment >>Structure>>Component>>Structure Component>>Gusset>>Rectangular Gusset

3. 10. 5. 2. 2 Triangular Gusset

The ZW3D 2024 structure module provides triangular gusset function, which allows users to customize the size of triangular gusset.

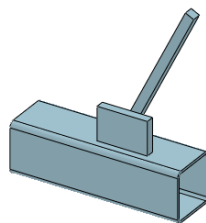


→Where is it

Assembly Environment >>Structure>>Component>>Structure Component>>Gusset>>Triangular Gusset

3. 10. 5. 2. 3 Side Rectangular Gusset

The ZW3D 2024 structure module provides side rectangular gusset function, which allows users to customize the size of side rectangular gusset.



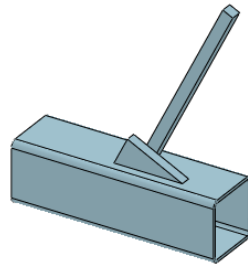
→Where is it

Assembly Environment >>Structure>>Component>>Structure Component>>Gusset>>Side

Rectangular Gusset

3.10.5.2.4 Side Triangular Gusset

The ZW3D 2024 structure module provides side triangular gusset function, which allows users to customize the size of side triangular gusset.



→Where is it

[Assembly Environment >>Structure>>Component>>Structure Component>>Gusset>>Side Triangular Gusset](#)

3.10.5.3 Edit Component

In ZW3D 2024, the user can select the component and click the right mouse button to edit the component through the function of edit component after completing the creation of the structural component.



→Where is it

[Assembly Environment >>Structure>>Edit Component](#)

3.10.5.4 Delete Component

In ZW3D 2024, after the component is created, users can select the component and click the right

mouse button to delete the component through the function of delete structure component.



→Where is it

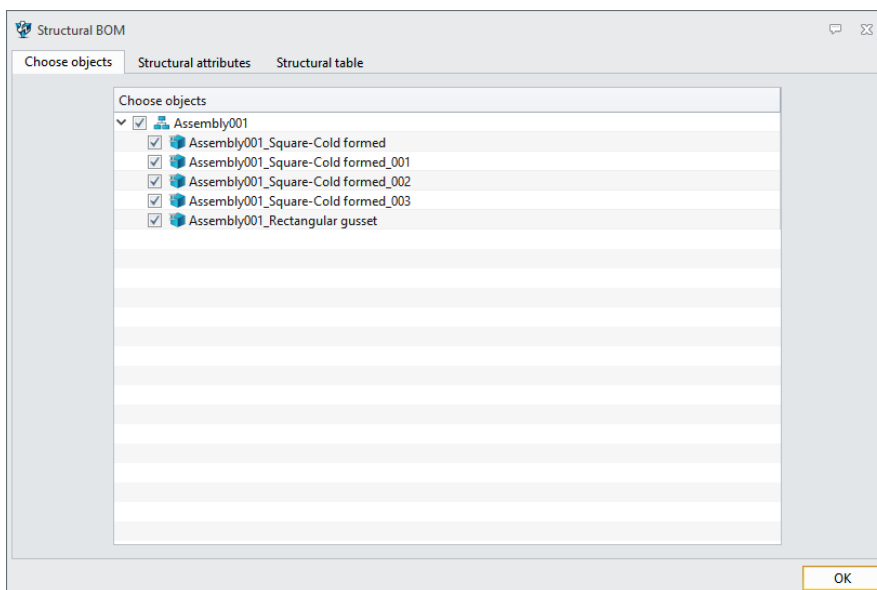
Assembly Environment >>Structure>>Delete Structure Component

3.10.6 Structural BOM

ZW3D 2024 Structure module provides the function of structural BOM. By default, all structural members and components in the current structural module are counted in the structural BOM.

3.10.6.1 Structural BOM in Assembly

Structural BOM in ZW3D 2024 structure module, whose main function is to list the information of each structural part, such as number, name, ID, specification, material, material standard, industrial standard, description, mass, total mass, quantity, length, etc.



Structural BOM supports users to add additional category information, export structural BOM and combine structure with same specification and material according to their own requirements.

Structural BOM

Choose objects | Structural attributes | **Structural table**

Assembly001_Square-Cold formed

Assembly001_Square-Cold formed_001

Assembly001_Square-Cold formed_002

Assembly001_Square-Cold formed_003

Assembly001_Rectangular gusset

Property name	Value
Name	Assembly001_Square-Cold formed
Number	
Material standard	GB/T600-2016
Industrial standard	GB_T 6728-2002
Description	
Angle1	0[*]
Angle2	0[*]
Rotation angle	
Angle direction	
Joint element type	
Specification	F20X1.2
Material	Q235A

Add
Delete

OK

Structural BOM

Choose objects | Structural attributes | **Structural table**

Available

Name

Number

Material standard

Industrial standard

Description

Angle1

Angle2

Rotation angle

Angle direction

Joint element type

Selected

ID

Specification

Material

Structure catalog

Mass

Quantity

Length

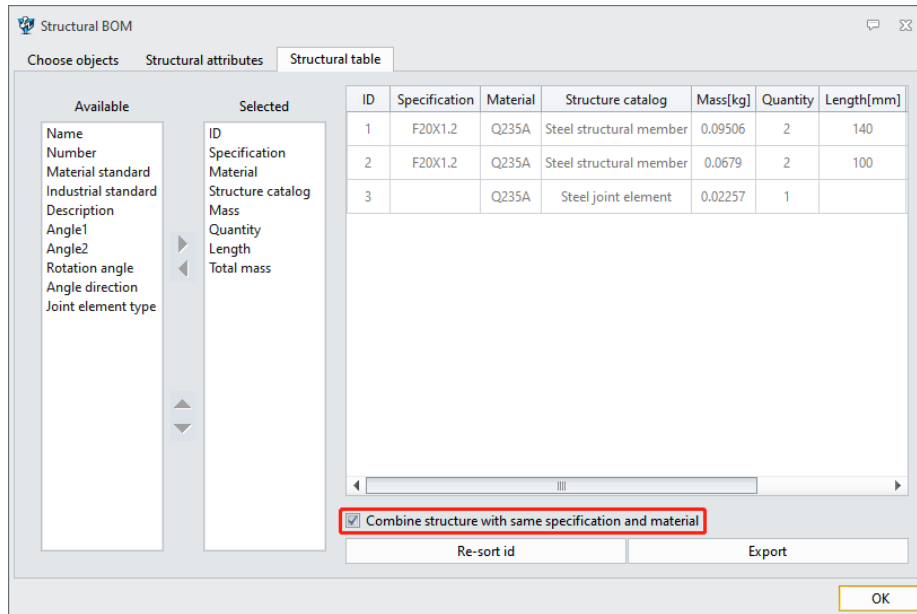
Total mass

ID	Specification	Material	Structure catalog	Mass[kg]	Quantity	Length[mm]
1	F20X1.2	Q235A	Steel structural member	0.09506	1	140
2	F20X1.2	Q235A	Steel structural member	0.0679	1	100
3	F20X1.2	Q235A	Steel structural member	0.09506	1	140
4	F20X1.2	Q235A	Steel structural member	0.0679	1	100
5		Q235A	Steel joint element	0.02257	1	

Combine structure with same specification and material

Re-sort id
Export

OK

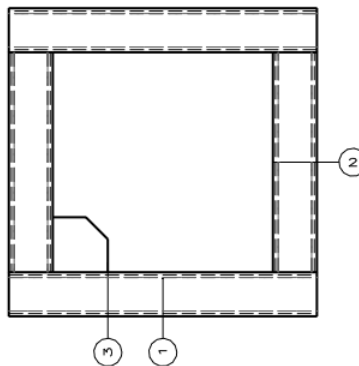


→Where is it

Assembly Environment >>Structure>>BOM>>Structural BOM

3.10.6.2 Structural BOM in Drawing Sheet

The main function of the structural BOM in the ZW3D 2024 engineering drawing environment is like the function of the structural BOM in the assembly.



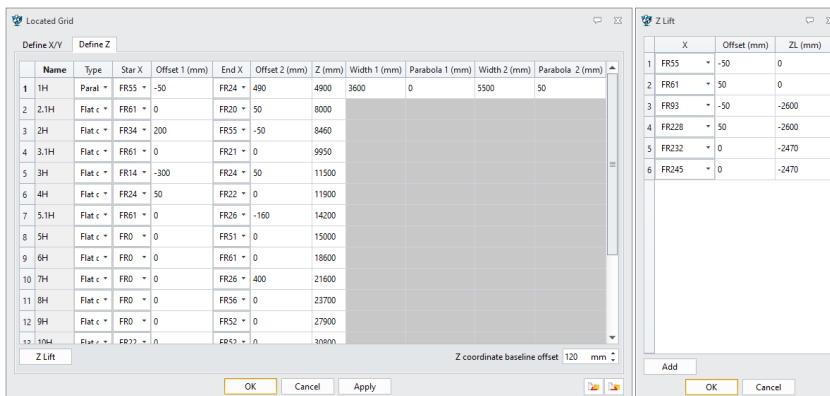
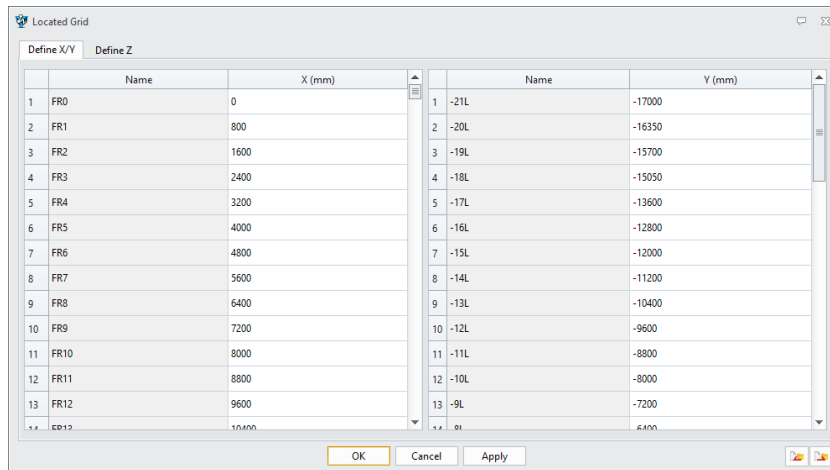
3		Q235A	Steel joint element	0.02257	1		0.02257
2	F20X1.2	Q235A	Steel structural member	0.0679	2	100	0.1358
1	F20X1.2	Q235A	Steel structural member	0.09506	2	140	0.19012
ID	Specification	Material	Structure catalog	Mass[kg]	Quantity	Length[mm]	Total mass[kg]

3.11 Piping & Tubing

3.11.1 New Located Grid

User can use the located grid function to add a coordinate system with engineering significance to the CSYS coordinate system to solve the coordinate conversion problem in large-scale scene pipeline design.

- Define location point of X axis to form cross section location of X axis.
- Define location point of Y axis to form cross section location of Y axis.
- Define location point of Z axis and shape of Y positive direction, and then define the lifting in X direction to form a complex Z-direction surface.



→ Where it is

Piping >> Piping Rules >> Define Grid

3.11.2 ★Optimized Spool Drawing Coordinate Dimension

In large scene model design, relative grid coordinates are more valuable than absolute coordinates. Spool drawing first identifies the located grid data, then the absolute coordinates located in grid range will be automatically converted into relative grid coordinates to improve the practical application significance of spool drawing.

- When located grid files are saved in the same location as the module, it can be identified automatically.

Connection Information			
Pipe End	Connector	Installing POS	Connecting OBJ
Main A	3	FR52-320, -2L+250, 7H-4250	
Main B	2	FR51-220, -2L-200, 6H-330	

→ Where it is

Piping >> Drawing >> Spool Drawing

3.11.3 Spool Drawing Processing Information Improvement

The fabrication of pipe spool includes two kinds of process: bending prior to welding and welding prior to bending. Bending prior to welding focuses on the start angle and the end angle of the pipe fittings while welding prior to bending pays attention to the relative angle of the pipe fitting. Spool drawing in new version solve the calculation problems in start angle, end angle, and the relative angle.

- Start angle (As shown in figure 31)
- End angle (As shown in figure 39)
- Relative angle (As shown in figure 118)

BOM							
ID	Name						Quantity
1	Stainless Steel Pipe-50-60.3X3.91-40S-PE-ASME B36.19-2018						1
2	Flange-50-Class125-SC-FLFF-ASME B16.1-2015						1
3	Flange-50-Class125-SC-FLFF-ASME B16.1-2015						1
Fabricating							
Spool	Feed	Rotate	Bend	PNT-PNT DIS	Pipe-PNT DIS	Radius	Cut
Main A-B	218	31	57	300	300	151	0
Main A-B	134	48	44	277	277	151	0
Main A-B	439	39	118	500	500	0	0

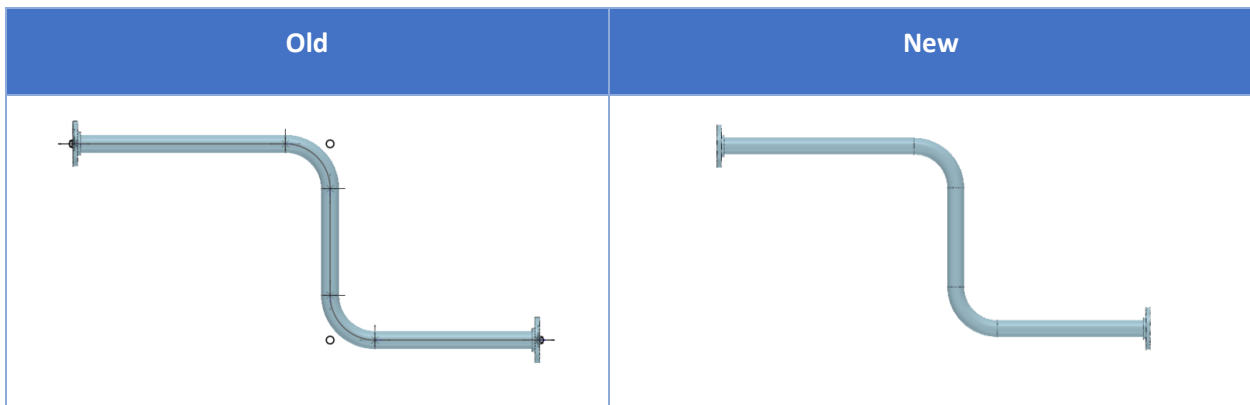
→ Where it is

Piping >> Drawing >> Spool Drawing

3.11.4 Routing Model Improvements

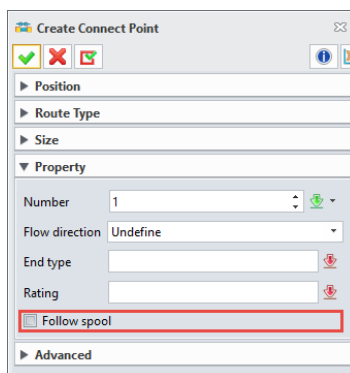
3.11.4.1 ★Default Enable Display Optimization

ZW3D 2024 does not display routing center line, connection point and insertion point by default, which can significantly improve routing model display. The routing model display is highlighted. Optional objects of routing center line, connection point and insertion point can display to allow being selected after the mouse is close to.



3.11.4.2 ★Connection Point Improvement

Connection point of standard part belongs to physical cut-off point in routing. Spool is pre-defined unit between the specified connection points according to the manufacture requirement. Thus, defining routing has this rule. To raise up the efficiency of defining segment, ZW3D 2024 optimizes “Create Connect Point” by adding self-defined spool and taking the follow-up spool of routing connection point as routing defining boundary to achieve self-defined spool. Therefore, remind to set “Follow spool” when creating library, and this can improve design efficiency.

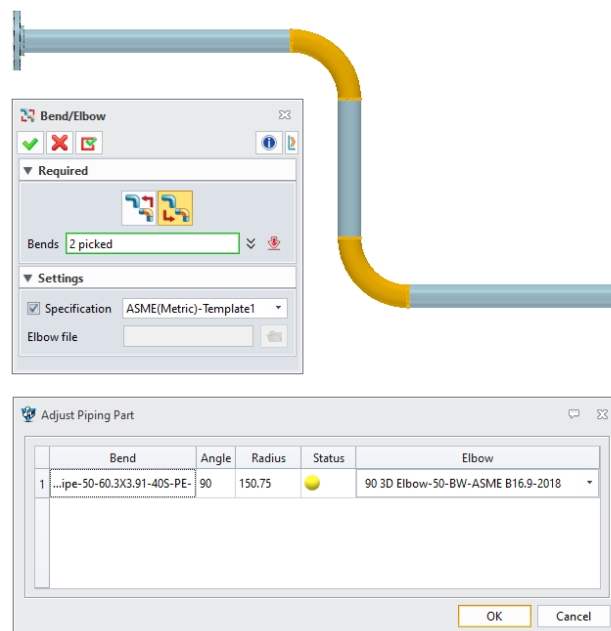


→ Where is it

Part/Assembly >> Insert >> Routing >> Create Connect Point

3.11.4.3 New Bend/Elbow

In 3D routing design, as the design purpose changes, it is usually necessary to change a pipe elbow to a bend or a bend to an elbow. ZW3D 2024 adds “Bend/Elbow” which achieve quick transfer between bend and elbow through batch modification.

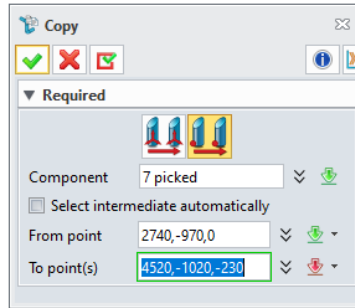
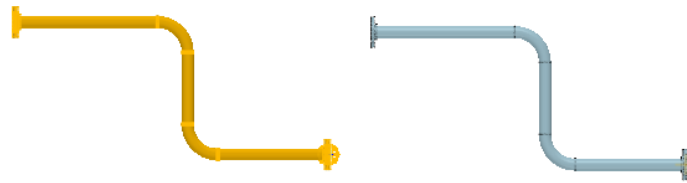


→ Where is it

Piping >> Modify >> Bend/Elbow

3.11.4.4 New Pipe Copy

In the piping design, it is necessary to copy a pipe to specified location because the design scenarios are similar. ZW3D 2024 newly adds a function “Copy” pipe to realize copy specified pipe, thus improving project’s design efficiency.

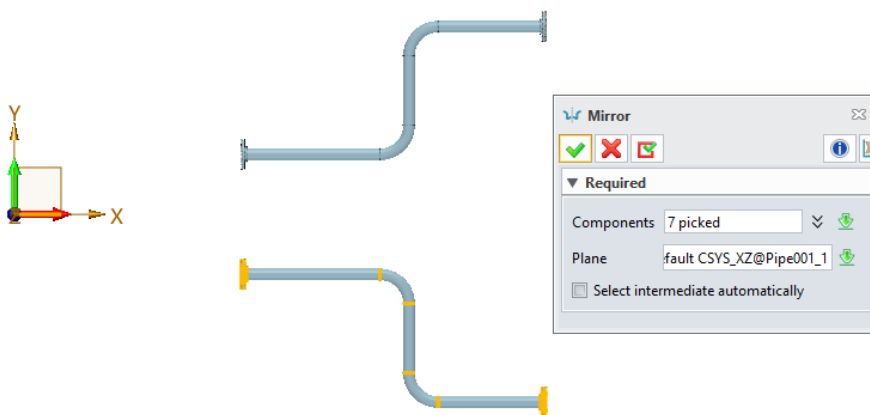


→ Where is it

Piping >> Modify >> Copy

3.11.4.5 New Piping Mirror

During the piping design, it is necessary to mirror the piping models because the design scenarios are similar. With the additional function of “Mirror”, ZW3D 2024 can achieve mirror the specified piping, thus improving the design efficiency of the project.

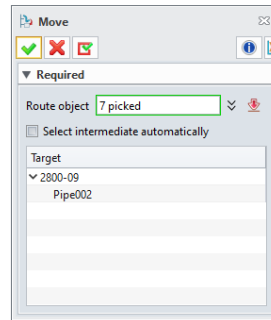
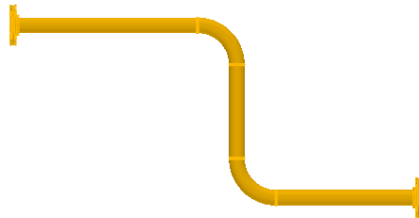


→ Where is it

Piping >> Modify >> Mirror

3.11.4.6 New Piping Transfer

During the piping design, it is necessary to transfer a piping assembly to another piping assembly due to the requirements of pipeline management. With the additional “Transfer” function, ZW3D 2024 can realize the transfer of specified pipeline, thus improving the pipeline management.

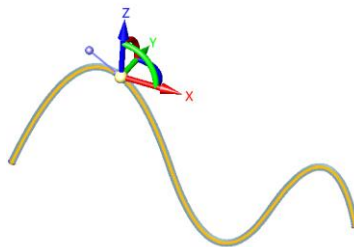


→ Where is it

Piping >> Modify >> Transfer

3.11.4.7 Modify Flexible Pipe Improvement

During the flexible pipe design, it is usually necessary to adjust the through points of flexible pipe. For easy control the direction of adjustment, ZW3D 2024 optimizes the “modify Flexible Pipe” function by adding Align type including WCS and View to realize the flexible pipe through points along the specified direction modification, thus improving the modification efficiency and usability.

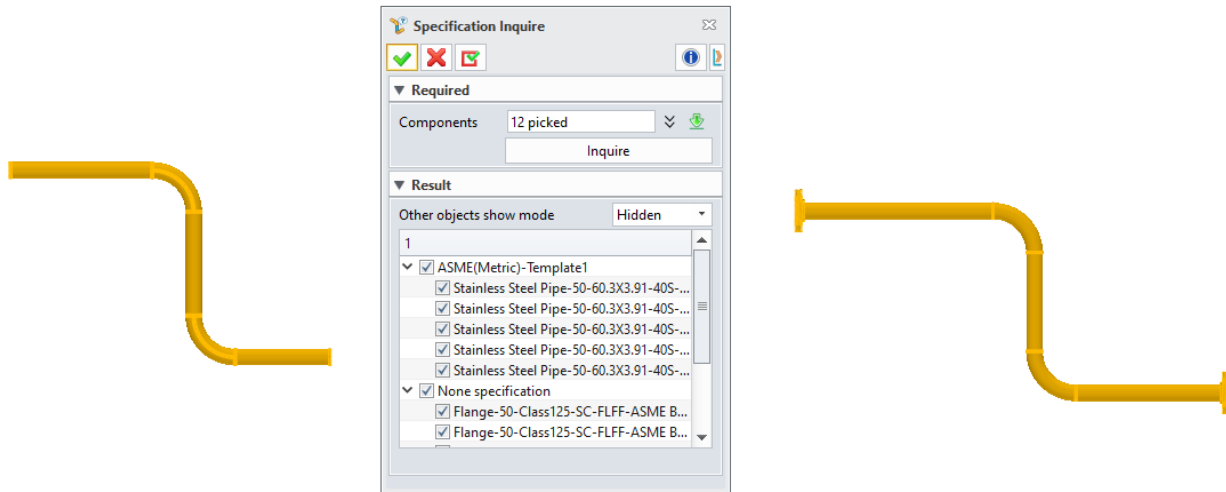


→ Where is it

Tubing >> Modify >> Modify Flexible Pipe

3.11.4.8 New Specification Inquire

During the pipeline design, you can choose specification rule or non-specification rule. It is necessary to inquire the specification rule of the piping model due to the hybrid design of pipeline model. With the additional “Specification Inquire” function, ZW3D 2024 realizes to inquire specification of the whole piping models of pipeline, which can separately display those models according to their specification.



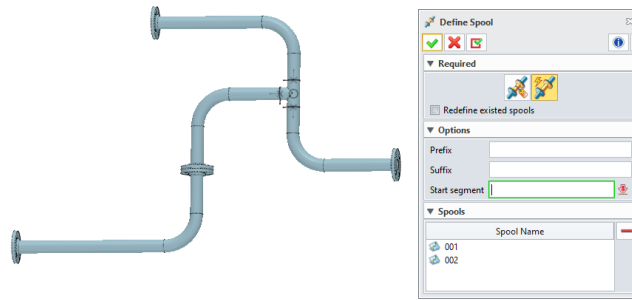
→ Where is it

→ Tubing >> Design Check >> Specification Inquire

3.11.5 ★Spool Drawing Improvement

3.11.5.1 Define Spool Improvement

Prior to generate spool drawing, it is necessary to define the entire pipeline model. ZW3D 2024 adds the “Auto define” option to the “Define Spool” function, which realizes batch defining spools of specified pipeline models and automatically completing spool definition according to the connection point attributes, thus increasing the design efficiency and standardizing the design.

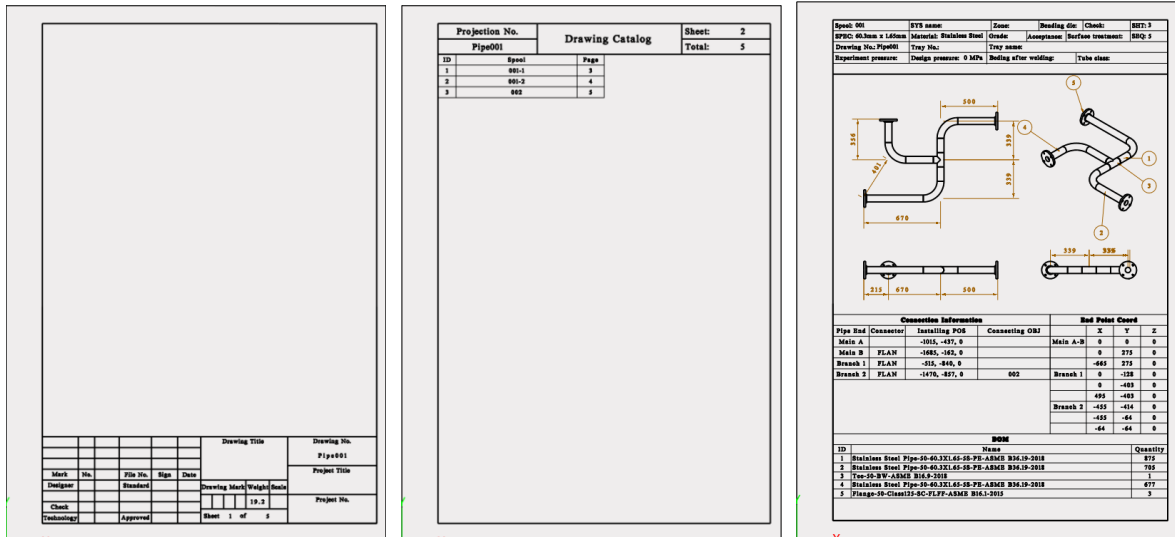


→ Where is it

Piping >> Design Check >> Define Spool

3.11.5.2 Spool Drawing Improvement

Spool drawing sheets are important sheets of spool section. This drawing is issued to the workshop as a brochure according to the standard flow. Thus, it is necessary to provide a complete drawing brochure that includes a cover, table of contents and main sheets. At the same time, the tables in the drawings are different due to various complexities of the spools. With the improvement of “Spool Drawing” function, ZW3D 2024 realizes the spool drawings generating an entire sheet and the tables in sheets aligning or switching pages, which take full advantage of smart layout to generate sheets. Thus, it improves the ability to customize drawings.



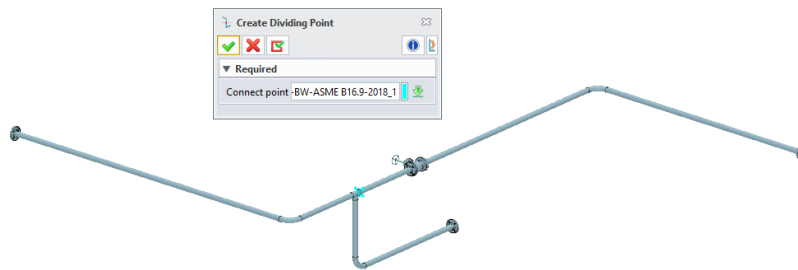
→ Where is it

Piping >> Drawing >> Spool Drawing

3.11.6 ★New Piping ISO Drawing

3.11.6.1 New Diving Point

It is impossible to express all pipeline information in a single pipeline ISO drawing as the pipeline is rather complicated. Therefore, it is necessary to divide into multiple sheets. With the additional “Create Dividing Point” function, ZW3D 2024 achieves to add dividing points freely in models and exporting multiple ISO drawings according to the dividing points in the end.

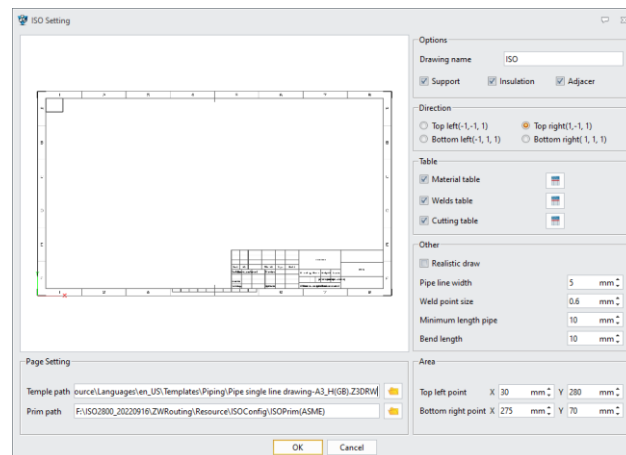


→ Where is it

Piping >> Drawing >> Create Dividing Point

3.11.6.2 New ISO Setting

Pipeline ISO drawing is an important sheet to guide pipeline construction. It is necessary to generate different sheets according to the realistic situation. With the additional “ISO Setting” function, ZW3D 2024 realizes the information settings of graphic library, sheet template, drawing name, view direction, and tables. The final ISO drawing will generate to pipeline ISO drawing according to the options in settings.

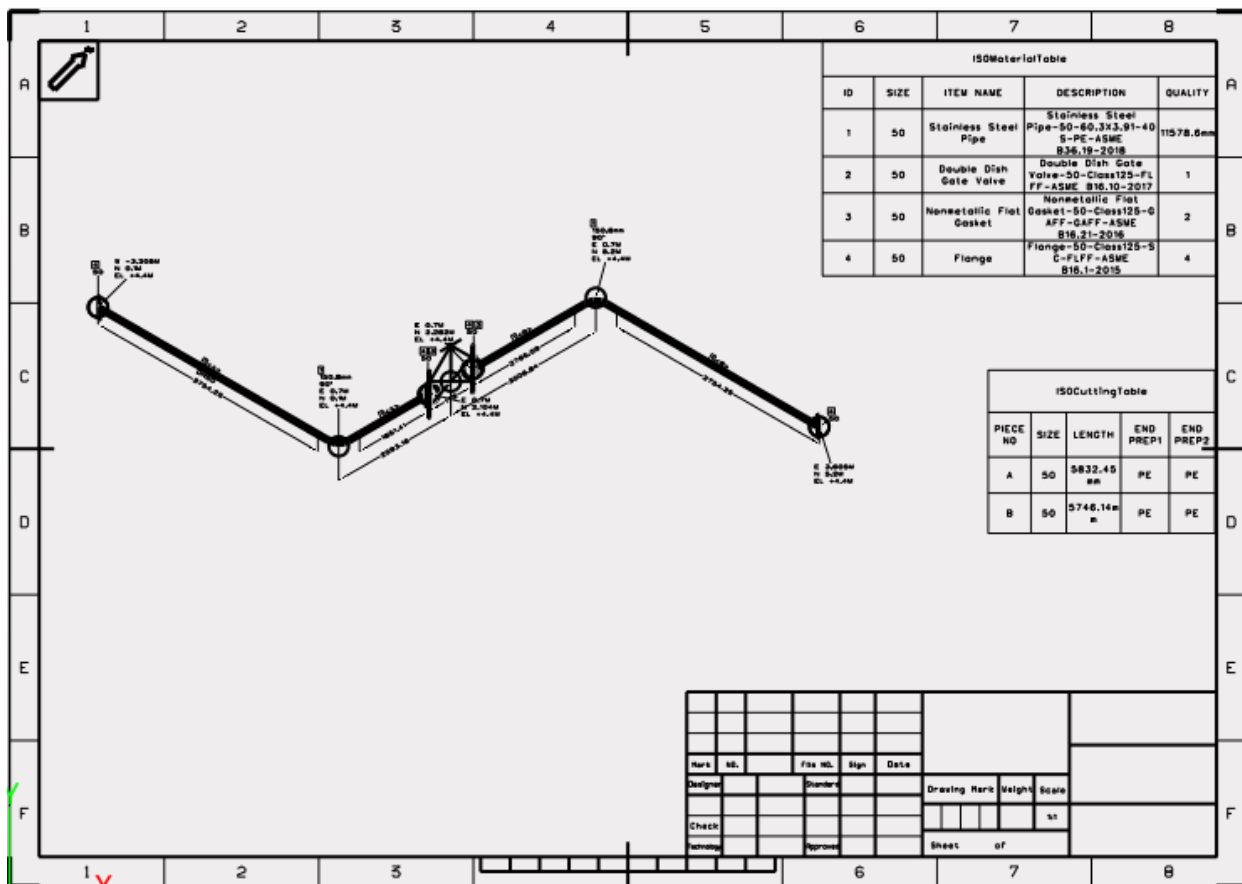


→ Where is it

Piping >> Drawing >> ISO Setting

3.11.6.3 New ISO Generate

Pipeline ISO drawing is an important sheet to guide pipeline construction. With the additional “ISO Generate” function, ZW3D 2024 realizes one-click to export pipeline ISO drawing. A pipeline ISO drawing can automatically dimension sizes but export BOM information.



→ Where is it

Piping >> Drawing >> ISO Generate

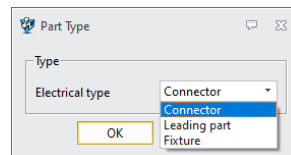
3.12 Harness

3.12.1 Harness Standard Part

3.12.1.1 New Harness Standard Part

ZW3D supports converting a general part into the harness standard part with the standard part type information. The electrical type includes Connector, Leading part, and Fixture.

- Connector: Components that connect wires and cables, such as switches, power supplies, connectors, and splitters.
- Fixture: Components for fixing wires and cables, such as clips.
- Leading part: Components that guide through wires and cables, such as elbows, tees.



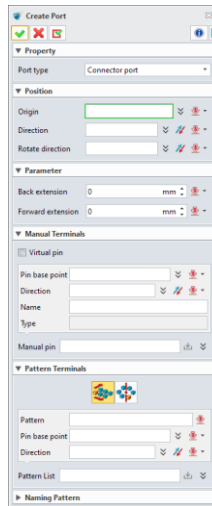
→ Where it is

[Insert >> Harness>> Define Standard Type](#)

3.12.1.2 ★New Create Port

To make it easier for user to add position, direction, and name information in connection, ZW3D added a Create Port to harness standard part, supporting circle and rectangle pattern to achieve quick definition of connectors.

there are two forms of defining terminals: Manual terminals and Pattern terminals. It should be noted that the pattern terminal currently only supports the array defined in ZW3D.



→ Where it is

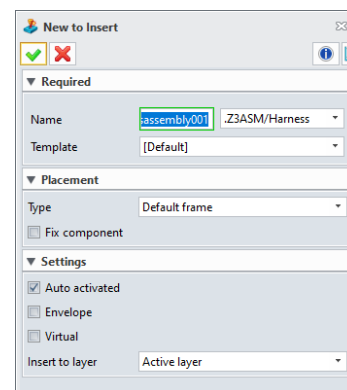
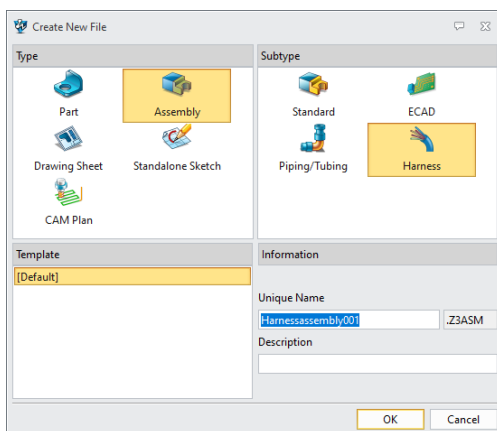
Insert >> Harness>> Create Port

3.12.2 Harness Assembly

3.12.2.1 New Harness Assembly

ZW3D added the harness assembly to manage the basic data of Harness. It can be created through creating a new file or inserting a part in the assembly.

- Creation of cable library and protective layer library in the wiring harness assembly.
- Multiple harness parts can be created within a harness assembly, with final routing done within the harness part.



→ Where it is

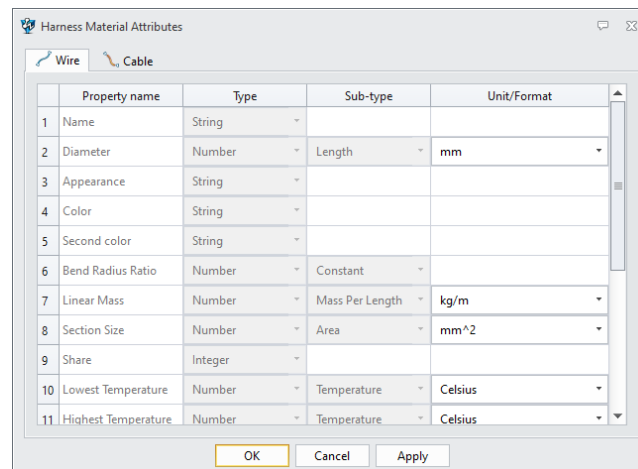
Create New File>> Harness

Assembly >> Component>> New to Insert

3.12.2.2 New Wire & Cable Attributes

We added Wire & Cable Attributes in harness assembly to meet complex cable design scene with the user-defined attribute support.

- Default attributes cannot be deleted, new attributes can be deleted.
- Adjusted attributes can be reflected in the “Define Wire/Cable” command.



→ Where it is

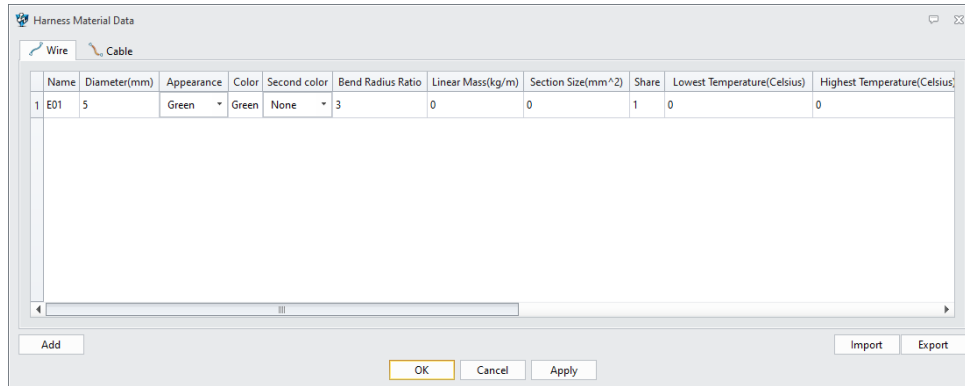
Harness >> Harness Material Attributes>>Wire/Cable Attributes

3.12.2.3 ★New Define Wire & Cable

In harness assembly, we added the cable definition to establish the library of wire (single core) and cable (multi-core). (Note: Wire and cable library is not provided in ZW3D temporarily).

- Define wire (single core wire) to build the wires library.

- Define cables (multi-core wire) and complicated cores include insulator, shield, twisted and wire cores and build cable library. (Note: You can define a cable core through "Show cable core" by right clicking the cable name).
- Automatically copy the property value created last time to improve the efficiency of cable material library creation.



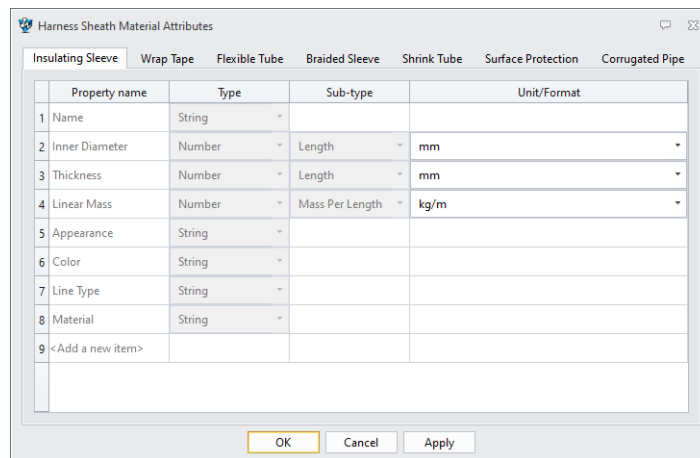
→ Where it is

Harness Assembly >> Harness Material Attributes>> Define Wire/Cable

3.12.2.4 New Protection Attribute

In harness assembly, we add protection attribute definition to meet more complex protective layer design scenario with user-defined protection attribute support.

- Default attributes cannot be deleted, new attributes can be deleted.
- Adjusted attributes can be reflected in the “Define Protection” command.



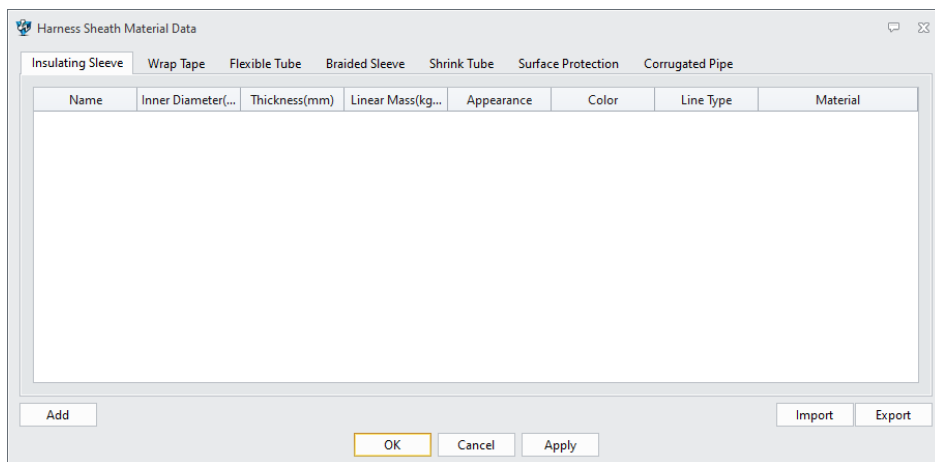
→ Where it is

[Harness Assembly >> Harness Material Attributes>> Protection Attributes](#)

3.12.2.5 New Define Protection

We added define protection in the harness assembly to solve the problem of creating protection database, which provides support for adding protection on the base of wire and cable. (Note: Protection library is not provided in ZW3D temporarily)

- Define various of protections, including insulating sleeve, wrap tape, flexible tube, braided sleeve, shrink tube, surface protection and corrugated pipe. Create the protection library in need.
- Automatically copy the attributes created last time and improve the creation efficiency of protection material library.



→ Where it is

[Harness Assembly >> Harness Sheath Material Attributes>> Define Protection](#)

3.12.3 Harness Part

3.12.3.1 New Harness Part

We added the harness part to manage the harness objects including wires and cables and protection (Note: harness part can only be created in the harness assembly). All wiring-related functions are

performed within the harness parts. Furthermore, harness parts are virtual zero-levels and cannot be opened individually.

A wiring harness part manages a wiring harness as a unit for drawing and statistics of the wiring harness.

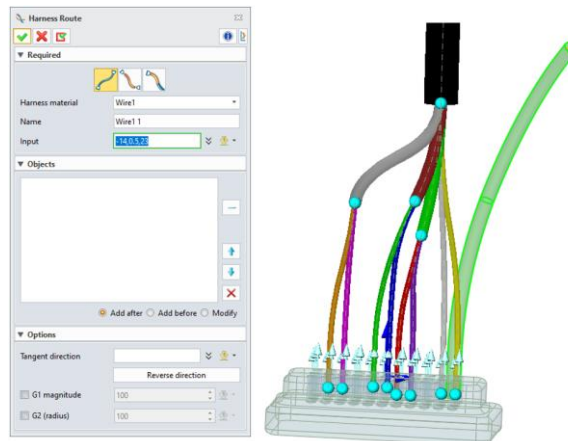
→ [Where it is](#)

[Harness Assembly >> Harness File>> Create harness](#)

3.12.3.2 ★New Harness Route

The newly added functionality of harness route can solve the problem of wiring. The wires and cable are created without historical features with support in straight section.

- Modify harness supports selecting any point for free modifying harness and can also select key points based on the reference structure, such as the center of the circle and the offset point.
- Modify harness supports adjusting, adding harness path location and its tangent direction information.
- Support the creation of straight state cables between two points during modifying harness.

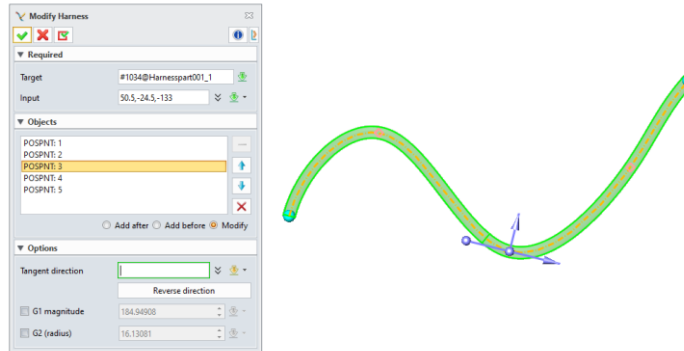


→ [Where it is](#)

[Harness Part >> Harness Route>> Harness Route](#)

3.12.3.3 New Modify Harness

The newly added modify harness supports users to add, modify, delete, and adjust the order and parameters of the position points of wire and cable.

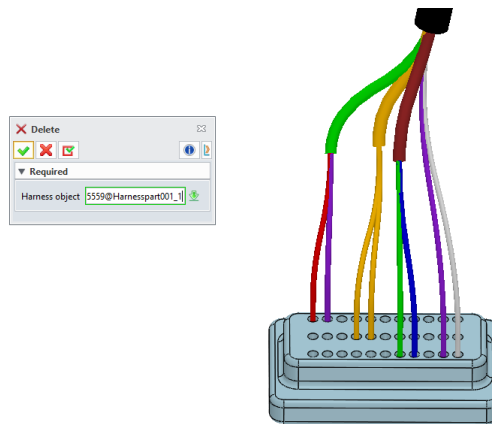


→ Where it is

Harness Part >> Harness File>> Modify Harness

3.12.3.4 New Deletion

The newly added Deletion can delete the created wires, cables, or cores, with support to delete the entire wire and partial core of cable to meet on-demand deletion in different scenarios.

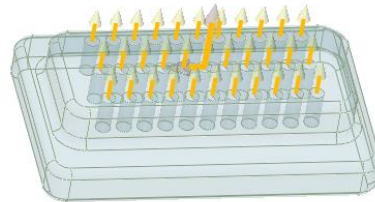
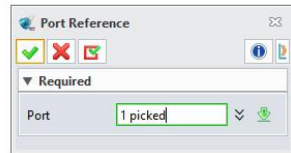


→ Where it is

Harness Assembly >> Edit>> Delete

3.12.3.5 New Port Reference

The newly added port reference is used to refer the port (including pins) position and information of the wire harness standard parts to the wire harness parts so that it can be used in the wiring process. When the actual port changes, it can be refreshed through the reference.



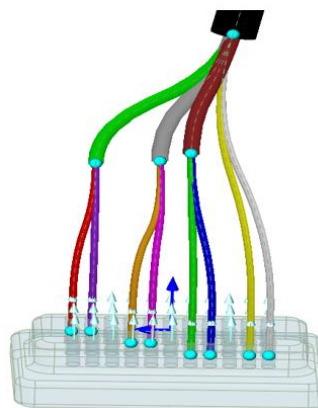
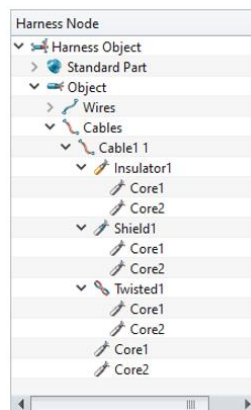
→ Where it is

[Harness Assembly >> Reference>> Port Reference](#)

3.12.3.6 ★New Harness Object Manager

The newly added harness object manager to manage the created objects. Harness Object Manager provides following functions:

- List all the wires and cables of the current harness parts and distinguish display their internal composition through different node forms.
- Support the selection of each cable object in the node on the tree and highlight the cable graphic object.
- The node supports the right-click menu. Supports delete, show, hide and transparent switching of node objects.

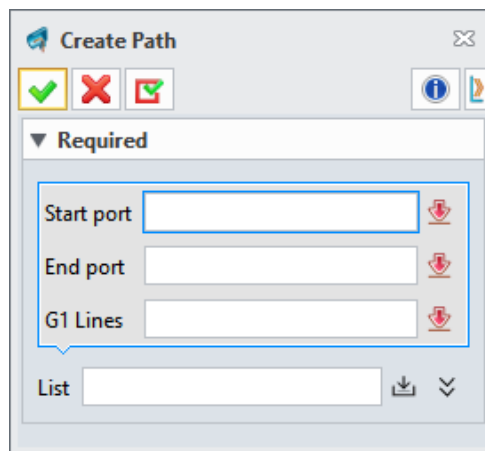


→ Where it is

Harness Part >> Manager >> Harness Object Manager

3.12.4 ★Path Setting in Leading Part

In the harness standard parts, it is necessary to set leading path in addition to set leading direction. Therefore, ZW3D 2024 adds a new function of creating leading path that can be easy to precisely route lines by referring to the leading part path.



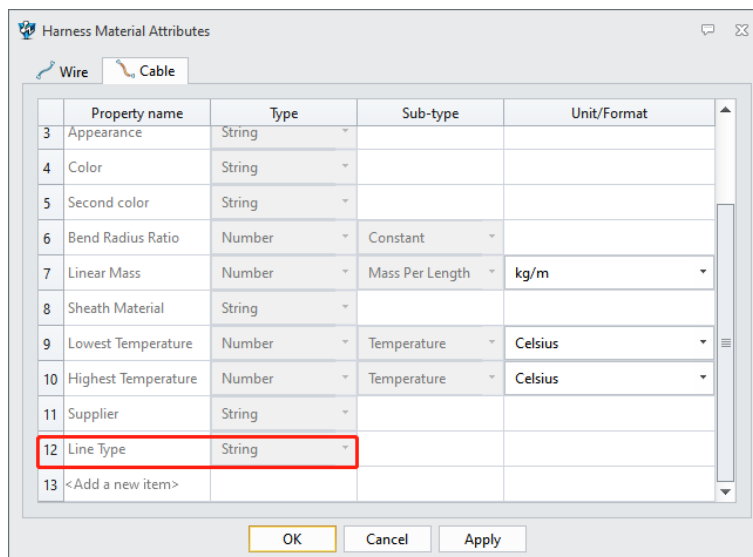
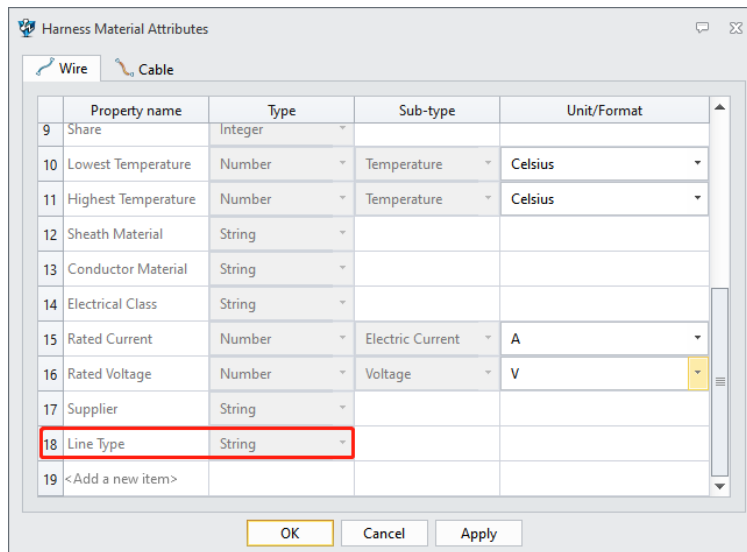
→Where is it

Insert >> Harness >> Create Path

3.12.5 Line Type

3.12.5.1 New Definition of Line Type

To prepare auto-drawing for later nail board drawing, Wire/Cable Attribute adds an option “Line Type”. The line type can be expressed in the nail board drawing in the later version, which can be distinguished according to the line pattern.

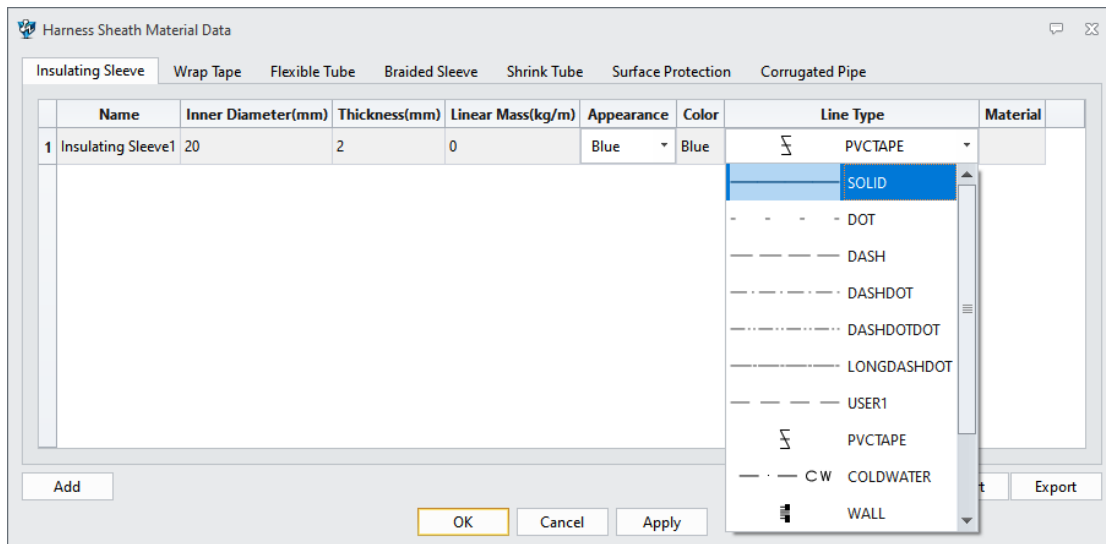
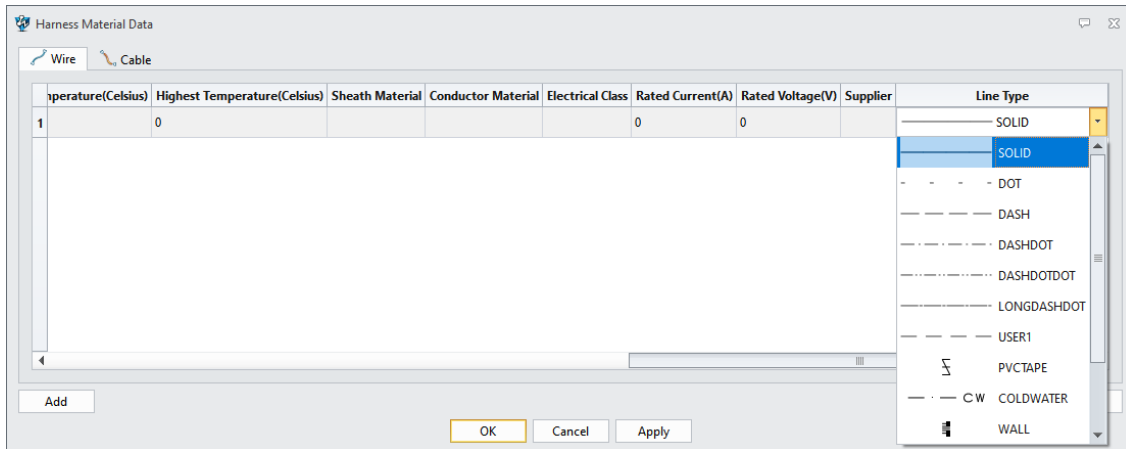


→Where is it

Harness >> Attribute >> Wire/Cable Attribute

3.12.5.2 New Line Type

As “Line Type” has been added to wire and cable attribute, you can select the line type when defining wire and cable. Similarly, you can select the line type for protection when defining protection. There are 13 line-types for user to select and use.



→Where is it

Harness >> Attribute >> Wire/Cable Attribute

Harness >> Attribute >> Protection Attribute

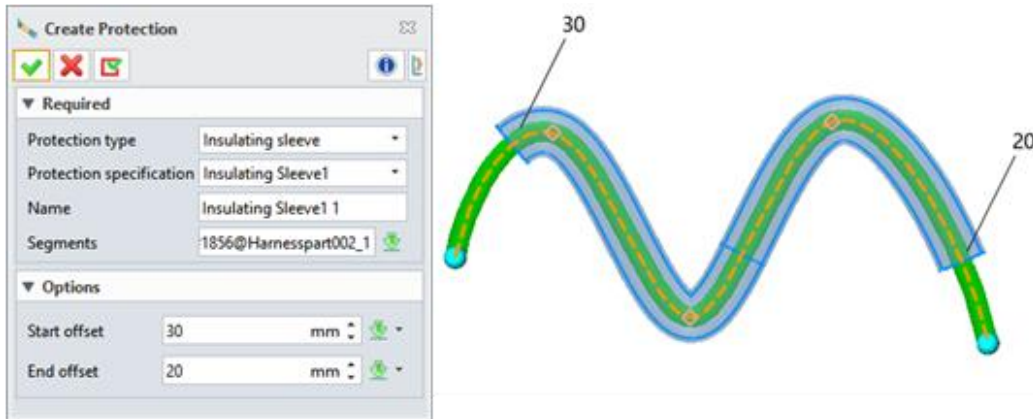
3.12.6 Harness Part

3.12.6.1 ★New Create Protection

During the harness design, wire and cable is often covered by protection. Thus, ZW3D 2024 supports creating protection.

- A protection can be placed on the outside of wire and cable.

- Support selecting protection type and renaming it.
- Support setting protection distance between the two ends of internal wire and cable



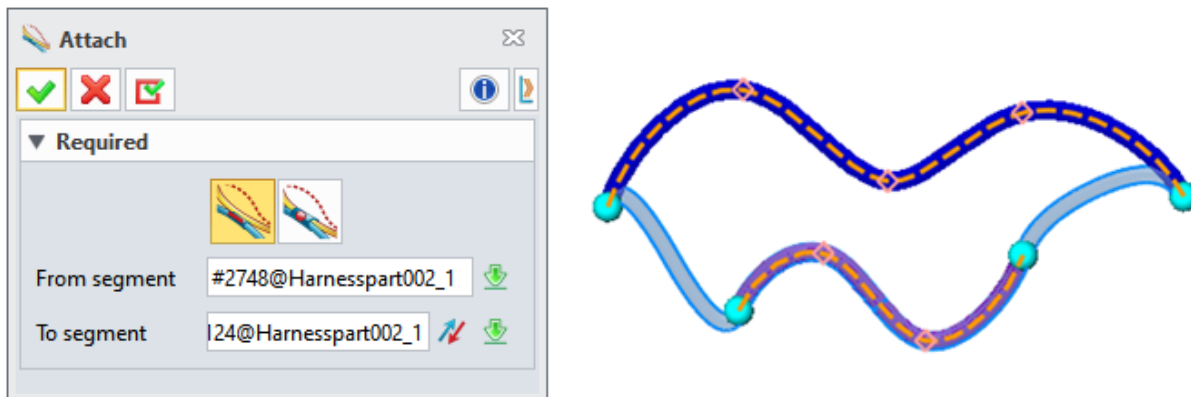
→Where is it

[Harness >> Harness File >> Create Protection](#)

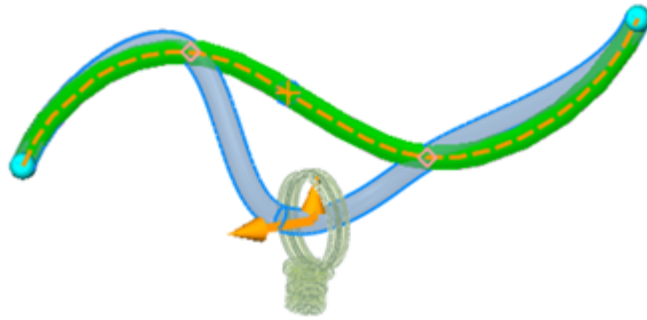
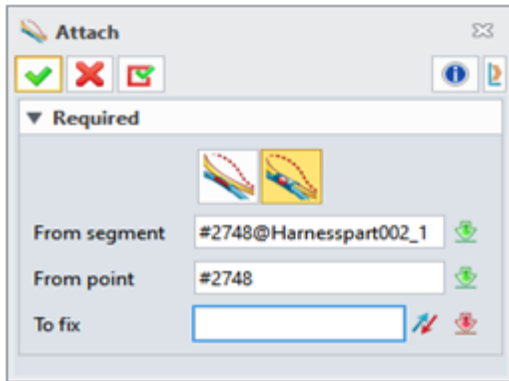
3.12.6.2 New Attach

During harness routing process, there is a scenario where a wire harness is arranged with another. Thus, the function of attach is added. Currently, there are two attaching functionalities:

- Attach to segment: allow wire and cable attaching to one segment of wire and cable for routing.



- Attach to port: allow some point of wire and cable attaching to the port of fixture and leading part.

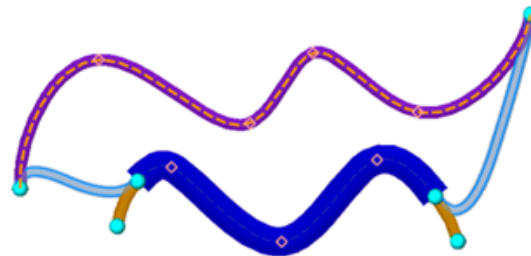
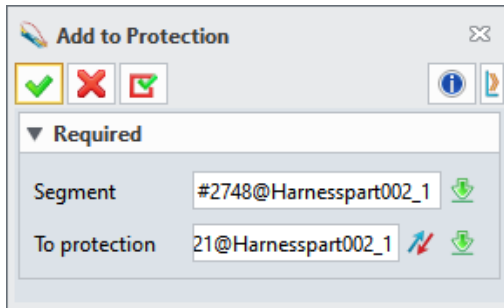


→Where is it

Harness >> Edit >> Attach

3.12.6.3 Add to Protection

The function of Add to Protection is added to wire and cable. During the harness creation process, there is a scenario when multiple wire and cable shares with some protection. The current protection can only be created in one wire and cable. Thus, it is necessary to add other wire and cable to the existing protection.



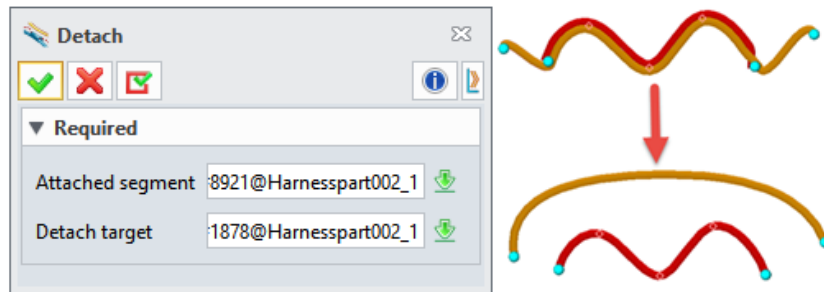
→Where is it

Harness >> Edit >> Add to Protection

3.12.6.4 New Detach

For the wires and cables that have attached to other wires and cable or added to the protection, the

detach function can be used to free the attachment relations in the wires and cables.



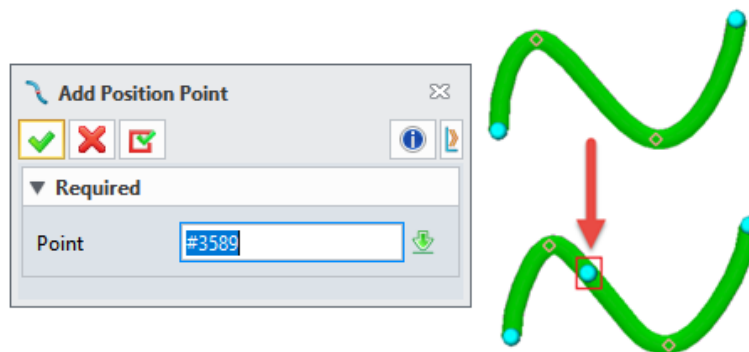
→Where is it

Harness >> Edit >> Detach

3.12.6.5 Add Position Point

Users usually only need execute the above-mentioned operations on part of wires and cables such as Create Protection, Attach, and Add to Protection. Thus, ZW3D 2024 adds the function of “Add Position Point”.

- Position point can be manually added to wire, cable, and protection.
- Protection point can drive wire, cable, and protection.
- Position point interval can cut a harness wire into several harness segments which can be selected alone (harness segment plays as a part of harness).

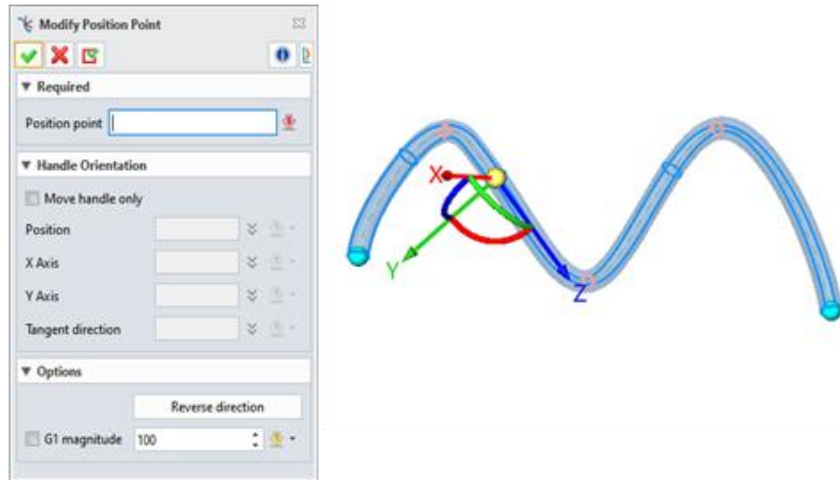


→Where is it

Harness >> Edit >> Add Position Point

3.12.6.6 New Modify Position Point

With the additional modifying position point, ZW3D 2024 can adjust the axial direction and rotation direction of the position point.

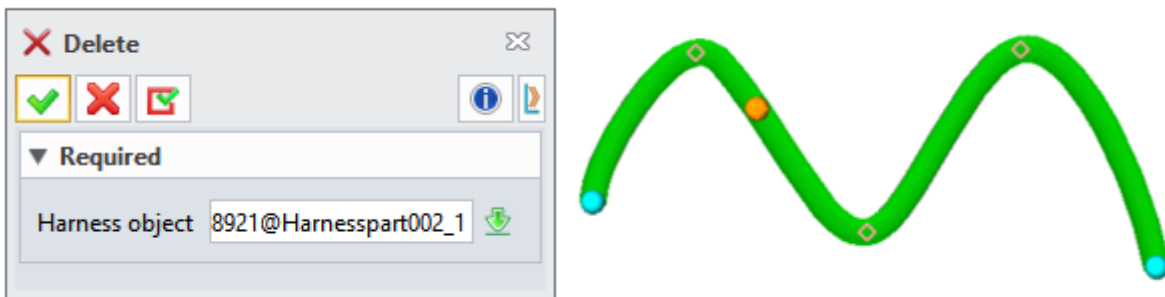


→Where is it

Harness >> Edit >> Add Position Point

3.12.6.7 Deletion Improvement

ZW3D can only delete wires and cables while ZW3D 2024 supports deleting protection and manually added position point.

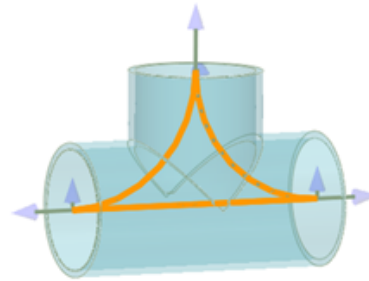
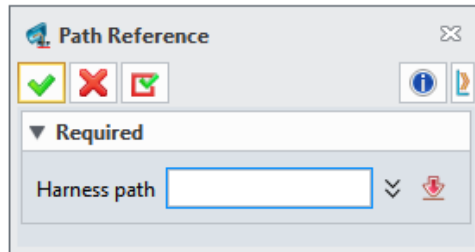


→Where is it

Harness >> Edit >> Delete

3.12.6.8 New Path Reference

For leading path of lead part, during the harness drawing, the path also needs reference to the harness part environment. Thus, ZW3D 2024 adds “Path Reference”.



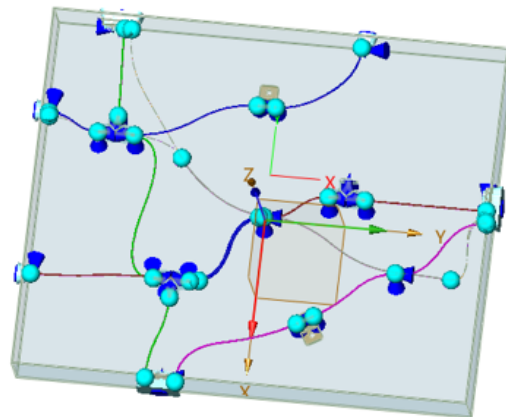
→Where is it

Harness >> Reference >> Path Reference

3.12.6.9 Harness Object Manager

The object manager in ZW3D 2024 can display the created wire and cable object but also display the created protection. In addition, the object manager adds the option of “Length” which can display the created wire and cable and the protection length for user checking.

Name	Length
▼ Harness Object	
▼ Standard Part	
> Connector	
> Fixture	
> Leading Part	
▼ Object	
▼ Wires	
Wire1 1	773
Wire2 1	894
Wire3 1	1106
Wire4 1	811
▼ Cables	
▼ Cable1 1	
Core1	1188
Core2	1190
▼ Bundles	
Insulating Sleeve1 1	204



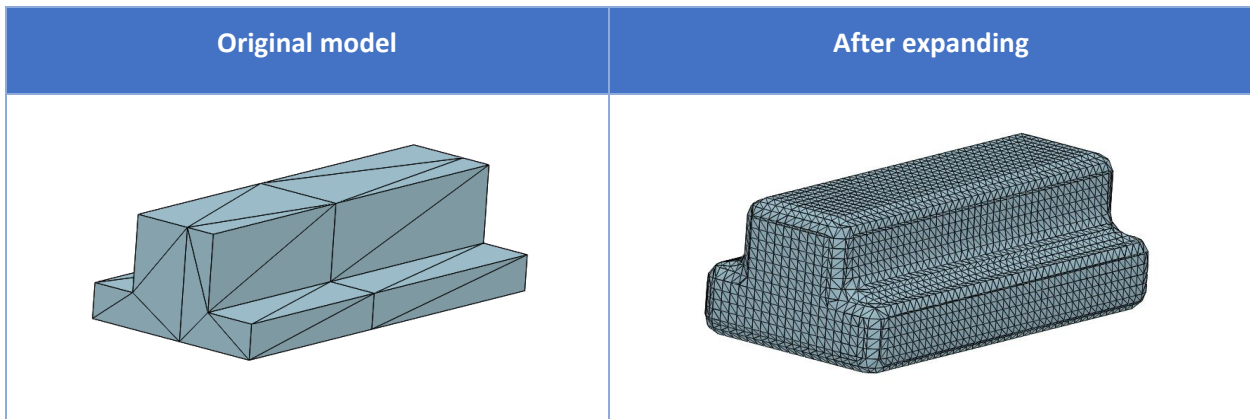
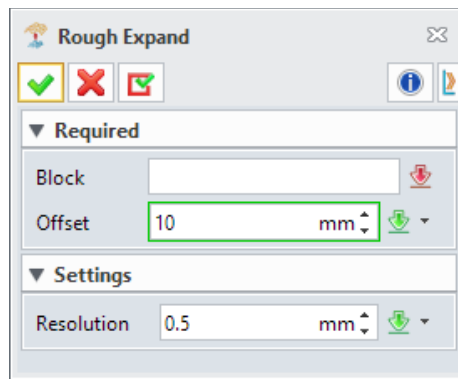
→Where is it

Harness >> Manger >> Harness Object Manager

3.13 Point Cloud

3.13.1 New Rough Expand

Rough Expand command supports generating expansion grid model according to the specified offset and resolution. The result includes smoothing processing of discontinuous edges.



→ Where is it

Part/Assembly >> Point Cloud >> Edit Mesh >> Rough Expand

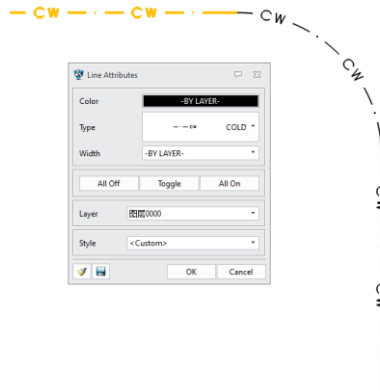
3.14 2D Drawing Sheet

3.14.1 ★New Complex Line Type

ZW3D expands the line type in drawing sheet model. New complex line type allows inserting texts and graphics.

3.14.1.1 Embedded Line Type of Text

Lines, arcs, and other objects can be set as complex lines, which support embedded text. The case below is the commonly used “Cold_Water_Supply” line type in piping industry.

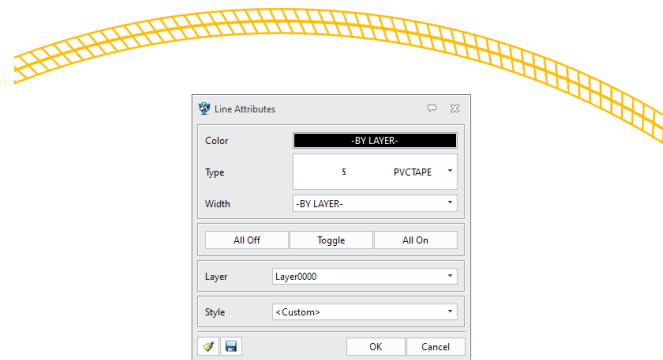


→ Where it is

Drawing Sheet Environment >> Select an Object >> Context Menu >> Attributes >> Lines

3.14.1.2 Embedded Line Type of Graphics

Lines, arcs, and other objects can be set to complex lines, which support embedded graphics. The case below is the commonly used Full_PVC_Tape line type in wiring harness industry.

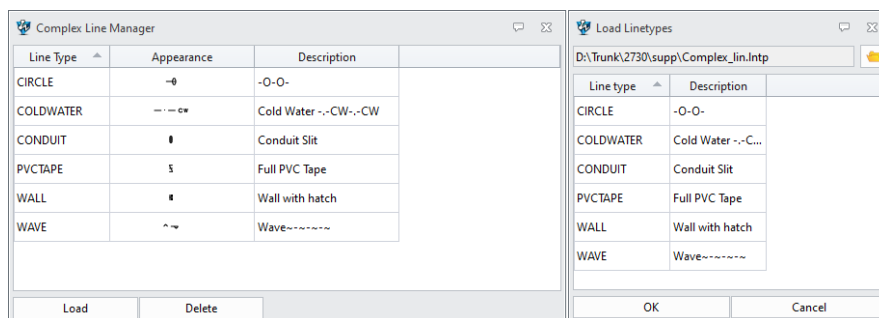


→ Where it is

Drawing Sheet Environment >> Select an Object >> Context Menu >> Attributes >> Lines

3.14.1.3 New Complex Line Manager

Complex Line Manager can read user-defined line type files and load complex line type. Complex Line Manager can list out the name, appearance, and description of each line type. Besides, the loaded complex line type will be saved in the drawing sheet to avoid line type information missing during file transfer.



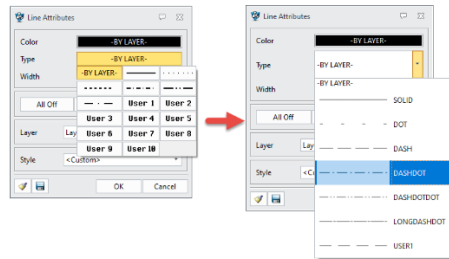
→ Where it is

Drawing Sheet Environment >> Tools >> Attributes >> Complex Line Manager

3.14.1.4 UI Improvement in Line Attributes

The line type in line attributes is upgraded to the drop-down menu which can display both line type name and appearance and make it easy to use. In addition, undefined simple line types are deleted to

shorten the list.

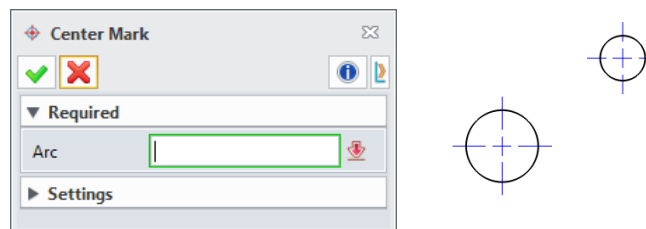


→ Where it is

Drawing Sheet Environment >> Select an Object >> Context Menu >> Set Line Attributes

3.14.2 Self-Adaptability of Center Line in Drawing Sheet

In drawing sheet, when dimensioning hole center mark with Center Mark command, the system would automatically adjust the center line length according to the hole size to avoid disparity between center line and hole.

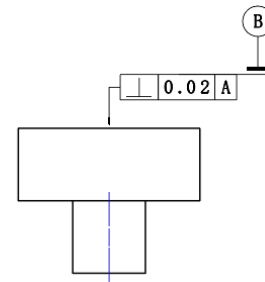
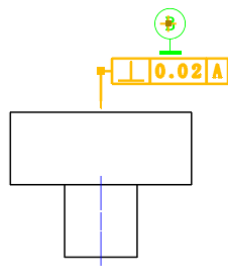


→ Where it is

Drawing Sheet Environment >> Dimension >> Annotation >> Center mark

3.14.3 Support Creating Datum Feature in Feature Control

ZW3D supports direct creation of datum features onto the feature control. A datum feature can be created by its Entity field with a picked feature control. The datum feature created with the method can be dragged to move in horizontal or vertical direction along the feature control by mouse.

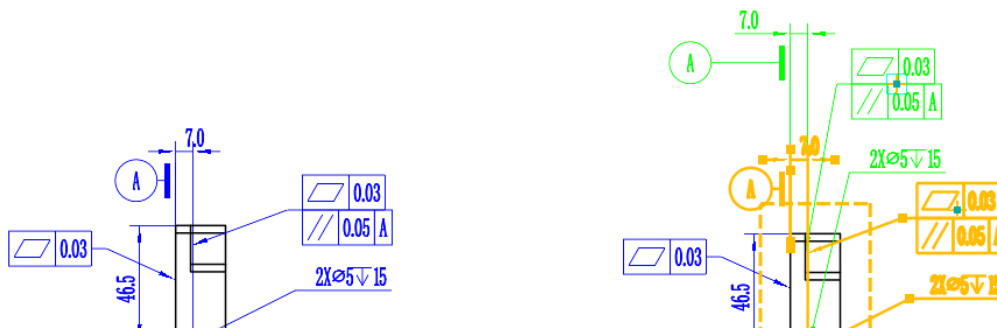


→ Where it is

Drawing Sheet Environment >> Dimension >> Datum Feature

3.14.4 Dragging Picked Dimensions as A Whole

ZW3D supports the whole dragging size dimension. Select feature control symbol, angle, linear, radius and other dimensions, and dragging all to adjust the position.




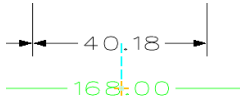

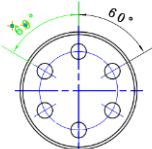
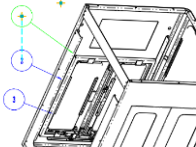
→ Where it is

Drawing Sheet Environment >> Dimension

3.14.5 ★New Dimension Attaching Alignment

We added the function of dimension attaching alignment to solve the problem that dimensioned line is not easily aligned in the drawing sheet. Linear dimension supports three alignment methods, which are aligning by setting value according to dimension line spacing, aligning by text, and aligning by dimensioned line respectively. Alignment between the dimensioned lines of concentric equal radius and alignment

between the balloon are supported by angular dimension.

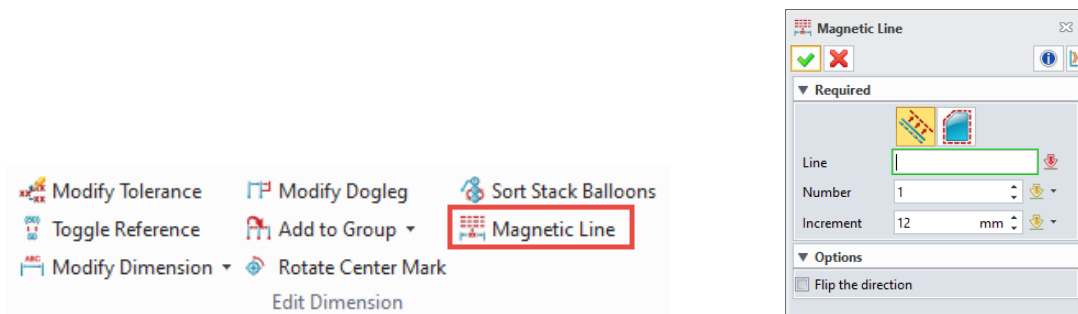
<p>Linear dimension Set value to align by spacing</p>	
<p>Linear dimension Text alignment</p>	
<p>Linear dimension Dimension line alignment</p>	
<p>Angular dimension Dimension line alignment</p>	
<p>Balloon alignment</p>	

→ Where it is

Drawing Sheet Environment >> Dimension

3.14.6 ★New Magnetic Line

To quickly align balloons and dimensions, we added the “Magnetic Line” function.

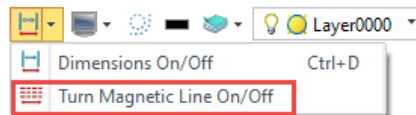


There are two methods to create a magnetic line: based on object and based on view frame. You can

set distance of offset and number of magnetic lines when creating magnetic lines. Magnetic line has an attaching effect on dimensions and balloon symbols to layout them along a straight line. When moving the position of magnetic line, the attached objects will move along with the magnetic line, facilitating the dimension and bubble management.



In the drawing sheet, you can turn on/off magnetic line display in DA toolbar.



→ Where it is

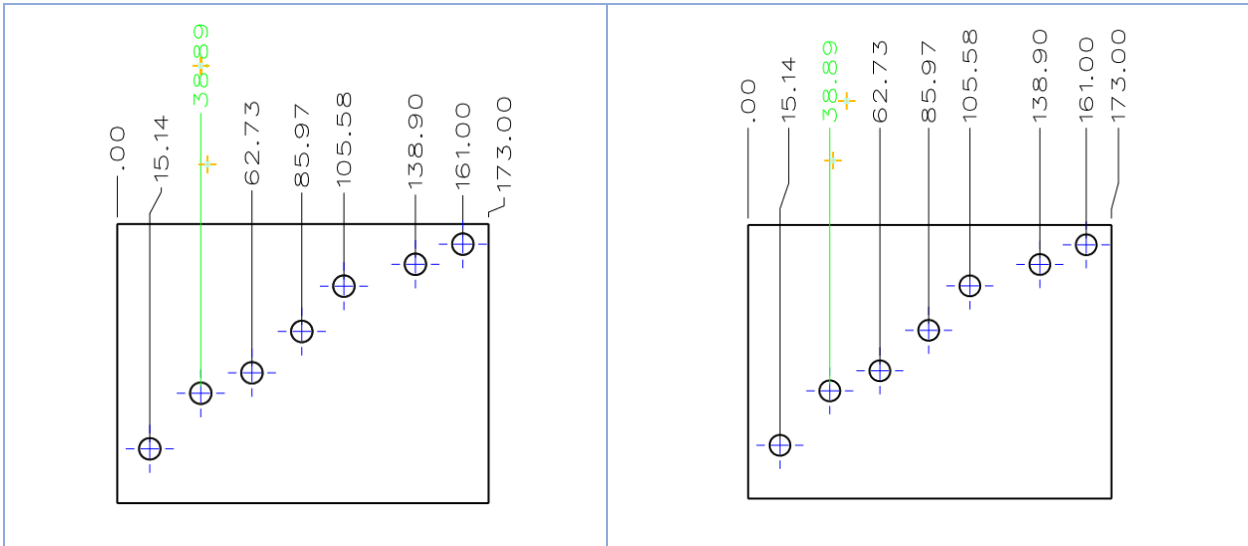
[Drawing Sheet Environment >> Dimension >> Edit Dimension](#)

3.14.7 Group Dimension Improvement

3.14.7.1 New Dimension as Group

ZW3D 2024 adds an option “Dimension as group” in the three commands: Baseline, Continuous, and Ordinate. Use this option to control the dimension result whether can be move as group. When check it, the entire group of dimensions can be moved; when uncheck it, the dimension in group can be moved alone.

Be moved alone	Be moved as a group
----------------	---------------------



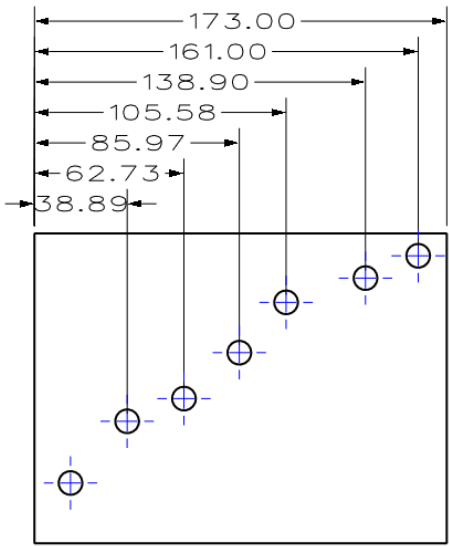
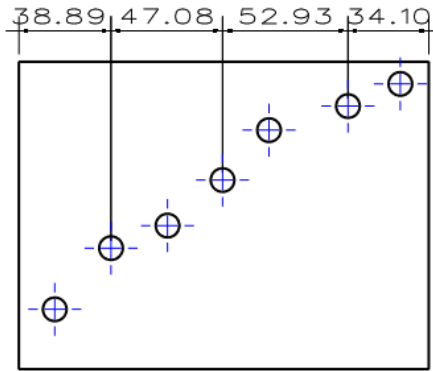
→ Where is it

2D Sheet >> Dimension >> Dimension >> Baseline/Continuous/Ordinate

3.14.7.2 New Smart Sorting in Dimension as Group

ZW3D 2024 improved the sorting of group dimension. When creating group dimension or adding to group, the system will automatically sort the order based on the dimension point to avoid crossing the dimension lines.

Dimension Type	Sorting Rule	Diagram
----------------	--------------	---------

<p>Baseline Dimension</p>	<p>Add baseline dimensions. The system can automatically record the dimension as group (original and new) from small to large so that to avoid crossing dimension lines.</p>	
<p>Continuous Dimension</p>	<p>Create/add continuous dimension. The system can automatically reorder all selected points based on the locations to ensure each dimension is leading out from adjacent dimension point so that to avoid crossing the dimension lines.</p>	

→ Where is it

2D Sheet >> Dimension >> Dimension >> Baseline/Continuous

3.14.7.3 Ordinate Dimension Improvement

1. Ordinate dimension text orientation adjustment

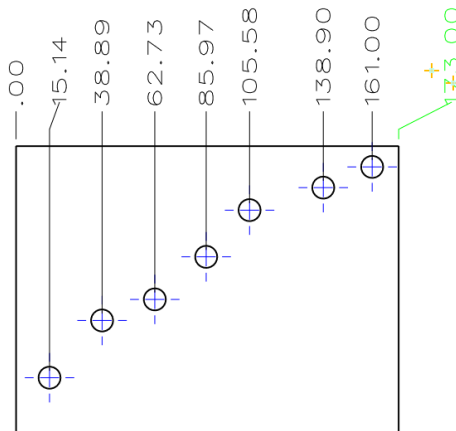
ZW3D 2024 adjusts the text orientation in ordinate dimension. In the standards ANSI, ISO, DIN, GB, and JIS, you can adjust the text along with dimension line.



2. Dogleg interval adjustment improvement

ZW3D 2024 improves the Dogleg interval adjustment in ordinate dimension. The dimension with Dogleg in the ordinate dimension supports spacing adjustment along the four directions of the dimension line direction and the vertical direction of the dimension line.

Support multiple dimensions in the selected dimension and drag overall to move.



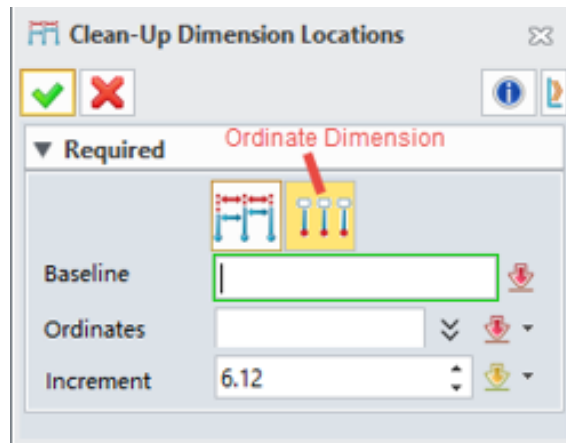
3. Batch processing ordinate dimension interval

ZW3D 2024 adds batch processing ordinate dimension which is used to adjust the intervals between coordinate dimensions.

Baseline: select an ordinate dimension as an adjustment baseline. When adjusting baseline, the placement is unchanged. Any of the coordinate group dimensions can be selected as the baseline.

Ordinates: select an ordinate dimension which is the same group of baselines. This field selects the Ordinate dimension that needs to adjust placement.

Increment: define the ordinate dimensions distance between the two.



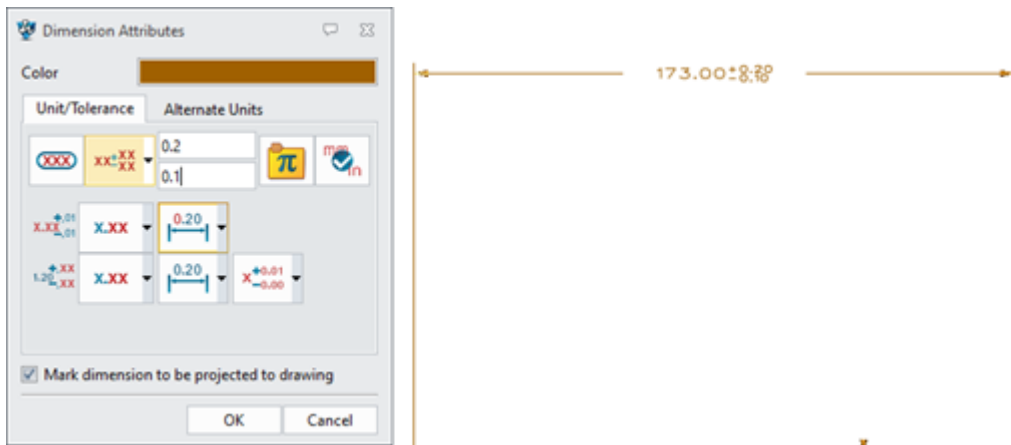
→ Where is it

[2D Sheet](#) >> [Dimension](#) >> [Edit Dimension](#) >> [Clean-up Dimension Locations](#)

3.14.8 New Inherit Tolerance to Drawing Sheet from Sketch/3D Dimension

In general, the mechanical industry and distributed design tasks are assigned from top to bottom by the chief engineer. Upstream designers will assign some cartographic information (such as tolerance, etc.) when defining the framework, and downstream engineers will make drawings according to this information.

ZW3D 2024 supports setting dimension's tolerance information such as tolerance type, decimal place, and unit in the sketch or 3D environments. Check the option "Mark dimension to be projected to drawing" in Dimension Attributes to inherit the dimensions to the drawing sheet while projecting the drawing sheet including inherit tolerance information. After the dimension is inherited to the drawing sheet can keep associated with the sketch/3D environment.



→ Where is it

Part/Assembly/Sketch >> Dimension >> Dimension Context Menu >> Attributes

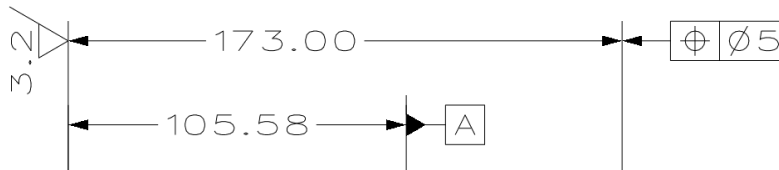
2D Sheet >> Layout >> View >> Standard >> Show dimensions from part

3.14.9 Symbol Improvement

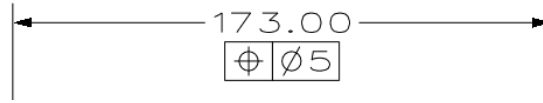
3.14.9.1 Symbol Alignment and Move Improvement

To meet the engineering drawing standard, ZW3D 2024 improves the alignment and movement of drawing sheet symbols. The improvements include:

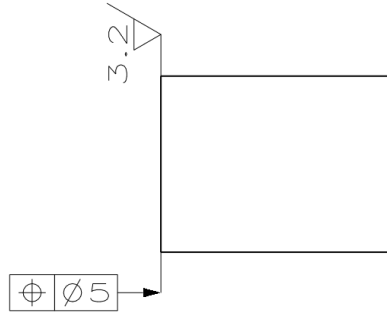
- Feature Control, Datum Feature, and Surface Finish support for capturing the aligned dimension lines.



- After capturing dimension lines, support for synchronously moving with dimension line.
- Feature Control can be created by attaching to dimension. The attached creating feature control can be moved along with the dimension lines.



- Surface Finish and Feature Control can create extension lines.



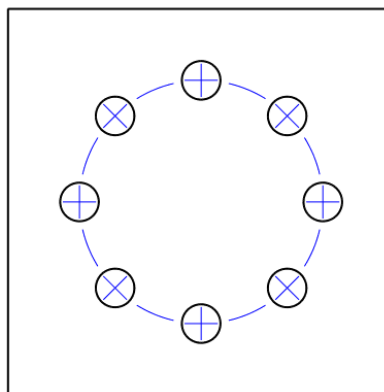
→ Where is it

[2D Sheet >> Dimension >> Annotation >> Feature Control/Datum Feature](#)

[2D Sheet >> Dimension >> Symbol >> Surface Finish](#)

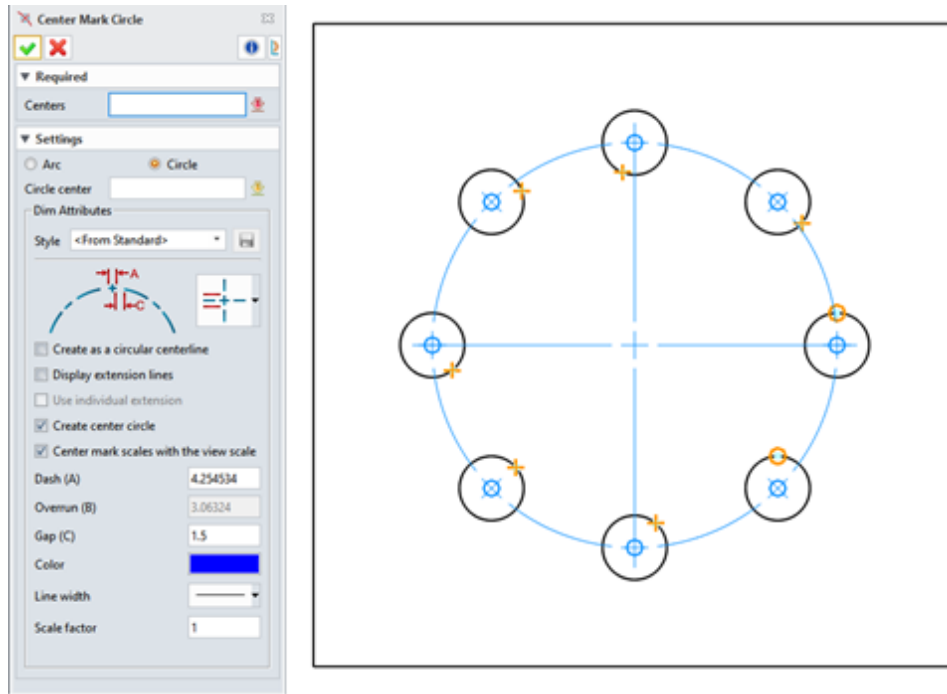
3.14.9.2 Center Mark Circle Improvement

To faster create center mark circle, ZW3D 2024 optimizes the creation logics of center mark when view is projecting. For the hole generated by the circular pattern in 3D trial, the center mark circle is automatically generated when the center line is generated by the project drawing projection.



Center Mark Circle command adds a new option “Create as s circular centerline”. If check it, center

lines will be created at the same time when creating center mark circles.

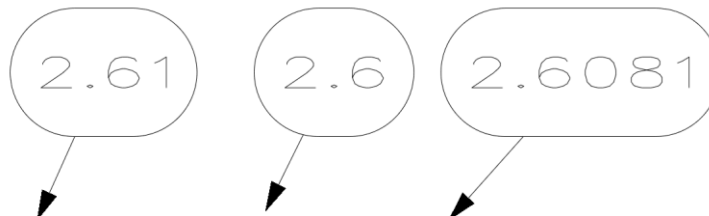


→ Where is it

2D Sheet >> Dimension >> Symbol >> Center Mark Circle

3.14.9.3 New Precision Control in Balloon

ZW3D 2024 adds balloon decimals control which is used to control the display of decimal places for variables of number types in a balloon. The precision field can only take effect to the physical properties, such as quality, volume, total mass, density, size, length, width, height, area, and stock size.

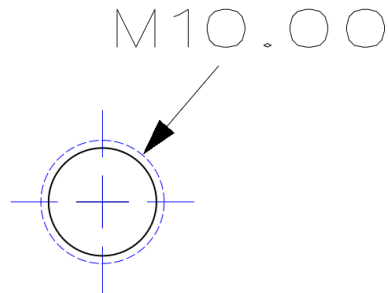


→ Where is it

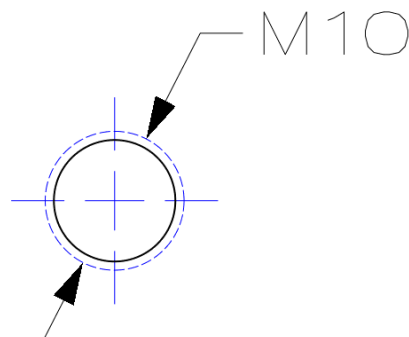
2D Sheet >> Dimension >> Annotation >> Balloon

3.14.10 Hole Callout Improvement

- Dimension thread line displays thread mark M.



- Hole callout adds the second arrow style.



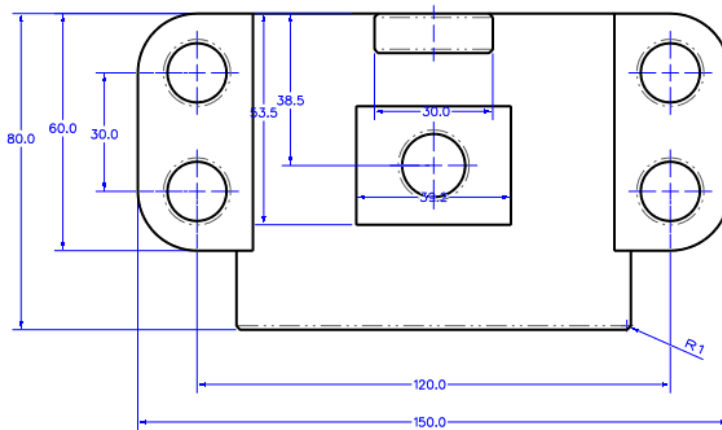
→ Where is it

[2D Sheet >> Dimension >> Hole Callout](#)

3.14.11 Text Command Improvement

1. Support referencing dimension.

Text supports referencing the drawing sheet dimension when the associated dimension can be displayed in text object. When writing technical requirement, you can use it to illustrate the specified dimensions. This function is used in the following case, and no technical requirement needs to modify when the fillet parameters of model are changed.



Technical Requirement
1. Unmarked fillet R1.

2. Support inserting hyperlink.

Text supports adding hyperlink to make the whole text objects become hypertext link which can insert file path (including relative path) or URL. When the mouse hangs over the text object with inserting hypertext link will show the hyperlink contents. Ctrl+ Mouse click can link to the corresponding file or URL.

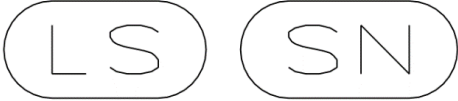



→ Where is it

2D Sheet >> Drawing >> Drawing >> Text

3.14.12 Expand Self-defined Character Type and Special Character Quantity

ZW3D 2024 supports waist shape and hexagon as border for self-defined characters and adds 53 special characters.

Effect of self-defined characters in waist shape	Effect of self-defined characters in hexagon
	



→ Where is it

[2D Sheet](#) >> [Dimension](#) >> [Annotation](#) >> [Feature Control](#)

3.14.13 General Table Function Improvement

1. Support splitting cell

The drawings sheet table supports splitting into cells in ZW3D 2024, which can be used in the same column to fill in different contents.

No.	Wire	Part Number	Color		Length(mm)
1	SW001	E0400019	BR		120
2	SW002	E0400023	BK	RD	340
3	SW003	E0400032	YE		100
4	SW004	E0400072	BK	WH	80
5	SW005	E0400089	GY		120

2. Support filling in the cell's background color.

The drawing sheet table supports filling in the cell's background color in ZW3D 2024, which can directly imply each part color in the list and make table look beautiful.

No.	Wire	Part Number	Color	Length(mm)
1	SW001	E0400019	BR	120
2	SW002	E0400023	BK RD	340
3	SW003	E0400032	YE	100
4	SW004	E0400072	BK WH	80
5	SW005	E0400089	GY	120

3. Support inserting table name.

Add "Sheet_tablename" variable to the drawing sheet's variable manager. When the variable is inserted to the cell of table, the table name will be displayed. This function is used to make a table with a title.

Wire BOM				
No.	Wire	Part Number	Color	Length(mm)
1	SW001	E0400019	BR	120
2	SW002	E0400023	BK RD	340
3	SW003	E0400032	YE	100
4	SW004	E0400072	BK WH	80
5	SW005	E0400089	GY	120

→ Where is it

2D Sheet >> Table >> Text edit >> Insert system variable

3.14.14 Hole Table Improvement

1. Support combing the same ID and counting quantity.

Hole table adds an option "Combine same ID" so that the same size holes can use the same ID. Hole attributes add an option "Quantity" to support counting the hole quantity on each row.

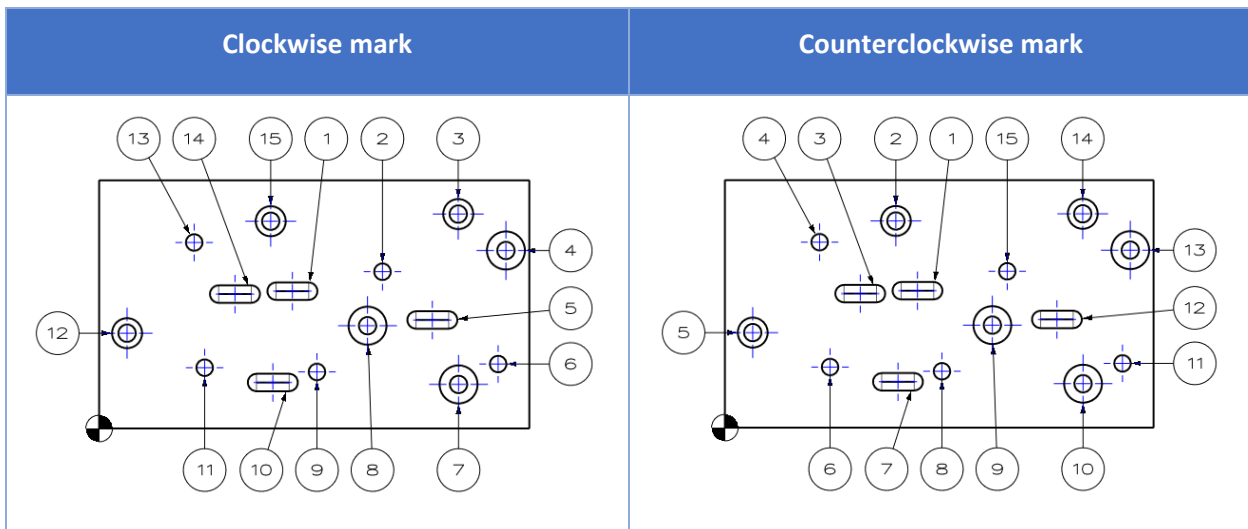
Check "Combine same ID" (original function)	Check "Combine same ID" and count quantity (new function)
---	---

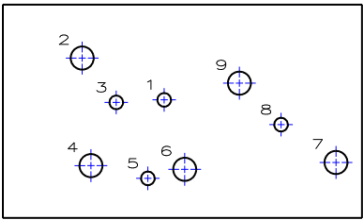
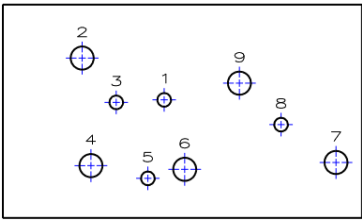
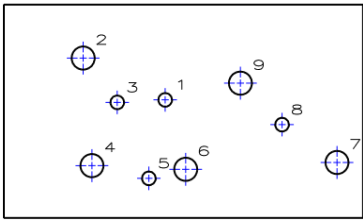
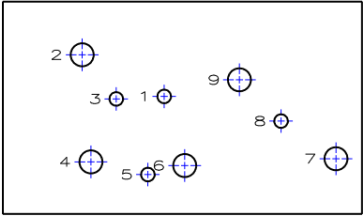
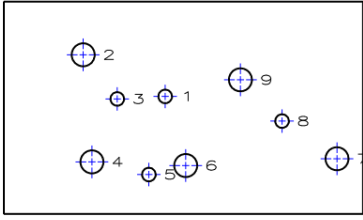
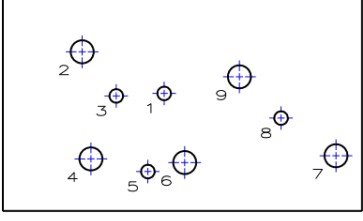
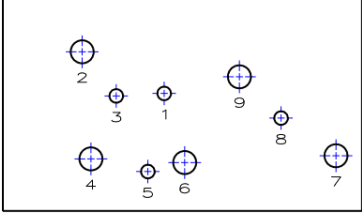
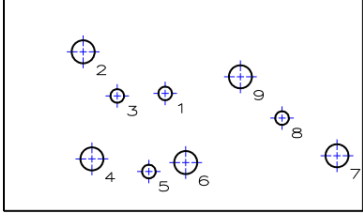
ID	Size	X-center	Y-center	Remarks
A1	∅10.00 √20.00	63.05	36.27	
A2		56.79	110.92	
A3		169.29	93.50	
A4		130.15	33.72	
A5		238.40	38.46	
B1	∅10.50 THRU └┐∅18.00 √10.80	16.69	56.85	
B2		102.52	123.61	
B3		214.55	127.89	
B4		165.89	138.08	
C1	∅10.50 √50.00 ∨∅22.73×90°	243.05	106.16	
C2		214.75	26.29	
C3		160.37	61.31	

ID	Size	Quantity	Remarks
A	∅10.00 √20.00	5	
B	∅10.50 THRU └┐∅18.00 √10.80	4	
C	∅10.50 √50.00 ∨∅22.73×90°	3	

2. Label mark improvement

Label with leader supports marking clockwise or counterclockwise; hole callout without leader supports adjusting the placement of mark.



Top			
Waist		/	
Bottom			

3. Support switching view/table drawing sheet

When a hole table and its associated view are not in the same sheet, you can switch to the associated sheet by this function when redefining the hole table, redefining base point in hole table, or selecting holes from view, etc.

→ [Where is it](#)

[2D Sheet >> Table >> Text edit >> Insert system variable](#)

3.14.15 Add Sheet Code to BOM

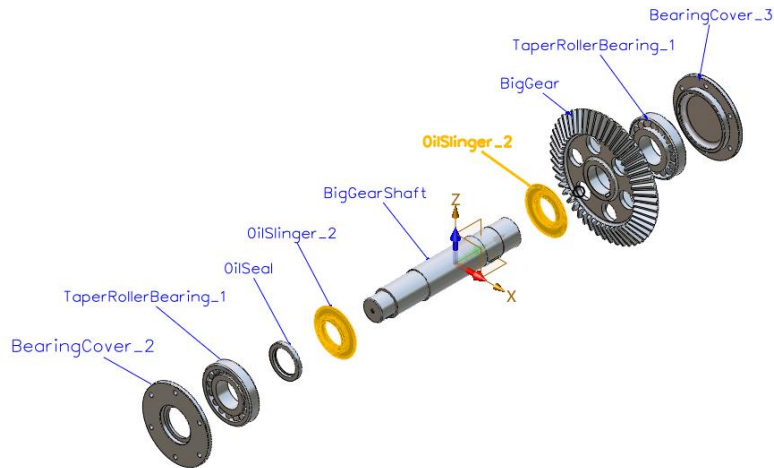
Add an option “Sheet code” to BOM column to support displaying the set sheet code of file attribute in part.

→ [Where is it](#)

[2D Sheet >> Layout >> Table >> BOM](#)

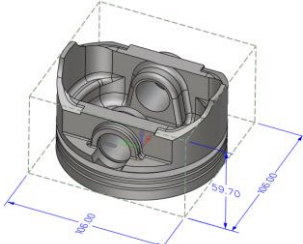
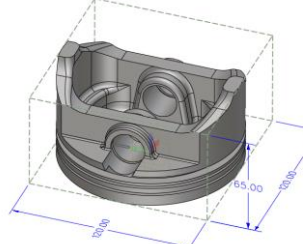
3.15 PMI

With the application of 3D dimension, the 3D model gradually becomes a carrier of processing information and manufacturing information. In the 3D sheets that provided by enterprise to customers, in general, part balloons are annotated to indicate the position relationship of each assembly and other information, so that the overall assembly information can be viewed more clearly.



3.15.1 Bounding Support Adaptive Size Change

In the PMI module, you can use the "Bounding" command to create a dimensioned bounding box that surrounds the active part. The size of bounding box is equal to the size of the part by default. The parameters of the bounding box adaptively changed as the size of part changes.

Before model size updated	After model size updated
	

→Where it is

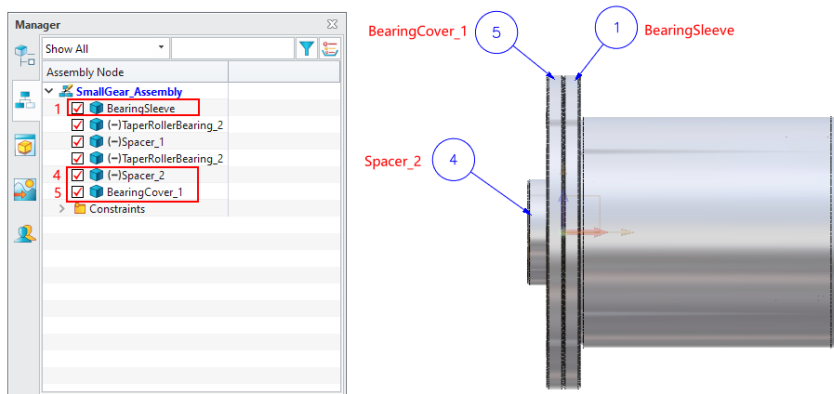
Parts/Assembly Environment >> PMI >> Dimension >> Bounding

3.15.2 New Balloon Dimension

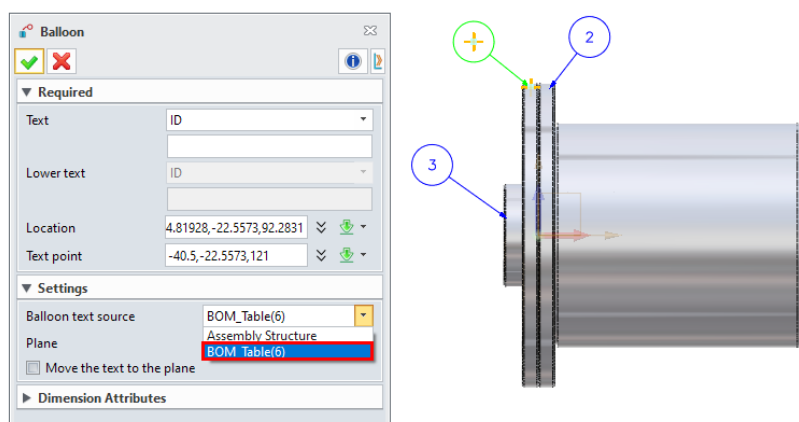
ZW3D 2024 adds balloon dimension in PMI module, which user can create balloon annotation in the 3D environment. The balloon can be used to mark parts in the assembly and be associate parts with ID in BOM.

- 1) The source of balloon can be selected from “assembly” and “BOM”.

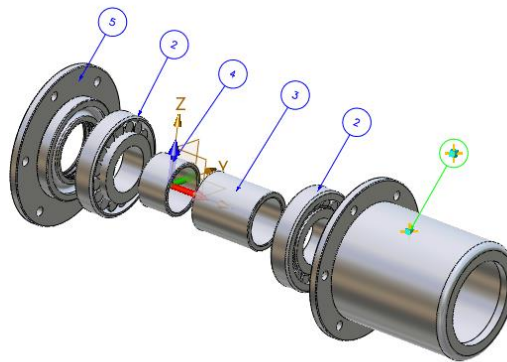
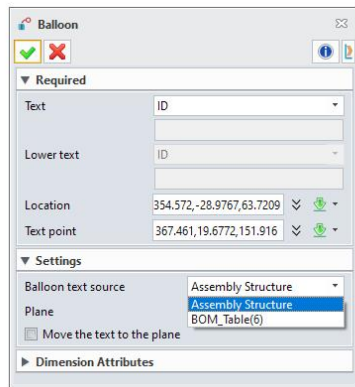
When the assembly has no BOM, the balloon properties source from the activated sheet “Assembly structure” by default.



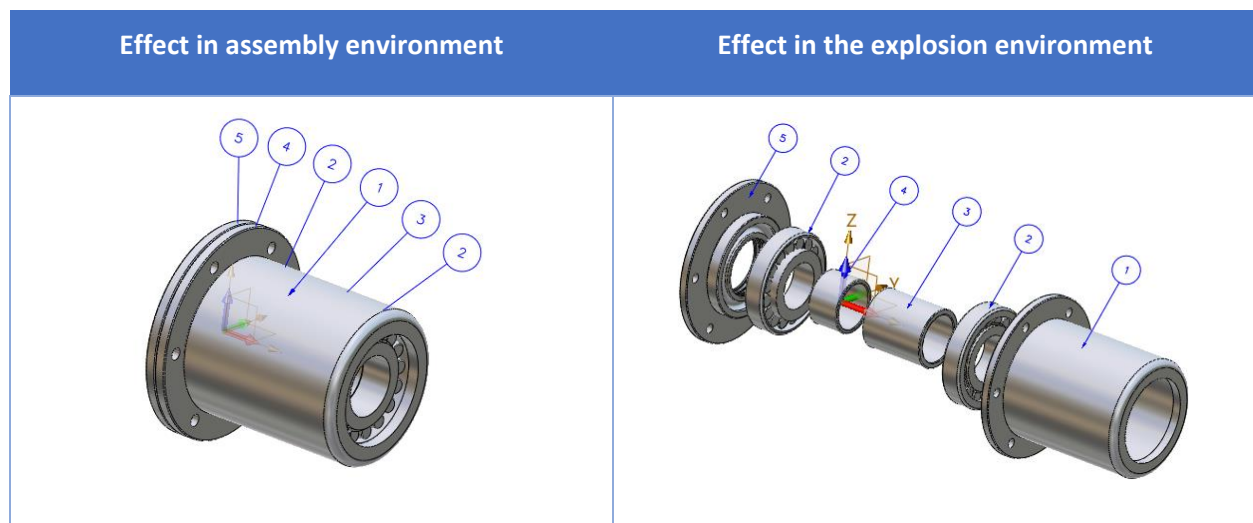
When the activated sheet is inserted to BOM, the balloon sources from BOM and is applied to the attributes in the BOM.



In the exploded view environment, the creation of balloon annotation is also supported.

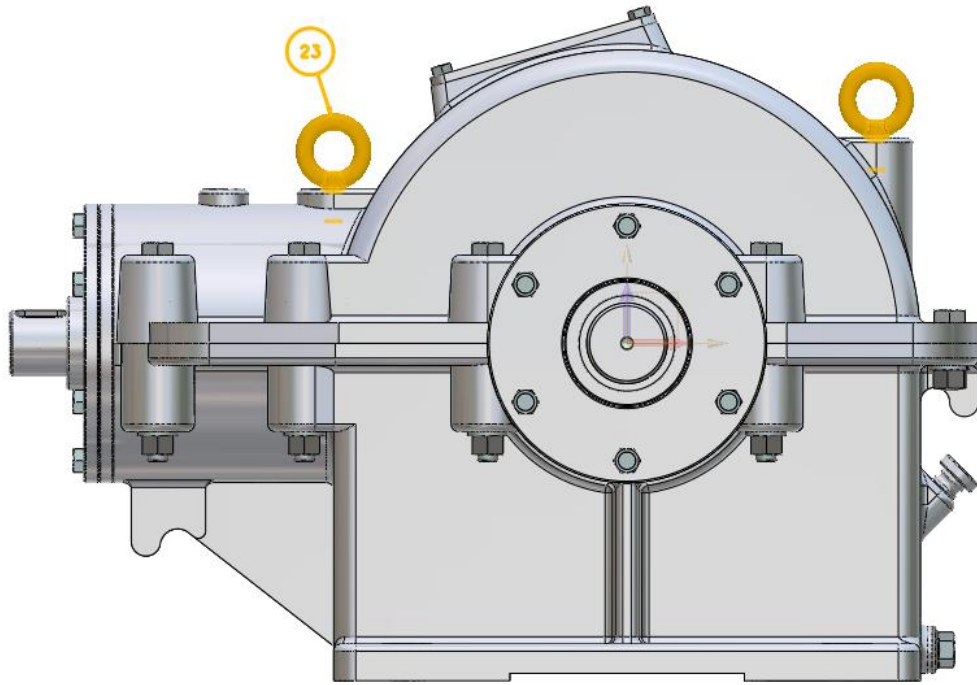


It's noted that when exiting the exploded environment, the balloon can automatically assembly by following the part; on the contrary, when user enters again the assembly view environment, the auto-explosion can be achieved along with the part.



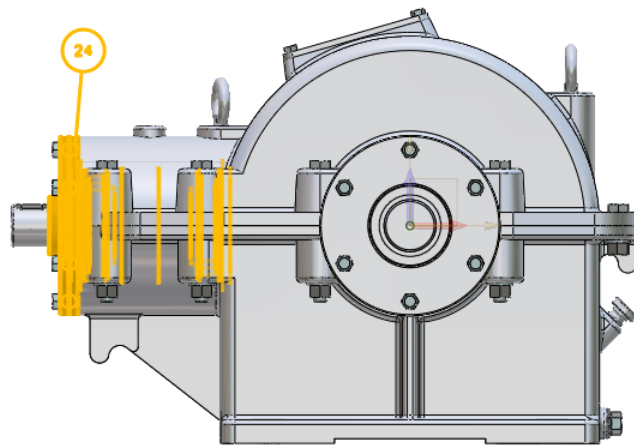
To quickly search the relation between the annotated balloons and the parts, the balloons keep associated with the dimensioned objects. Mouse left clicking the balloon, you can highlight the dimensioned part objects.

Balloons that source from "assembly structure" can highlight the balloon dimensioned part and highlight the same part object.



Balloons that source from “BOM” can highlight according to the BOM structure. For instance, when BOM is created according to “Only top level”, it can highlight the entire sub-assembly.

ID	Name	Cost	Number	Quantity	Material
1	ANSIB18234M_HXFQSW_M16L25			1	Steel
2	ANSIB18235MHB_M14L140			8	Steel
3	ANSIB18235MHB_M14L45			2	Steel
4	ANSIB18241MS1_NUT_M14_T12.8		Hex nut, Style 1, ASME B18.2.4.1M, M14 x 12.8	10	Steel
5	ANSIB1867M1_HXMCSSW_M10L30			18	Steel
6	ANSIB1867M1_HXMCSSW_M8L20			4	Steel
7	Base			1	Aluminum
8	BearingCover_2			1	Aluminum
9	BearingCover_3			1	Aluminum
10	BigGear			1	Aluminum
11	BigGearShaft			1	Aluminum
12	Cover			1	Aluminum
13	EndRing			1	Aluminum
14	FlatKey14X36			1	Aluminum
15	FlatKey18X46			1	Aluminum
16	GB7244_LCK_WSHR_SPLT_M16			10	Aluminum
17	HoleCover			1	Aluminum
18	OilDipstick			1	Aluminum
19	OilSeal			1	Aluminum
20	OilSlinger_1			1	Aluminum
21	OilSlinger_2			2	Aluminum
22	SmallGear			1	Aluminum
23	SmallGearShaft			1	Aluminum
24	SmallGear_Assembly			1	Aluminum
25	SuspensionLoop			2	Aluminum
26	TaperRollerBearing_1			2	Aluminum

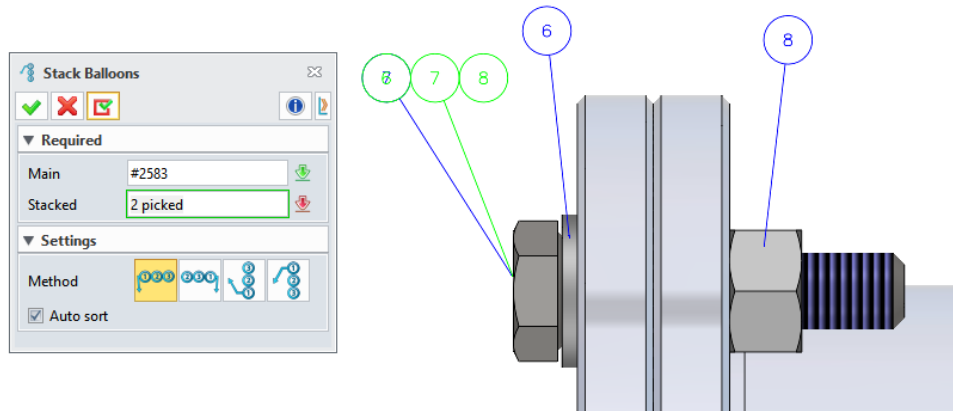


→ Where is it

Part/Assembly/Exploded View >> PMI >> Annotation >> Balloon

3.15.3 New Stack Balloons

PMI adds the function of stack balloons which can stack and sort the 3D balloons. For instance, balloons dimension on bolts and nuts.



→ Where is it

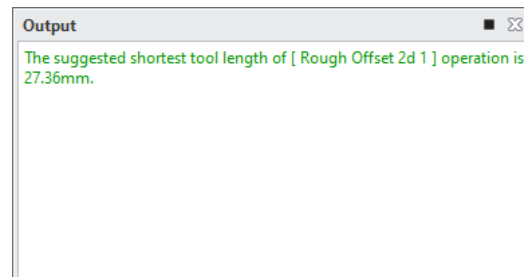
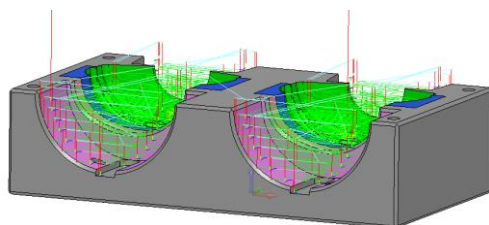
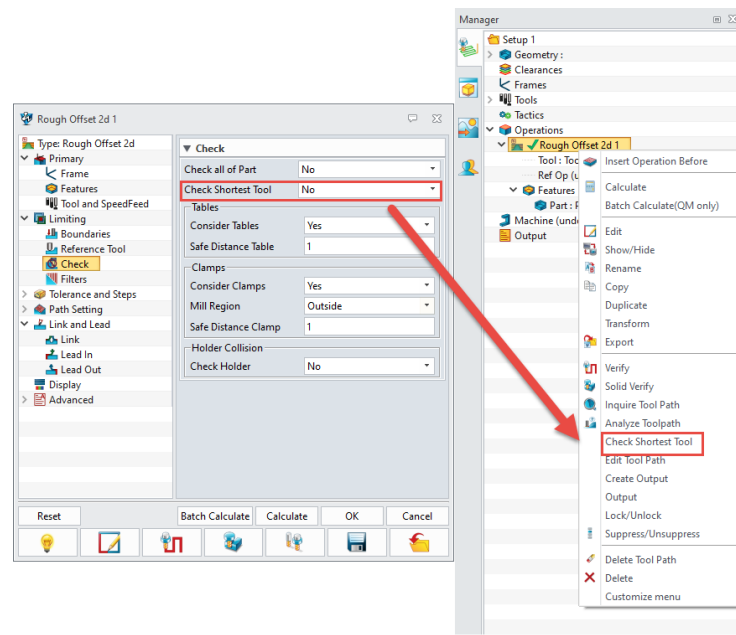
Part/Assembly/Exploded View >> PMI >> Annotation >> Stack Balloons

4 CAM

4.1 ★Independent Shortest Tool Calculation

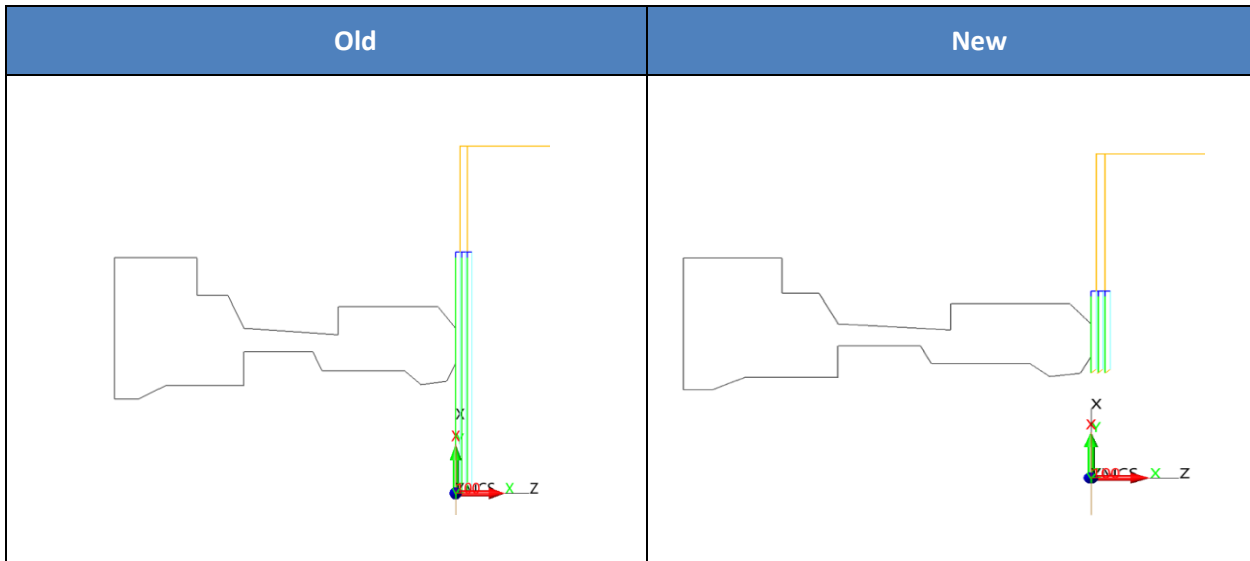
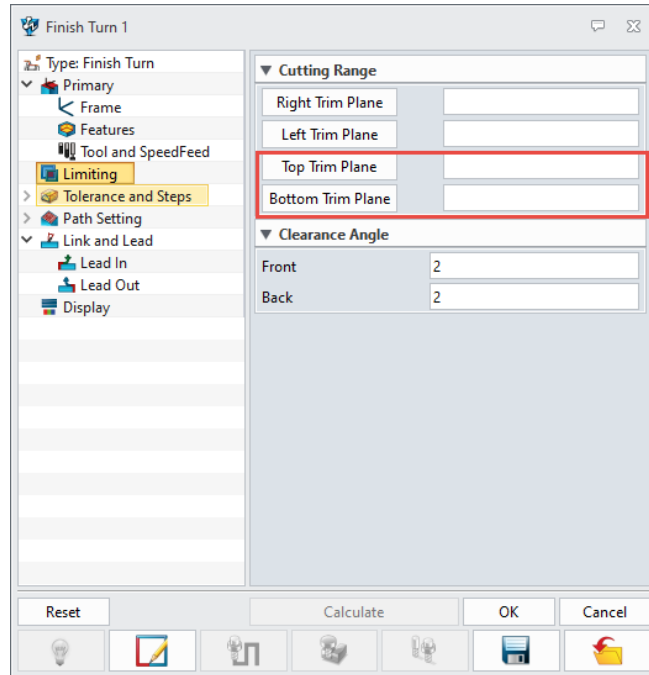
The functionality of “Check Shortest Tool” is moved from the operation parameter to the context menu of the operation as an independent function. When calculating the shortest tool length, it does not need to recalculate the toolpath anymore. We also optimized the inside algorithm which supports calculating the shortest tool length in the rest roughing scene and multiple operations at the same time, and the calculation result becomes more accurate. The calculation results contain 1mm safety clearance value used as a recommended value of the shortest tool length to ensure the machining security.

The functionality of check shortest tool length can only be utilized in QM module at present.



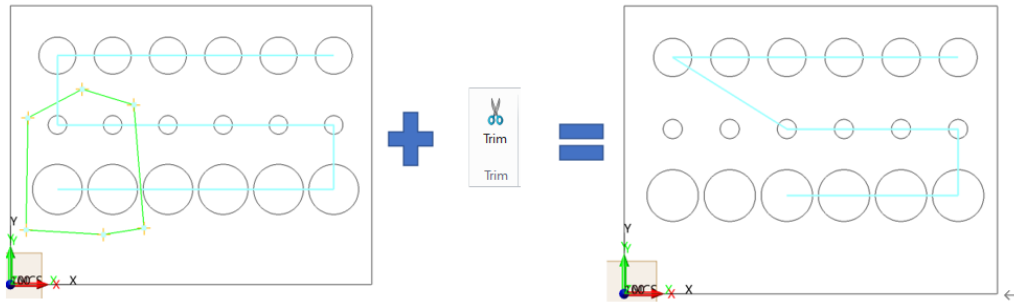
4.2 ★New Limiting in Turning Operation

ZW3D added limiting function in turning operation, which allows user to control the top and bottom range of turning toolpath and improve the machining efficiency.



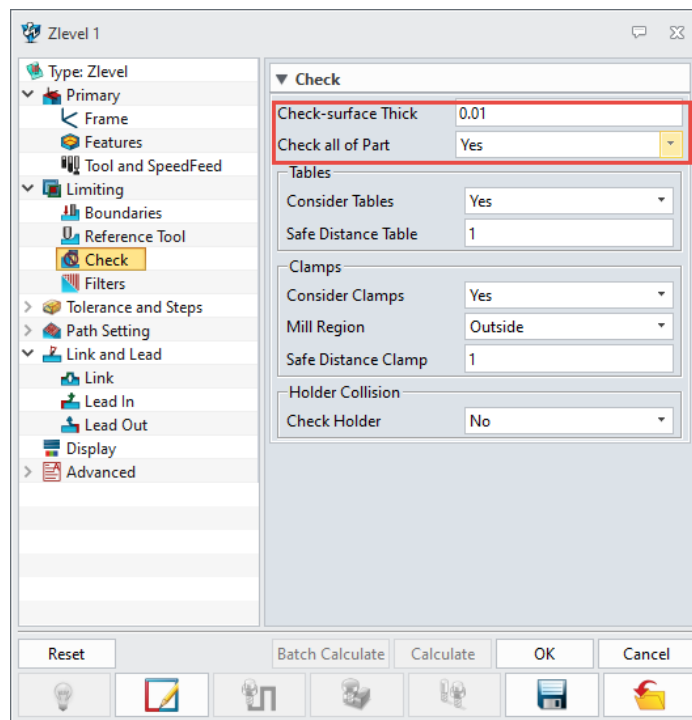
4.3 ★Tool Path Trim Supports Drill Operation

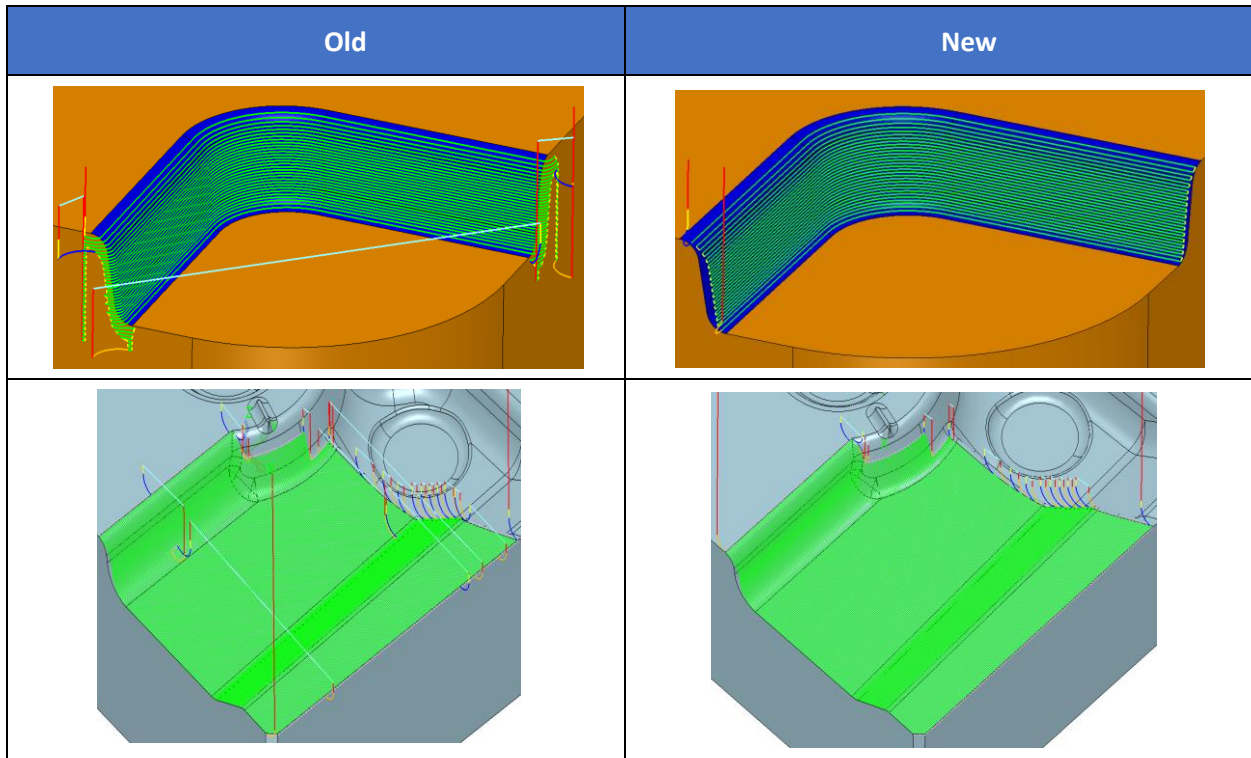
Tool path trim function supports trimming the drill hole toolpath, which is convenient for users to reduce the unnecessary toolpaths.



4.4 Toolpath of Pick Surface Processing Constant Improvement

Toolpath quality of picking surface processing is improved constantly (Create surface feature as operation feature, open “Check all of Part” option and set “Check-surface Thick” as 0.01). The toolpath in Z-level and Lace operations benefit from this, as the comparison shown in the table below:

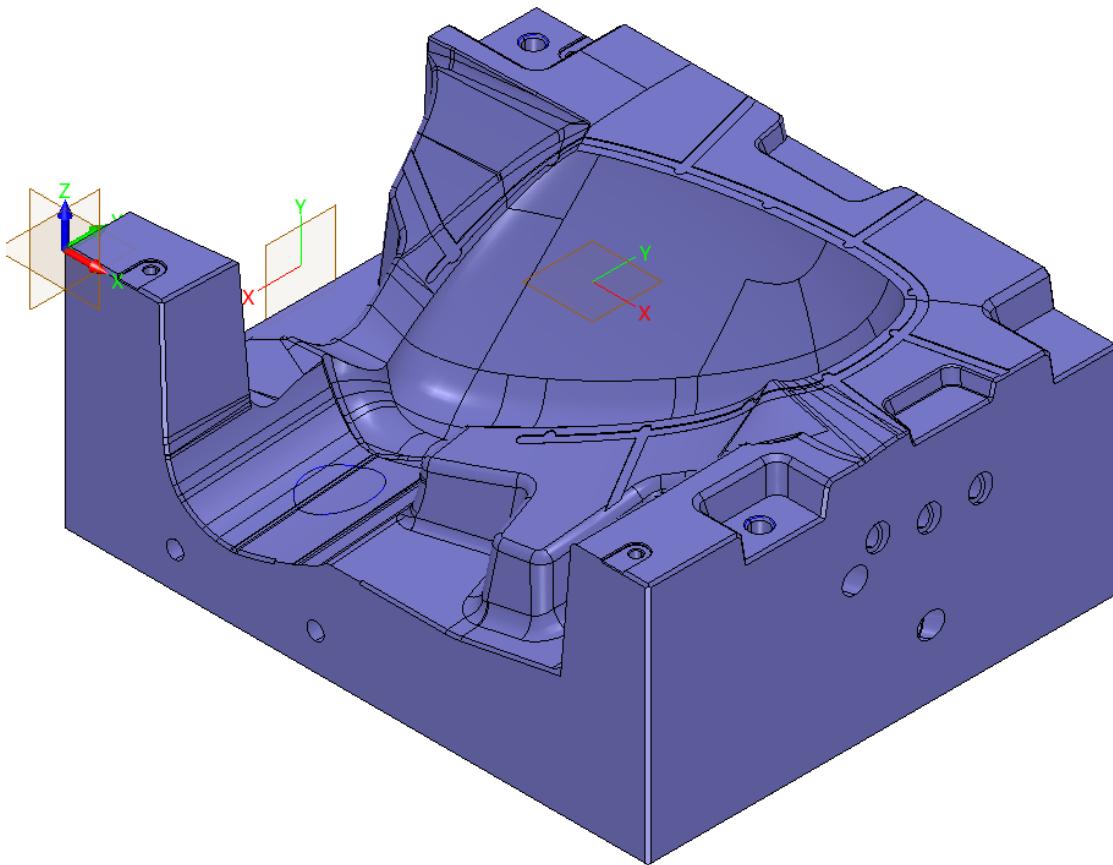




4.5 ★Apply to New Render Engine

In ZW3D 2024, CAM module adapts to the function of new render engine which optimizes the display style of auxiliary points and auxiliary lines. The display efficiency of large assembly components is significantly improved. The software lag issues of opening or close in large and complicated parts have been solved. The display frame (FPS) comparison table is as follows: (part size: 340*275*150).

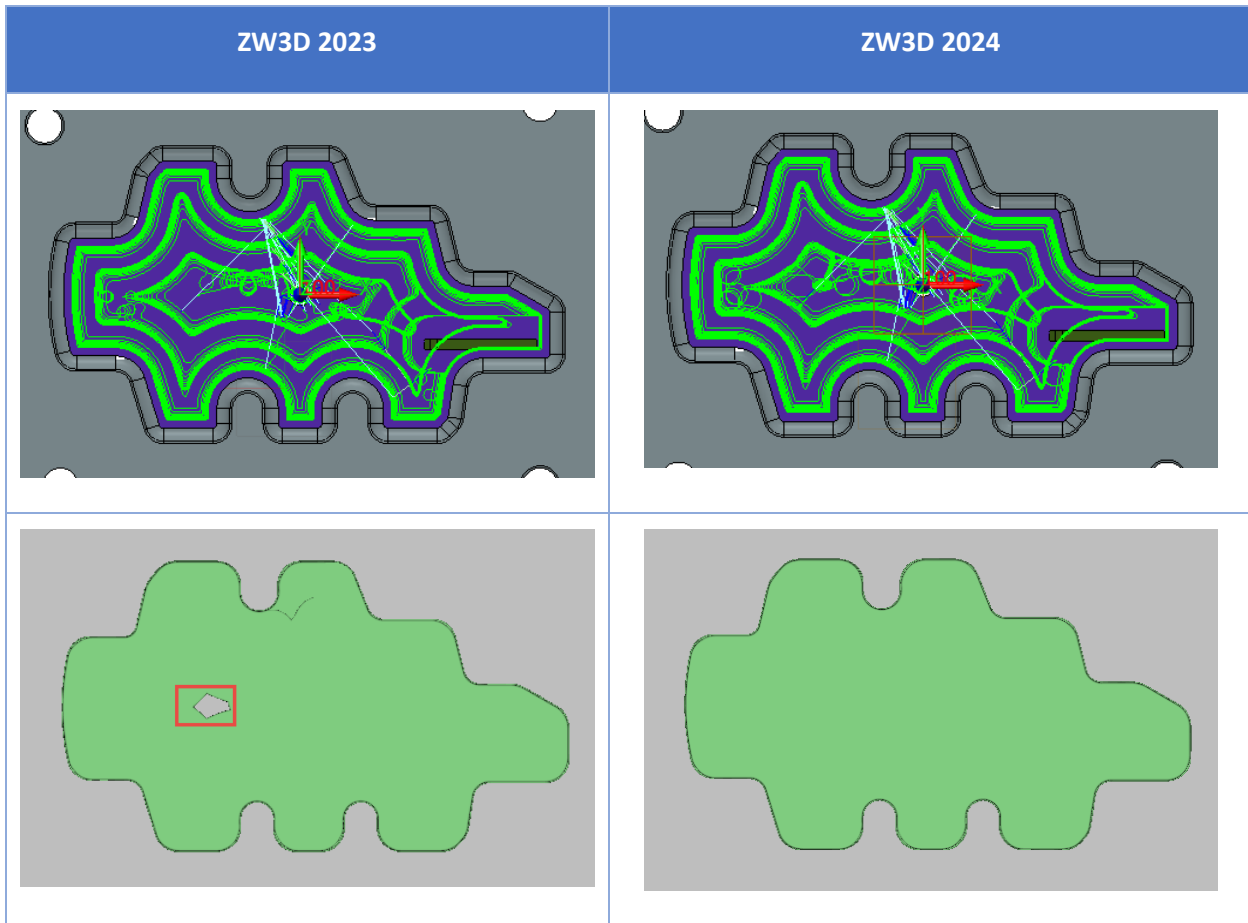
Meanwhile, regulate user's operational flow. Multiple operation parameter boxes are not supported to open at the same time. Ribbon bar cannot be switched more open at will. The creating operation will no longer guide user to select machining feature and tool. User is not allowed to modify parameter or execute other operation in non-main setting interface to reduce operational errors.



Display frame (FPS) in ZW3D 2023	Display frame (FPS) in ZW3D 2024
3.15	113.3

4.6 ★QM Step Max Roughing

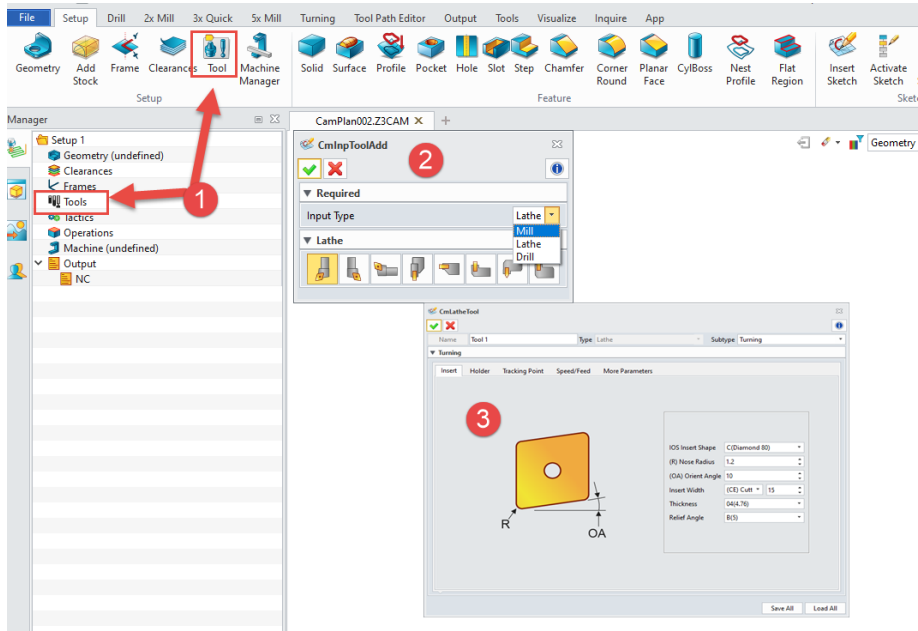
ZW3D 2024 solves the problems of missing machining of the tool path when user sets the step max distance of QM operation roughing. At the corner control, the original D-LOOP tool path style is deleted. When the user selects Smooth, the software will automatically add “task” or “slot” tool path in the detection residual area, making the tool path cleaner and eliminating the abnormal leakage operation. At the same time, the restriction that the step distance must be less than 90% of the tool straightening to improve the operation efficiency, so that ZW3D can meet more application scenarios.



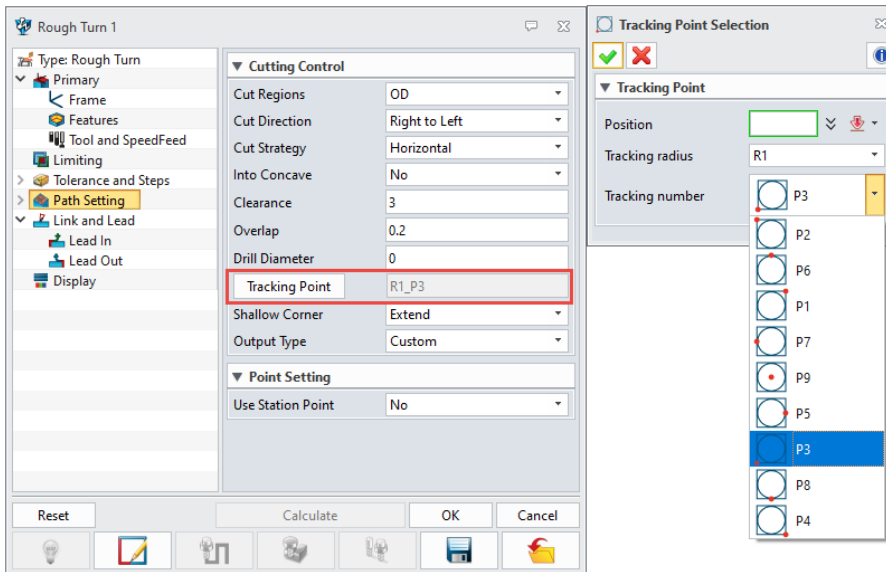
4.7 ★Whole Workflow Improvement of Lathe Tool

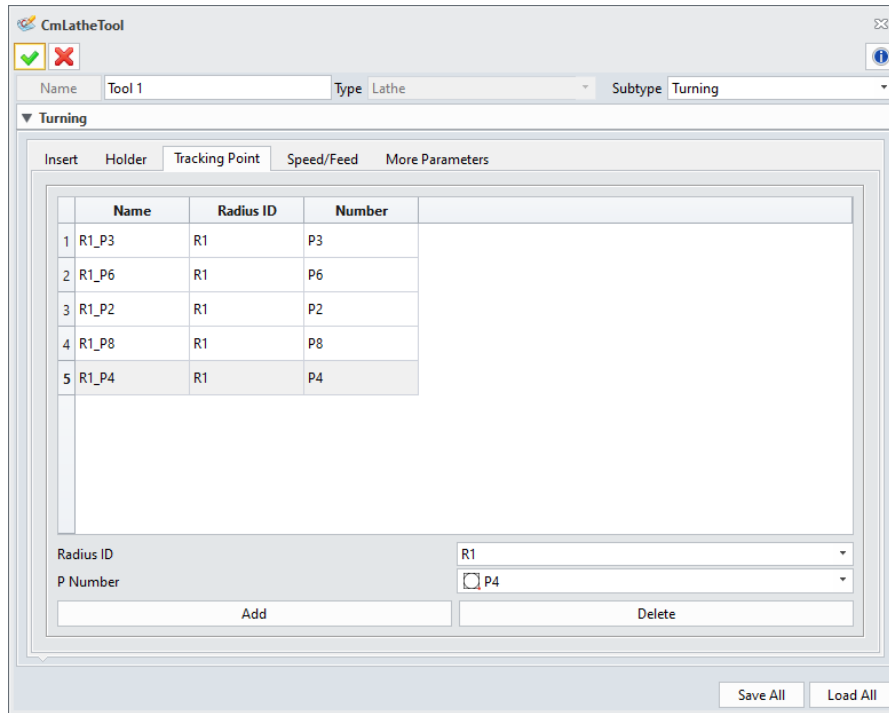
The lathe tool has made a huge improvement in ZW3D 2024, which makes a big change in workflow and insert is set independently and separately from shank. Standardize the parameters and greatly improve the ease to create lathe tool.

Lathe tool creation workflow improvements: when user creates a tool form Ribbon bar or CAM tree, it will pop-up “Create Tool” interface where user can switch to different tool type according to realistic process. After selecting tool, user can select more detailed tool type in “Lathe” bar and click OK to enter the tool parameter interface. The parameter setting interface of insert and shank are independent. All insert types and shank types are using ISO standards. User can set details about insert, shank, tracking point, and more parameters.

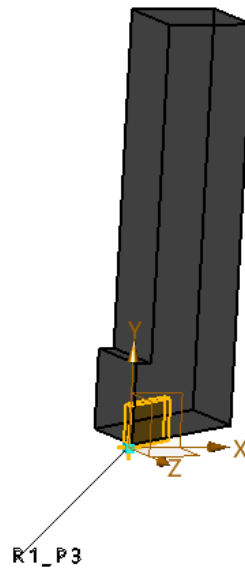


Tracking point improvement: In ZW3D 2024, both tool and operation add “Tracking Point” option. For tool, user can add multiple tracking point according to the actual operation requirement. In the specific operation, users can independently select a tracking point created in the tool as the tracking point generated by the tool path in this operation. For grooving, support selecting three different radius ID in the tool creation interface, which are R1, R2, and R3, separately. They are corresponding to left, middle, and right of the tool.



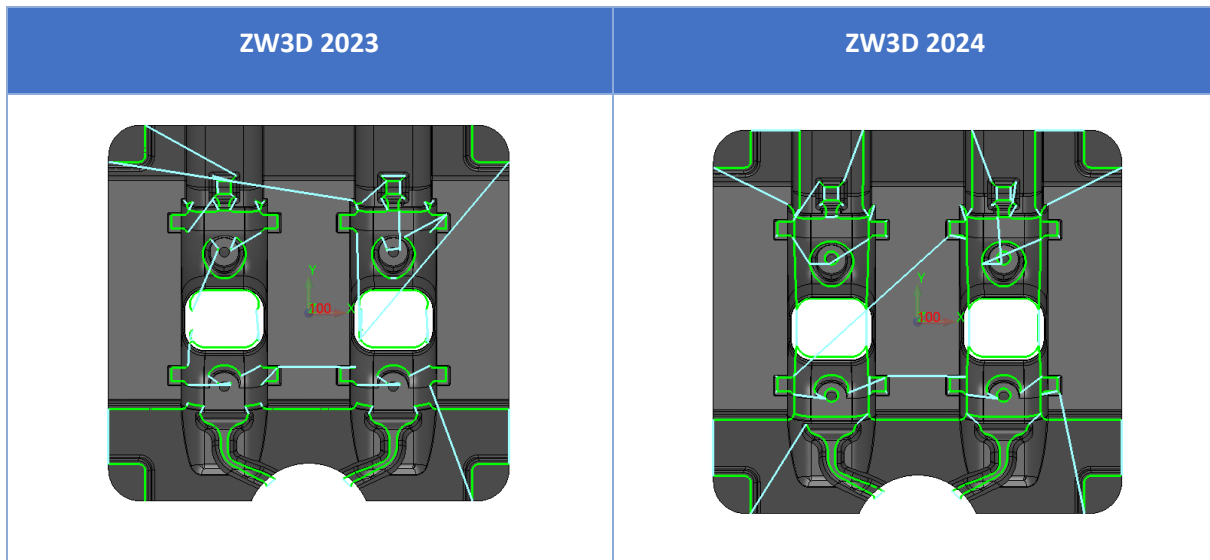


Tool preview improvement: ZW3D 2024 adds the real-time display of 3D preview and tracking point of lathe tool in CAM working interface. The 3D preview map and tracking point are update in real time as the changes in user's setting parameter. At the same time, 3D preview function is applied to wireframe simulation and entity simulation, which greatly improves user experience.



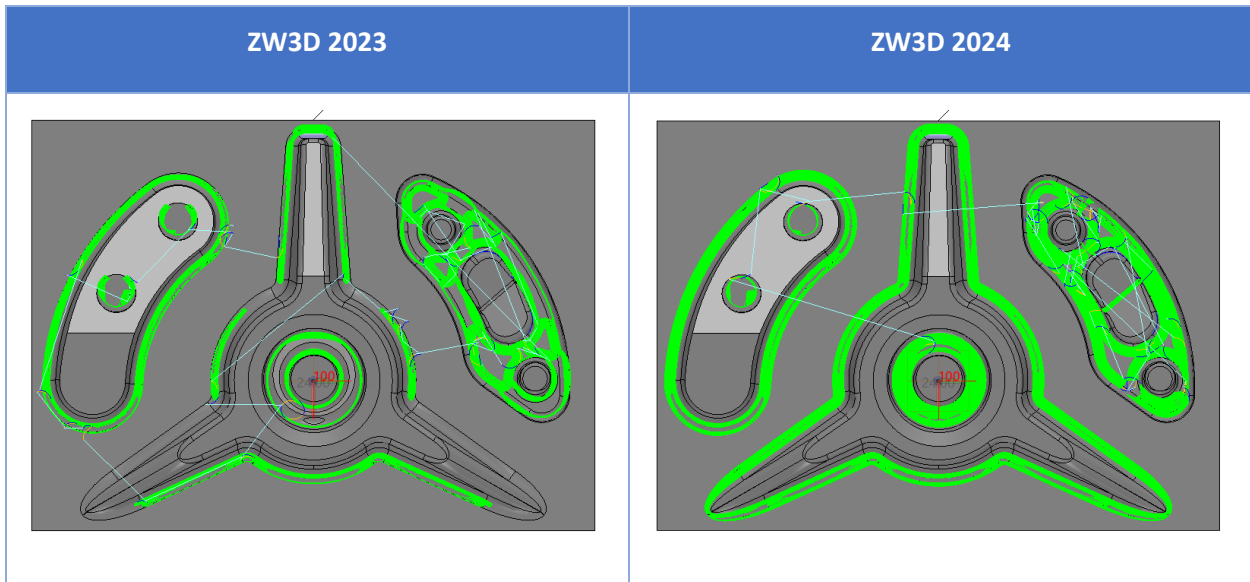
4.8 ★Pencil Finish Improvement

ZW3D 2024 optimizes and improves the root cleaning process, improves the ability to identify the root cleaning line, makes the tool path of pencil type root cleaning process more complete, solves the missing processing caused by the root cleaning position cannot be found, improves the computational efficiency by more than 38%, and indirectly improves the processing efficiency.



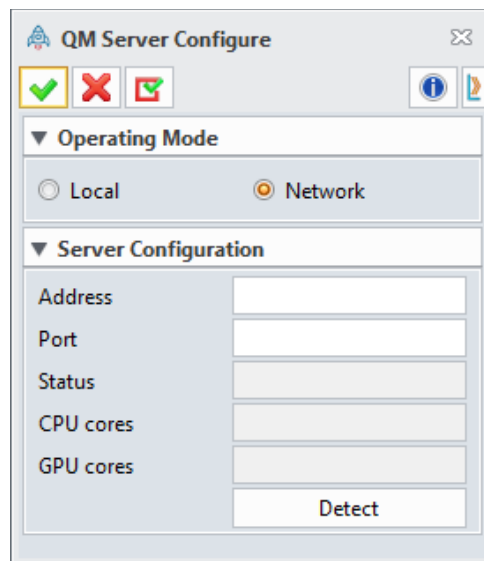
4.9 ★Corner Finish Improvement

Corner Finish cutting operation is optimized from the internal algorithm in ZW3D 2024, which not only improves the accuracy of machining area identification, but also provides two different cutting methods of "along" and "across" to meet more machining scenarios for users. It also improves the connection between steep and non-steep areas, eliminates incomplete machining and link confusion at the tool joint, and reduces the overall advance and fall of the tool path. More uniform step, more beautiful, both functional and visual to meet customer requirements.



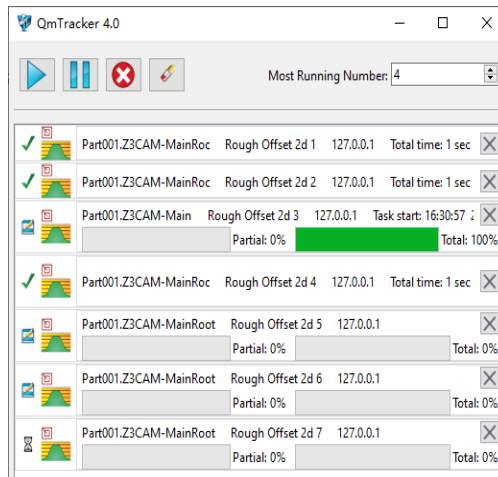
4.10 ★New QM Cluster Computing Function

ZW3D 2024 breaks the limitation of operation editing and computing a single host work by adding a new cluster computing framework to achieve the QM module in LAN cross - host computing and computing task management. User can select local computing or remote server computing through new “QM Server Configure”.

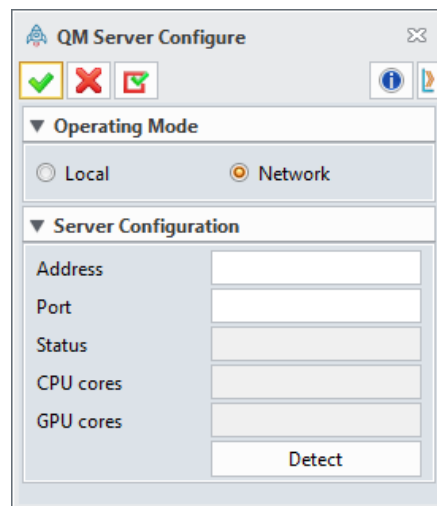


When users select local computing, they can check the status of calculation in the real time via

“QMTracker” interface. At the same time, single or multiple operation calculation are supported to pause or delete. During the local computing process, it supports users’ other operation, but the background computing on the local computer consumes CPU or GPU resources on the local computer, which may cause the machine stutter or delay.

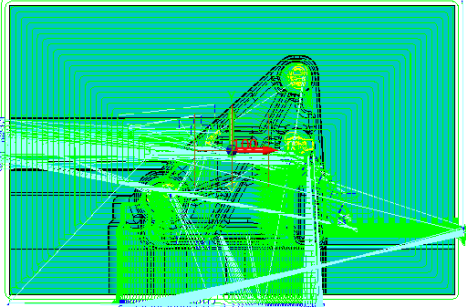


When users select network computing, they can configure server IP address and port and detect whether the server is available. After the server is configured successfully, the uploading toolpaths to be calculated will be computed on the server according to the uploading order. At the same time, users can check the calculation process of server operation in the client “QMTracker” interface. When the server completes toolpath calculation, click “QM Batch Import”, client can automatically download toolpath file to the client, which can eliminate the lag problem caused by the background computing and reduce the waiting time of operation and indirectly improve the efficiency of programming.



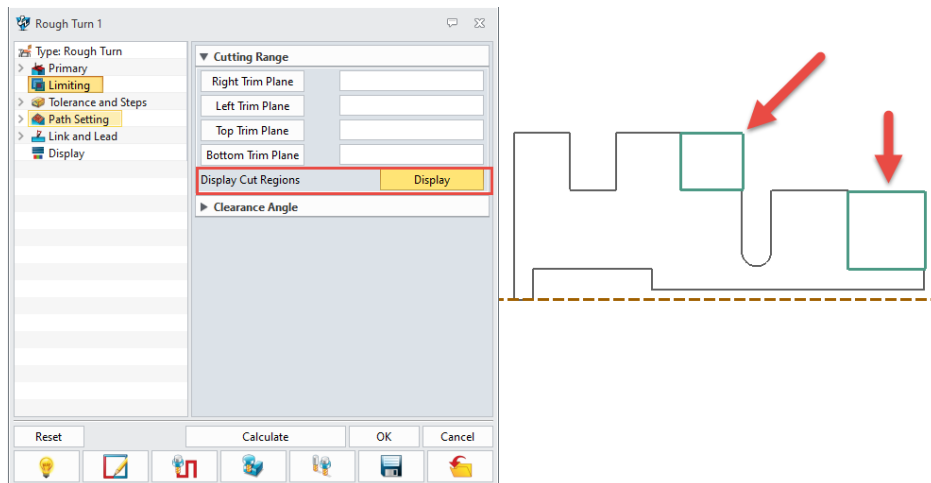
4.11 Shortest Tool Length Calculation Improvement

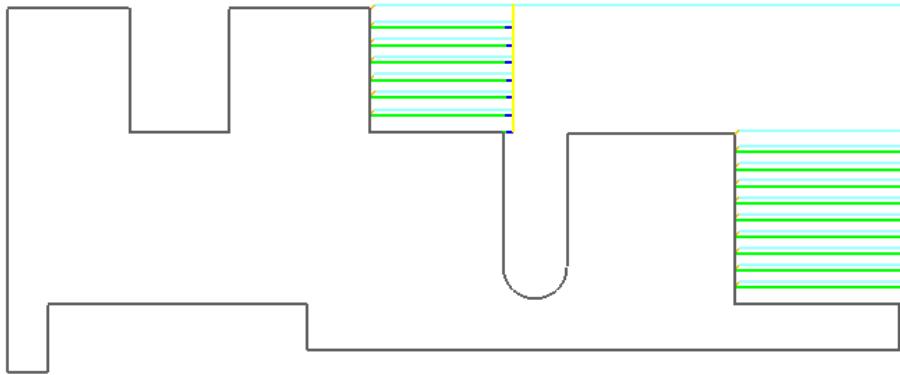
The calculation efficiency of the shortest tool length is more than 10 times higher than that of the old version, and the accuracy of the calculation of the shortest tool length is also slightly improved in ZW2D 2024.

	ZW3D 2023	ZW3D 2024
	90s	6s
	Efficiency improved by 14 times!	

4.12 Lathe Toolpath Supports Open Profile as Feature

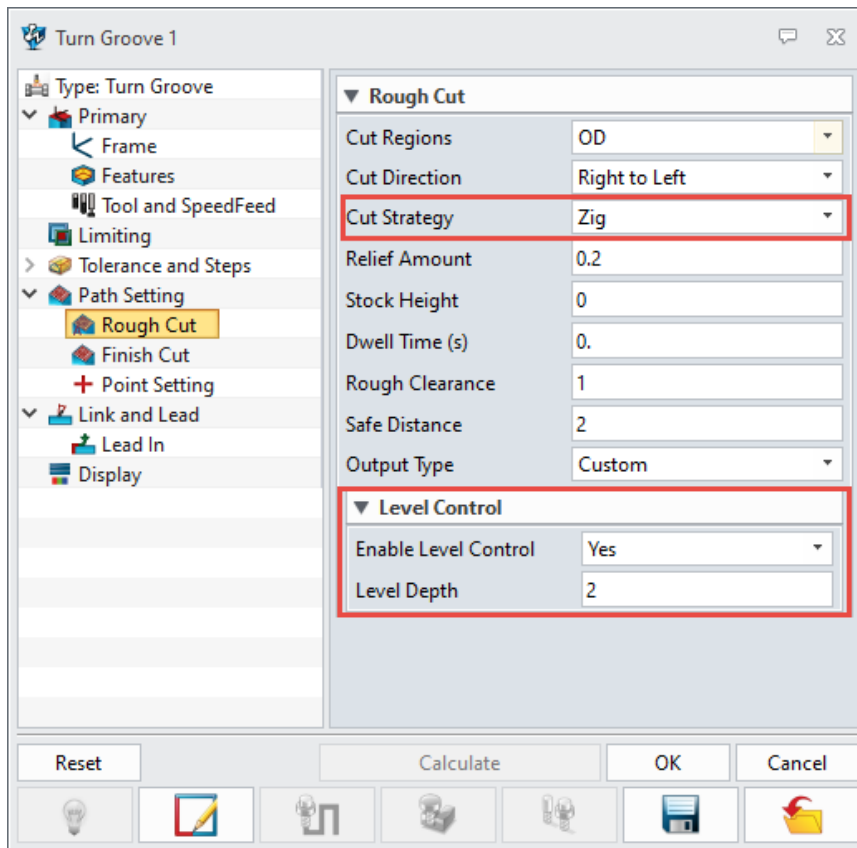
The lathe tool support open profile as feature to generate tool path in ZW3D 2024. In the actual programming scenario, it often needs to machine in some area, so that you need to redraw a closed sketch. Support selecting open profile that can greatly improve the programming speed and usability. At the same time, for the roughing and grooving, a new option “Display” is added to machining area, which can optimize user’s experience.

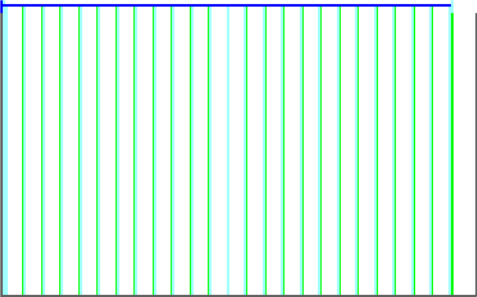
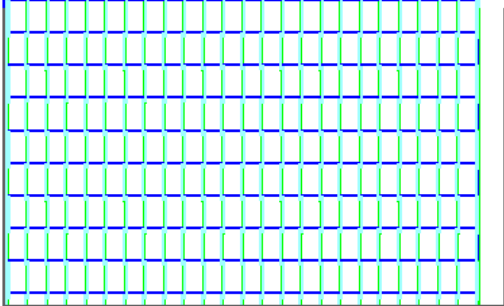


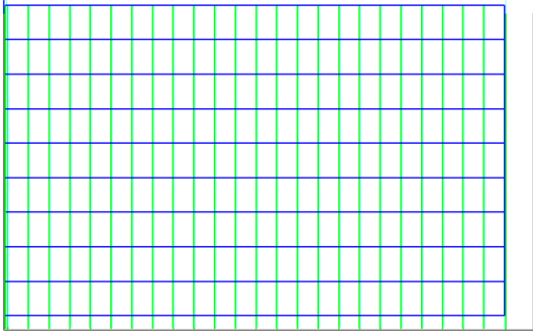
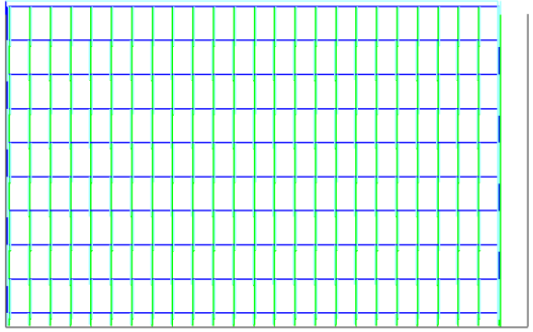


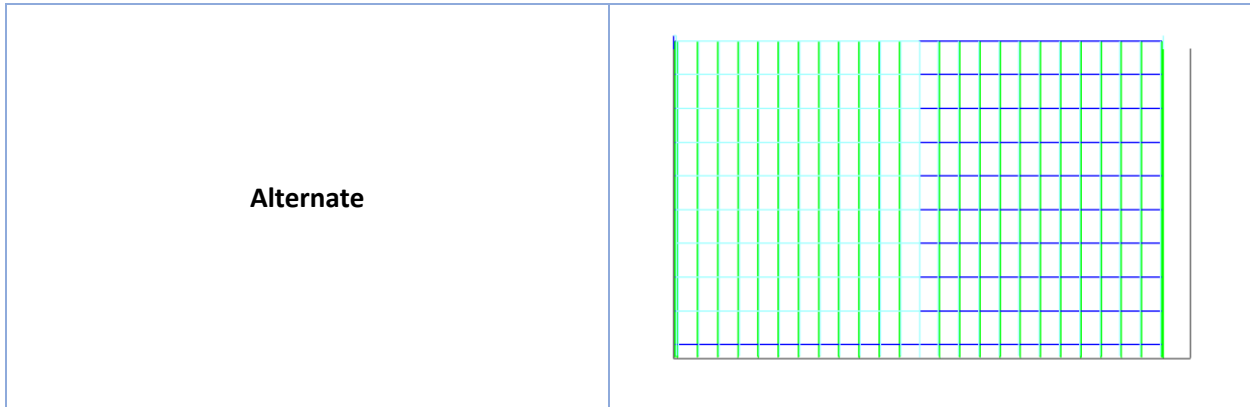
4.13 New Multiple Level Turn Groove

ZW3D 2024 adds level control to the turn groove and utilizes “Cut Strategy” to control different toolpath style. At the same time, support setting level connection and toolpath order and solve the machining abnormalities caused by deep groove machining, such as broken tool and damaged workpiece.



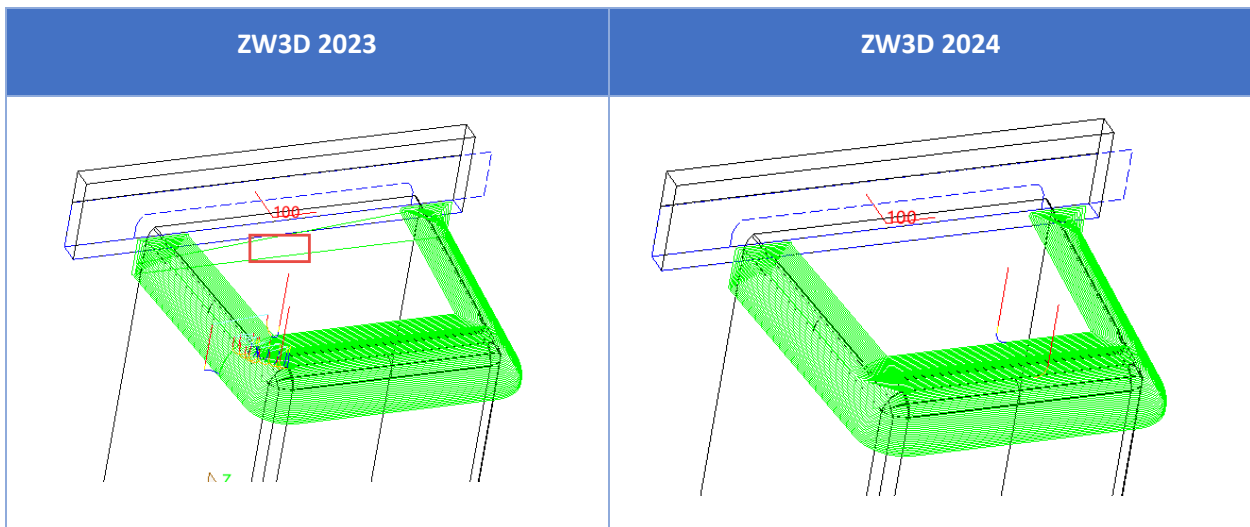
ZW3D 2023	ZW3D 2024
	

Cut Strategy	Tool Style
Zig	
Zigzag	



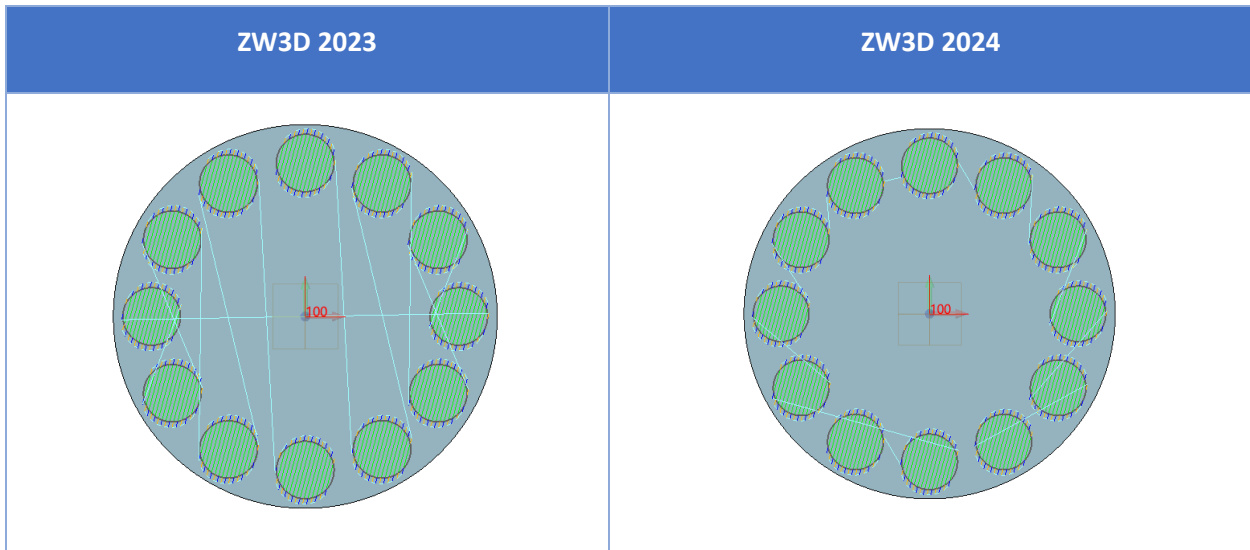
4.14 3D Offset Cut Security Improvement

ZW3D 2024 optimizes the security issue of 3D offset cutting operation at the bottom algorithm level and basically eliminate the tool path collision workpiece major security anomaly.



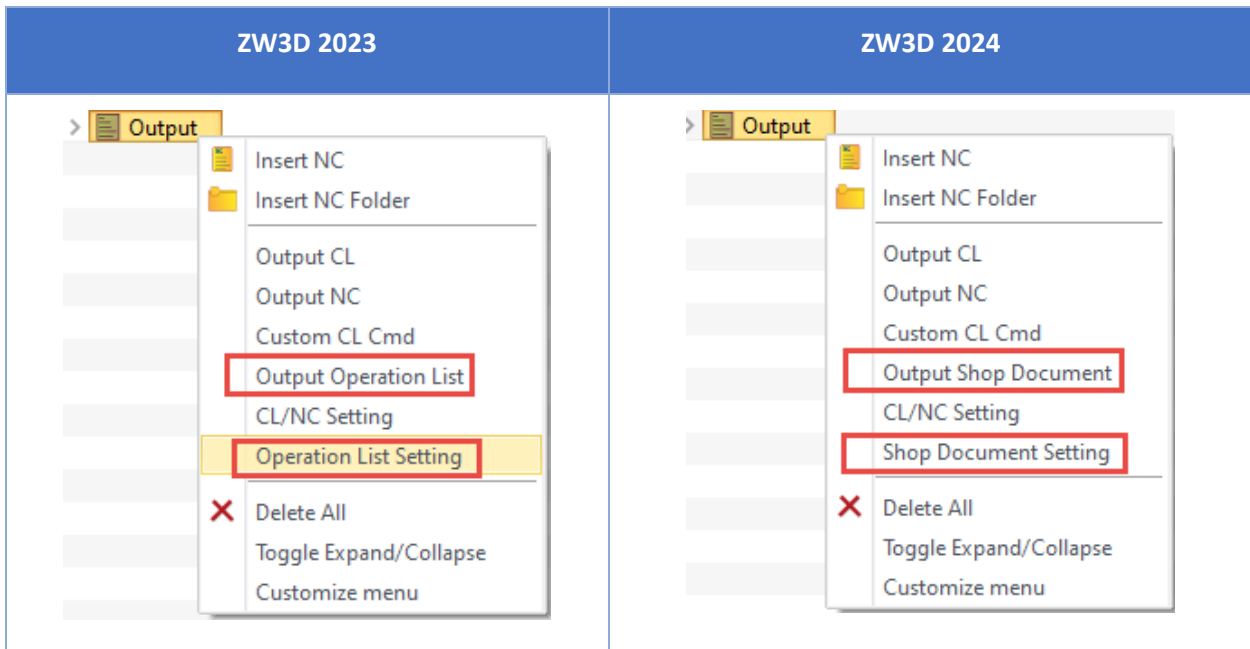
4.15 Base On TSP Algorithm and Lace Operation Order Improvement

ZW3D 2024 improves the inter logic of toolpath generating order when the parallel milling operation is machining multiple areas and avoids the abnormal which affect the machining efficiency, such as excessive skipping and chaotic tool path caused by non-optimization of machining sequence.



4.16 New Tool List Function

“Operation List” is changed to “Shop Document” in the interface in ZW3D 2024. User can select the document type to output in the setting interface. While optimizing the picture display effect in the operation list, the output function of the tool list is also added. Users can select the corresponding shop documents according to the actual production requirements.



CAM Operation List Settings

▼ Settings

Select Template

OperationList Template: default.htm

Custom Variables

Keyword	Name	Value

OK Cancel

CAM Shop Document Settings

▼ Settings

Select Template

Operation List Template: Default.htm

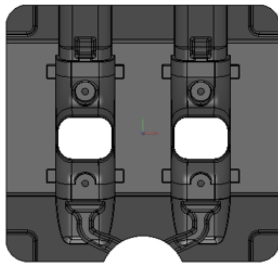

Tool List Template: ToolListExcel.xlsx

Custom Variables

Keyword	Name	Value

OK Cancel

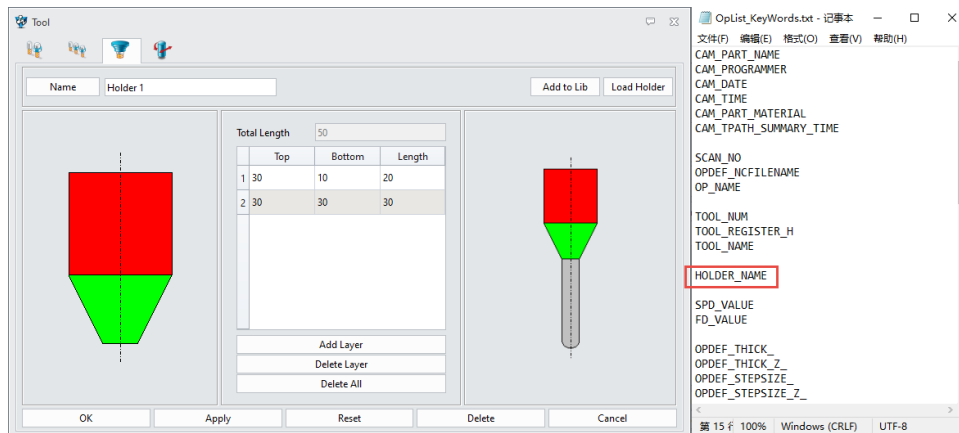
TOOL LIST

Machined Part	File2A									
Material	Aluminum-cast									
 										
Archived Path	C:\Users\ZW3D\Documents\ZW3D									
Programmer	ZW3D									
No	Program Name	Tool Name	Tool Num	Tool Dia	Overhang	Holder	Thickness (XY/Z)	Z min	Time	Comment
1	NC	D30R3	0	30	100		1/1	0.01	8h 29min	
2		D15R1	0	15	100		0/0	11.10	6h 1min	
3		D2R1	0	2	50		0/0	47.62	10min	
4		DSR2.5	0	5	50		0/0	44.35	16min	
5		D10R5	0	10	50		0/0	44.47	20min	
6		D2R1	0	2	50		0/0	47.62	10min	
7		DSR2.5	0	5	50		0/0	44.35	16min	
8		D10R5	0	10	50		0/0	44.46	1h 5min	
9		D2R1	0	2	50		0/0	47.57	26min	
10		DSR2.5	0	5	50		0/0	44.35	12min	
11		D10R5	0	10	50		0/0	44.46	54min	
12		D2R0	0	2	50		0/0	44.48	10min	
13		DSR0	0	5	50		0/0	44.48	17min	
14		D10R0	0	10	50		0/0	44.48	19min	
Estimated Machining time:										18h 59min 23s

4.17 Other

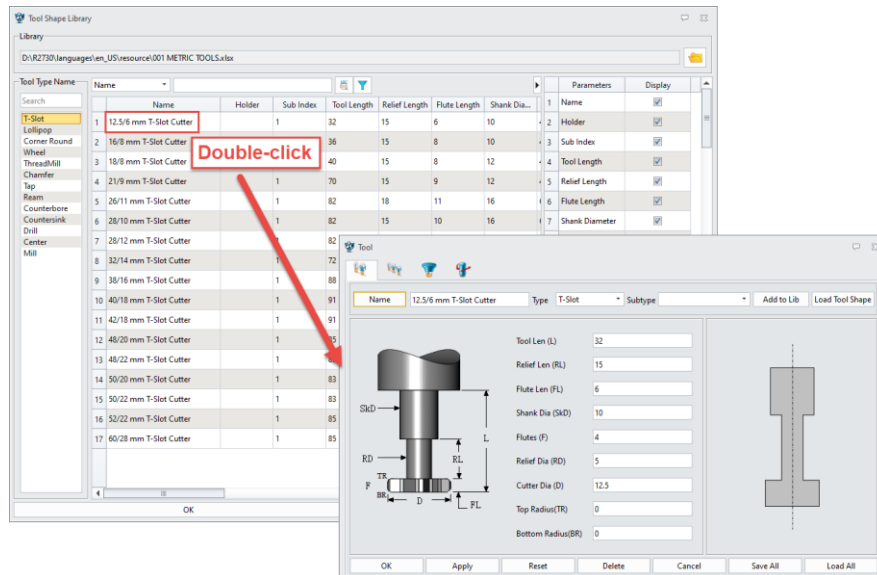
4.17.1 New Output Holder Name in Operation List

Newly added a variable `HOLDER_NAME` in ZW3D 2023X to output the tool holder name. Users can directly use the variable in the operation list template as required. The usable variables are recorded in `OpList_KeyWords.txt` file.



1.1.2 Double Clicking Tool Operation in Tool Library

ZW3D 2023X supports loading tools by double clicking.



4.17.2 Optimized Item Display Effect in Tool Library

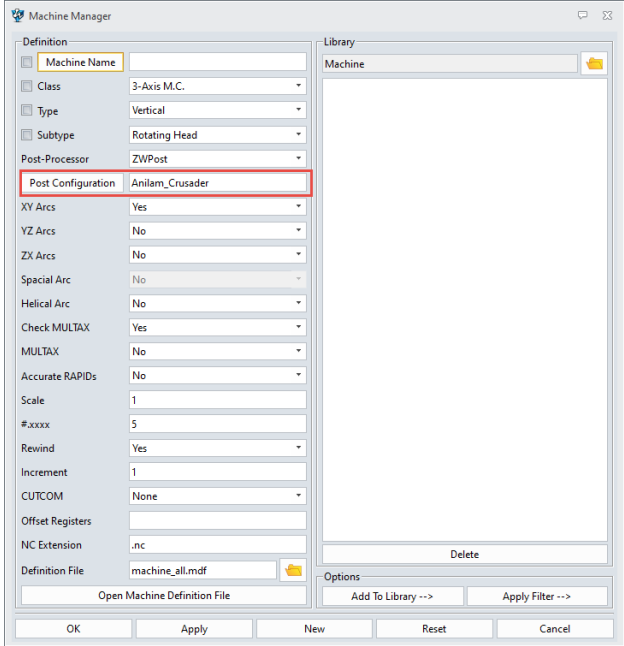
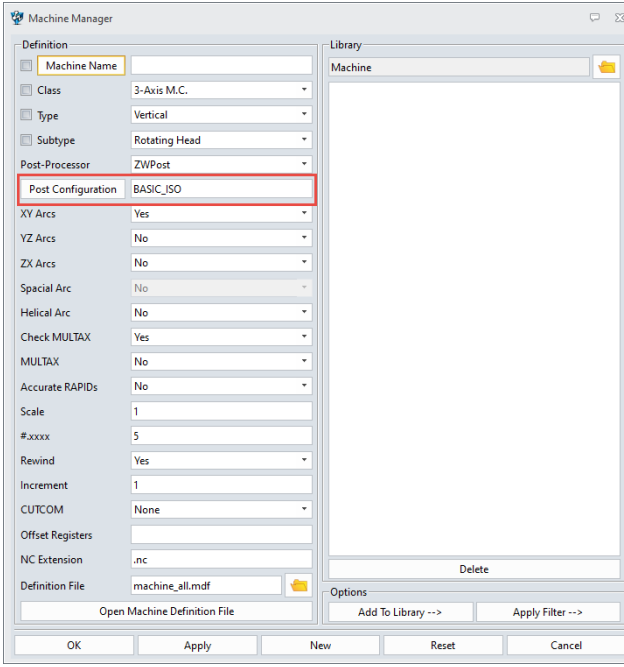
ZW3D 2023X optimizes the display effect of the items in the tool library, supporting a full display of the tool name with no more than 36 characters, convenient for user to view the name and parameter information of each tool completely.

ZW3
D 2023

ZW3
D 2023X

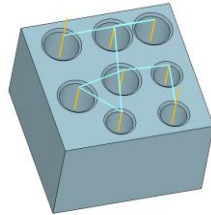
4.17.3 Modify Default Post-Processor File

The post processor configuration file in machine manager is changed to BASIC_ISO.znc in ZW3D 2023X, which is more suitable for user's usual practice.

<p>ZW3D 2023</p>	
<p>ZW3D 2023X</p>	

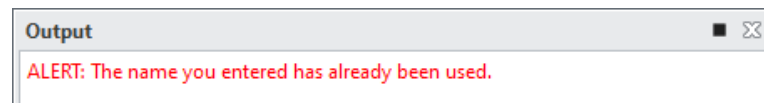
4.17.4 Support Countersink Operation Selecting Chamfer for Machining

ZW3D 2024 support countersink operation selecting chamfer tool for machining to realize the application scenario of small countersink chamfer machining.



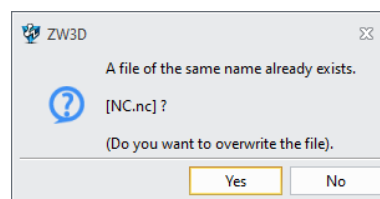
4.17.5 New Tool Duplicated Name Detection

ZW3D 2024 achieves to check the tool with same name. When users create a new tool, if they use the tool name already being used, click OK, and the output will pop-up error message. When users rename tool by right-clicking CAM tree and the name is already used, the tool will keep the original name.



4.17.6 New Output NC and CL File Duplicated Name Detection

In ZW3D 2024, when the NC or CL output through post-processing has a file with the same name in the same path, the software will pop up a prompt, as shown in the figure below. When the user clicks "Yes", the software will overwrite the original file with the same name, and the interface for viewing the new file will pop up. When the user clicks "No", the original file will not be overwritten, and the post-processing fails.



→Where is it

Part/Assembly >> Inquire >> Inspect Model >> Section

5 Simulation

5.1 ★Analysis Type

5.1.1 Add Fatigue Analysis

Under cyclic loading, the structure will often produce permanent damage at some points, which is called fatigue. The new version adds the fatigue analysis function, which supports two loading modes: constant amplitude loading and variable amplitude loading. It supports three damage calculation methods: maximum principal stress, maximum shear stress and von Mises stress. At the same time, it supports Goodman, Gerber, Soderberg and other mean stress correction methods. The results can show the structural damage, life and biaxial coefficient, and the rain flow counting results of the load time series.

→ Where

[Simulation >> New Structure >> Fatigue Analysis](#)

5.1.2 Add Random Vibration Analysis

Random vibration refers to the vibration that cannot be described by deterministic function but has certain statistical rules. The new version of the dynamic analysis module supports random vibration analysis. Support multi-point displacement, velocity, acceleration, node force PSD input. Support phase difference setting. PSD curves of displacement, velocity, acceleration, stress, strain, support reaction, RMS results and von Mises stress RMS results can be output.

→ Where

[Simulation >> New Structure >> Random Vibration](#)

5.1.3 Add Parameter Optimization

After parametric modeling, the model designer can quickly obtain the corresponding parameters that make the optimization target approach the target value. The parameter optimization function enables the model to be changed according to the simulation results. Realize the interaction between simulation end

and design end. It can reflect the characteristics of CAX integration.

→ [Where](#)

[Simulation >> Simulation Task >> New Parameter Optimization](#)

5.2 Analysis Function

5.2.1 ★Nonlinear Analysis Supports Large Displacement Analysis of Beam and Bar Element

In the type of nonlinear analysis, the first order beam element and the first order bar element of linear elastic materials support large displacement and large rotation analysis.

→ [Where](#)

[Simulation >> Simulation Task >> Nonlinear static analysis/nonlinear dynamic analysis >> Task options >> Geometric nonlinearity](#)

5.2.2 ★Nonlinear Analysis Supports Large Strain Analysis of Solid Element

In the type of nonlinear analysis, large deformation, large strain and large rotation analysis of second-order solid elements supporting elastic materials and plastic materials are supported.

→ [Where](#)

[Simulation >> Simulation Task >> Nonlinear static analysis/nonlinear dynamic analysis >> Task options >> Geometric nonlinearity](#)

5.2.3 Support Bolt Connection

The new version supports bolted connections, which can be set to simulate bolt behavior. The end faces of two cylinder faces (or two end faces of a cylinder face) whose central axes coincide are connected by bolts to constrain the relative motion of the two end faces.

→ **Where**

[Simulation >> Simulation tree >> Connection >> Bolting](#)

[Simulation >> Ribbon >> Connections >> Bolting](#)

5.2.4 Support Rigid Link Connection

The new version supports rigid link connections. It is simulated that two nodes are hinged through rigid bars. Any two vertices or reference points on the model can be tied together by a fixed bar hinged at both ends. The distance between these two positions remains constant during deformation.

→ **Where**

[Simulation >> Simulation tree >> Connection >> Rigid Link](#)

[Simulation >> Ribbon >> Connections >> Rigid Link](#)

5.2.5 Support Spot Welding Connection

The new version supports spot welding, which is used to weld faces between solids, solid faces and shells, shells and shells by defining spot welding.

→ **Where**

[Simulation >> Simulation tree >> Connection >> Spot Welding](#)

[Simulation >> Ribbon >> Connections >> Spot Welding](#)

5.2.6 Support Pin Connection

The new version supports pin connections, which simulate coaxial cylindrical faces connecting one or more entities by defining pin connections.

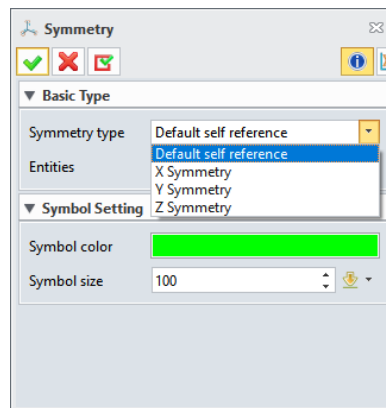
→ **Where**

[Simulation >> Simulation tree >> Connection >> Pin](#)

[Simulation >> Ribbon >> Connections >> Pin](#)

5.2.7 Support Symmetric Constraints

The new version supports symmetric constraints. This function uses the symmetry of the model to analyze the symmetrical part of the model to reduce the calculation of the problem. Symmetry constraints support setting the symmetry direction. It can also identify its own reference plane to create symmetrical constraints.



→ **Where**

[Simulation >> Simulation tree >> Constraint >> symmetric](#)

[Simulation >> Ribbon >> Constraint >> symmetric](#)

5.2.8 ★Support Thermal Effect Load

In the new version, static analysis and transient dynamic analysis support the setting of thermal effect loads. The node temperature values of one or more steps of the existing thermal analysis tasks (transient and steady state) in the current file can be used as an independent temperature load input.

→ **Where**

[Simulation >> Simulation tree >> Load >> Thermal Effect](#)

[Simulation >> Ribbon >> Load >> Thermal Effect](#)

5.2.9 Support Internal Heat Generation

In the new version, steady state thermal analysis and transient thermal analysis support the internal heat generation load, which is used to simulate the phenomenon of heat generation of components themselves.

→ Where

[Simulation >> Simulation tree >> Load >> Internal Heat Generation](#)

[Simulation >> Ribbon >> Load >> Internal Heat Generation](#)

5.2.10 Support Elastic Support

The new version supports elastic support. An elastic base can be defined between the selected solid surface of the part or assembly and the ground. This connection can resist tension, compression and shear.

→ Where

[Simulation >> Simulation tree >> Constraint >> Elastic Support](#)

[Simulation >> Ribbon >> Constraint >> Elastic Support](#)

5.2.11 Support Small Slip Frictionless Contact

In the new version of static analysis and nonlinear analysis types, the contact function supports the definition of small slip frictionless contact between solid and solid, solid and shell, shell and shell, which is used to simulate the small relative sliding between two contact surfaces under the frictionless state.

→ Where

[Simulation >> Simulation tree >> Contact](#)

[Simulation >> Ribbon >> Interaction >> Contact](#)

5.2.12 ★Support Small Slip Friction Contact

In the new version of static analysis and nonlinear analysis types, the contact function supports the definition of small slip friction contact between solids and solids, solids and shells, shells and shells. The defined contact surfaces will not penetrate each other. The friction effect generated at the contact location can be simulated by setting the friction coefficient of components.

→ Where

[Simulation >> Simulation tree >> Contact](#)

[Simulation >> Ribbon >> Interaction >> Contact](#)

5.2.13 Dynamic Analysis Supports Forced Motion

In the new version of transient dynamics and frequency domain dynamics analysis, it supports the imposition of forced velocity and acceleration loads in the selected direction of the constraint.

→ Where

[Simulation >> Simulation tree >> Load >> Acceleration Load/Velocity Load](#)

[Simulation >> Ribbon >> Load >> Acceleration Load/Velocity Load](#)

5.2.14 Support Shell to Shell Binding Contacts

Support shell-to-shell binding contacts, including the side of the shell facing the side of the shell, the side of the shell to the side of the shell, and the side of the shell facing the side of the shell.

→ Where

[Simulation >> Simulation tree >> Contact >> Select the face or edge of the shell](#)

[Simulation >> Ribbon >> Interaction >> Contact >> Select the face or edge of the shell](#)

5.2.15 Support Solid-Shell Couplings

The coupling connection between shell and solid is supported, and the edge of shell can be coupled to the surface of solid.

→ **Where**

[Simulation >> Simulation tree >> Connection >> Solid-shell Couplings](#)

[Simulation >> Ribbon >> Interaction >> Solid-shell Couplings](#)

5.2.16 Support Centrifugal Load

Two forms of velocity and acceleration can be defined, centrifugal force load applied to the axis of rotation, acting on the global coordinate system.

→ **Where**

[Simulation >> Simulation tree >> Mechanical Load >> Centrifugal Load](#)

[Simulation >> Ribbon >> Mechanical >> Centrifugal Load](#)

5.2.17 Support Moment Load

Supporting moment loads, which can be applied to beams and shells.

→ **Where**

[Simulation >> Simulation tree >> Mechanical Load >> Moment Load](#)

[Simulation >> Ribbon >> Mechanical >> Moment Load](#)

5.2.18 Support Bearing Load

Bearing loads along the axis radius are generated between the cylindrical surfaces in contact or between the circular edges of the shell, providing two distribution forms: sine and parabola.

→ Where

[Simulation >> Simulation tree >> Mechanical Load >> Bearing Load](#)

[Simulation >> Ribbon >> Mechanical >> Bearing Load](#)

5.2.19 Support Uniform Base Excitation

Support displacement, velocity, and acceleration of three basic excitation, applied to the whole space, support harmonic response analysis (modal).

→ Where

[Simulation >> Simulation tree >> Mechanical Load >> Uniform base excitation](#)

[Simulation >> Ribbon >> Mechanical >> Uniform base excitation](#)

5.2.20 Add Constraint Equation

Multiple degrees of freedom of geometry, geometric nodes and grid nodes can be constrained by constraint equations.

→ Where

[Simulation >> Simulation tree >> Constraint>> Constraint Equation](#)

[Simulation >> Ribbon >> Mechanical >> Constraint Equation](#)

5.2.21 Extend Mass Element Property

The mass point attribute is extended, and the mass matrix, moment of inertia and centroid offset of mass point attribute are supported.

→ Where

[Simulation >> Simulation tree >> Geometry Part >> Mass Element Property](#)

5.2.22 Add the Function of Composite Laminates

The composite material is widely used in Engineering due to its low weight and high strength property. The analysis of composite material, therefore, is usually an essential part of structural solver.

The new version adds the composite material laminate function and improves the multi-material (layer) laminate calculation function. Support for isotropic, orthotropic material plies (including hybrid plies). Composite laminate elements support all existing analysis types except nonlinear static and nonlinear dynamic analysis. In the function it adds 5 failure model.

5.2.23 Optimized the Compatibility of Compatible Meshes and Contact Functions

Improved the stability of the algorithm for bonded contact function, which allows bonded contact to be used with compatible mesh.

5.2.24 Contact Function Improvement

The contact function is improved, which make the setting of contact relation more convenient and the calculation more accurate. The function that can switch relationship between master and slave surface is added. It can make the setting and editing of master and slave surface more flexible. Fast detection algorithm contacts to find contact mesh elements is provided and calculation algorithm about coupling contact is optimized. Thus, the calculation efficiency and accuracy of contact function are both been improved.

5.2.25 Add Analysis Transform Function

Added the function of switching simulation tasks, which allows the simulation tasks to switch among the 11 simulation task types currently supported. The pre-processing information such as material properties, loads and constraints will be inherited or discarded automatically. The new simulation task can

directly use the mesh generated by the previous task.

This capability provides great convenience when you need to perform multiple types of simulation analysis on the model.

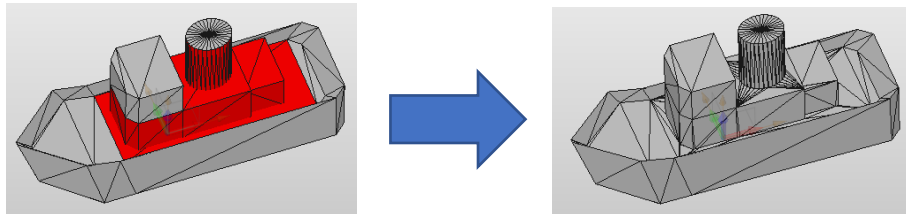
→ Where

[Simulation >> Base config >> Analysis Transform](#)

5.3 ★Mesh

5.3.1 Support 2D Mesh Intersection Repair Function

The new version supports the repair of 2D mesh intersection. This function can check the intersection of 2D meshes. In case of mesh intersection, the corresponding area can be re divided to generate non intersecting meshes.

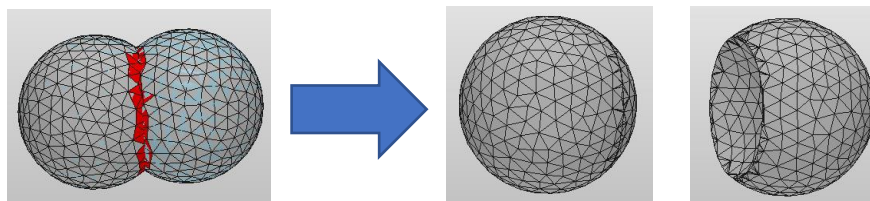


→ Where

[Simulation >> Ribbon >> Fix Mesh Flaw >> Fix 2D Mesh Intersection](#)

5.3.2 Support 3D Grid Boolean Function

The new version adds Boolean (minus) operations to the 3D mesh of the model. This function will check whether the model has 3D mesh intersection. And then trim according to priority to achieve Boolean (minus) operation of 3D mesh.



→ Where

Simulation >> Ribbon >> Fix Mesh Flaw >> 3D Mesh Boolean

5.3.3 Support 3D Mesh by 2D Mesh Function

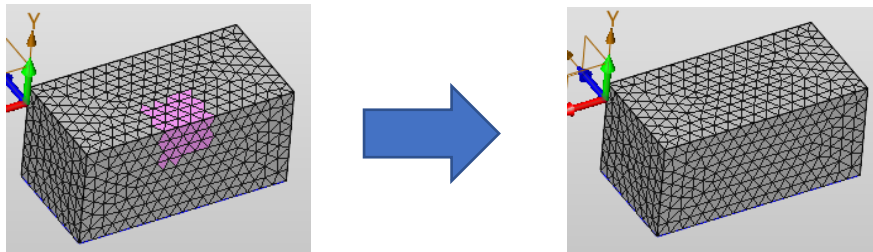
The 3D mesh function of the old version must be used under geometric conditions. The new version supports the generation of 3D meshes by 2D mesh filling.

→ Where

Simulation >> Ribbon >> Fix Mesh Flaw >> 3D Mesh by 2D Mesh

5.3.4 Support to Repair the Face Direction of 2D Mesh

The new version supports repairing the face direction of the 2D mesh, which can unify the face direction of the selected 2D mesh and reverse the face direction of the selected 2D mesh.

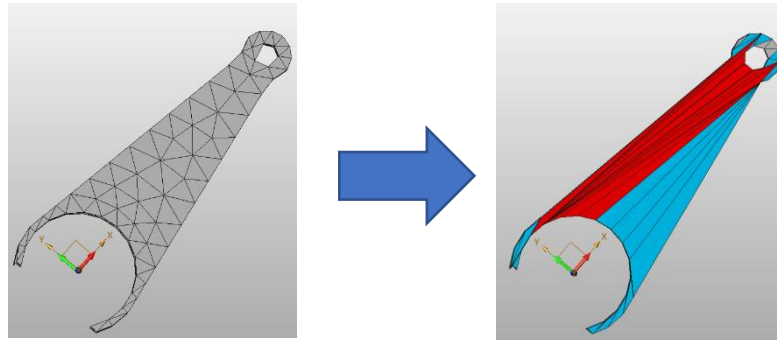


→ Where

Simulation >> Ribbon >> Fix Mesh Flaw >> Reverse 2D Mesh Direction

5.3.5 Support to Repair 2D Narrow Mesh

The new version can repair the 2D narrow mesh after checking them. Since the repair of slenderness cells will affect their neighborhood, it is necessary to select the neighborhood with the narrow mesh at the same time.



→ Where

Simulation >> Ribbon >> Fix Mesh Flaw >> Degenerate 2D Element

5.3.6 Support Local Mesh Update

In the previous version, when the geometry was modified, the mesh update function would subdivide all meshes. The new version of the mesh update command only updates the affected part of the mesh after geometric changes.

→ Where

Simulation >> Simulation tree >> Mesh >> Update Mesh

Simulation >> Ribbon >> Mesh >> Update Mesh

5.3.7 Add Auto Heal Function

The new version of the auto heal function supports the use of virtual topology operations to repair geometry. You can choose to heal all the faces of the selected geometry or only heal the failed faces. This function supports removing sliver faces, resolving self-intersect, merging short edges and removing small shells.

→ Where

Simulation >> Simulation tree >> Mesh >> Auto Heal

Simulation >> Ribbon >> Mesh >> Auto Heal

5.3.8 Support Generating Element Group

The grouping function of elements makes it easier to add parameters such as material properties to a group of elements. In the new version, new element groups can be created in mesh part. After creating a new group, users can generate an element group containing elements of the same type by selecting 1D/2D/3D elements. Each element can only be in one group.

→ Where

Simulation >> Simulation tree >> Mesh Part >> Group

5.3.9 Support Generating Mesh Set

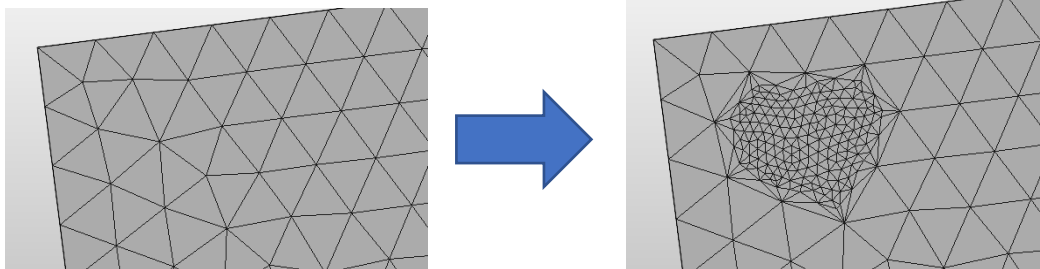
The new version supports placing a series of elements or nodes in a set to facilitate unified selection of elements or nodes in the set in the future. A set needs to be defined as an element set or node set. The same element or node can be stored in multiple sets at the same time.

→ Where

Simulation >> Simulation tree >> Set

5.3.10 Add Remesh Function

When the quality of the generated mesh elements is too poor or the inspection is unqualified, the remesh function can be used. The new version supports local regeneration from triangular mesh to triangular mesh or tetrahedral mesh to tetrahedral mesh.



5.3.11 Support for Ending Meshing Process

The new version supports pressing Esc to exit the meshing task during the meshing process.

5.3.12 Add Mesh Flaw Checking Function

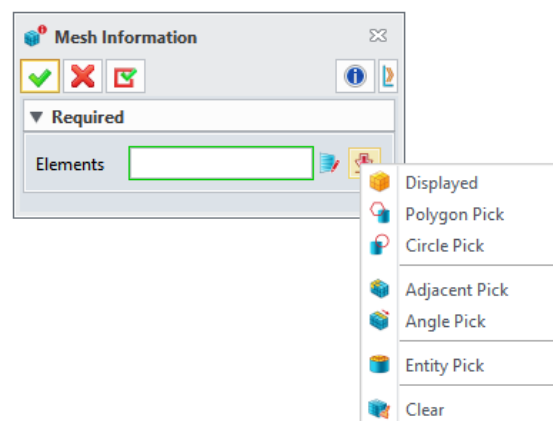
Geometry defects or improper meshing may lead to some improper meshing in the model. Incorrect mesh can result in failure during calculation or lead to a result of deviation. The new function supports mesh detection, which can locate defective meshes to support model repair or mesh adjustment. The software provides a wealth of defect detection rules, which can detect interference meshes, non-manifold meshes, open edges, degenerate meshes, self-intersecting meshes, duplicated meshes, etc., which may be detrimental to the calculation.

→ Where

[Simulation >> Check and information >> Flaw information](#)

5.3.13 Add Mesh Picking Tool

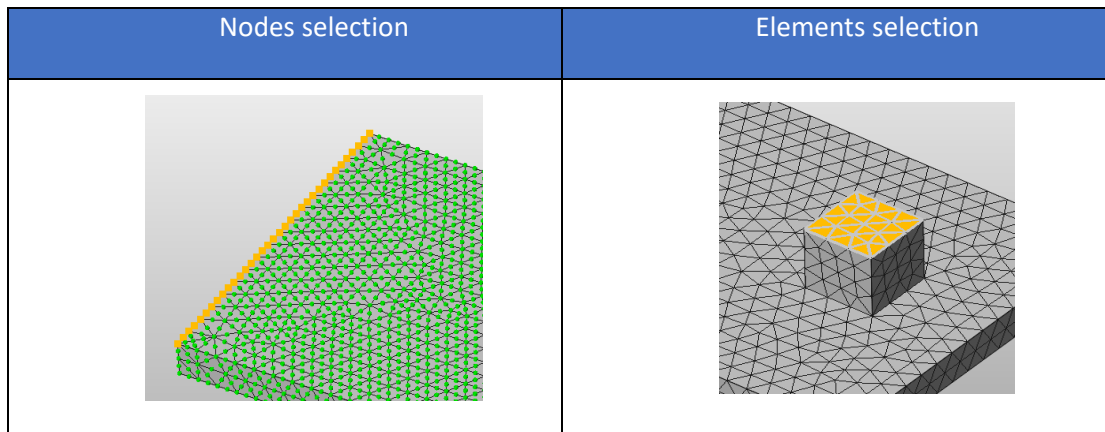
Add a new tool for mesh picking, which can help to quickly select all displayed meshes, mesh nodes, mesh edges or mesh faces.



5.3.14 Boundary Conditions or Loads Support the Selection of Mesh Nodes or Elements

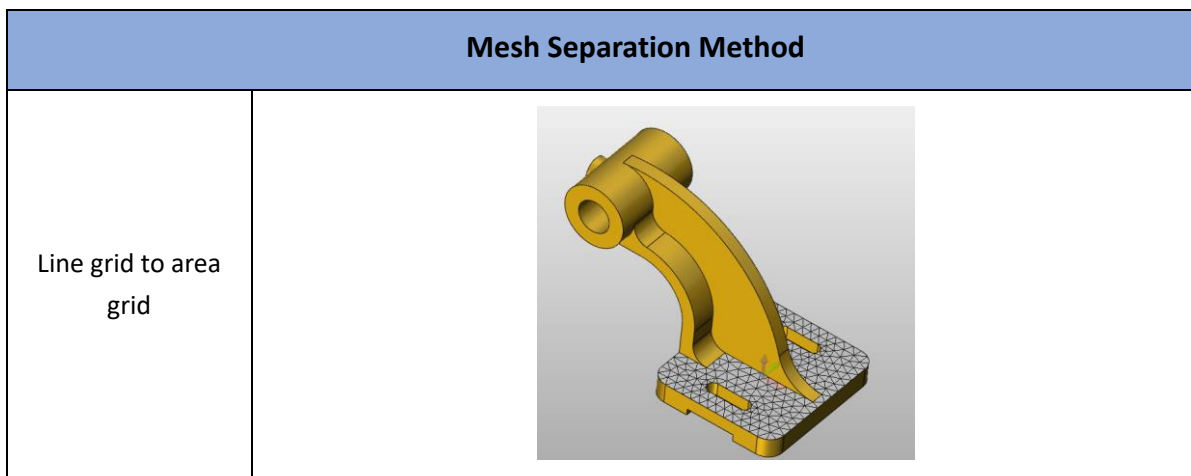
When defining the boundary conditions or loads of the model, users can select mesh nodes or elements.

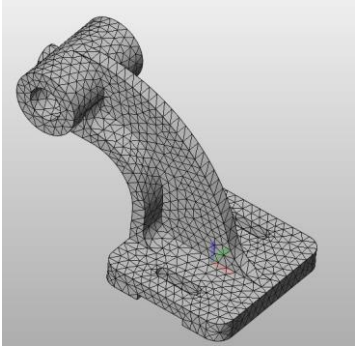
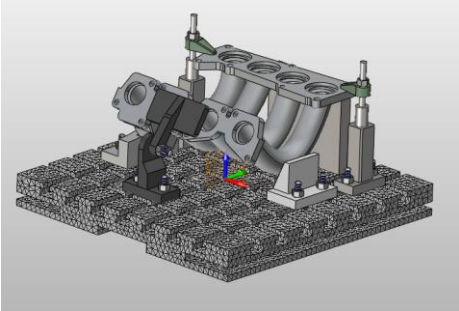
You need to change object type to Mesh node or Mesh face when you use this function. It supports box selection and click selection. See the figure below for details.



5.3.15 New Meshing Method for Separating Lines, Areas, and Volumes

The new version of the grid provides a more flexible grid division mode. It supports the meshing of local structural features of the geometry and the overall meshing of the structure; it supports the meshing of individual parts in complex assemblies and the global meshing of assemblies. Realize the flexible meshing method from line to surface to volume.



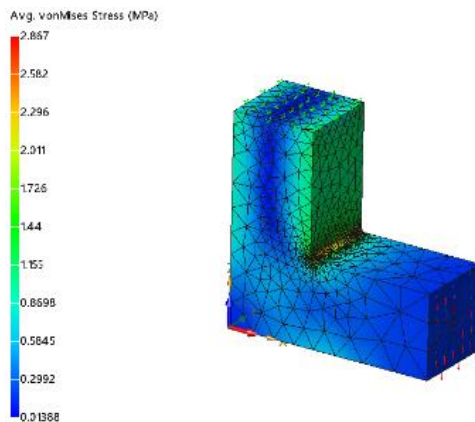
<p>Surface mesh to volume mesh</p>	
<p>Global meshing</p>	

→ Where

Simulation >> Ribbon >> Mesh >> 1D/2D/3D Mesh

5.3.16 Mesh Adaptive Function Added

Automatically refine the mesh according to the set goal. This function can be used for linear static analysis and steady state thermal analysis.



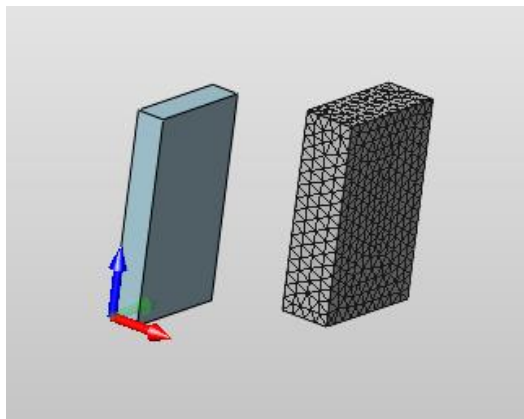
→ Where

Simulation >> Simulation tree >> Mesh Adaptive

Simulation >> Ribbon >> Mesh Adaptive

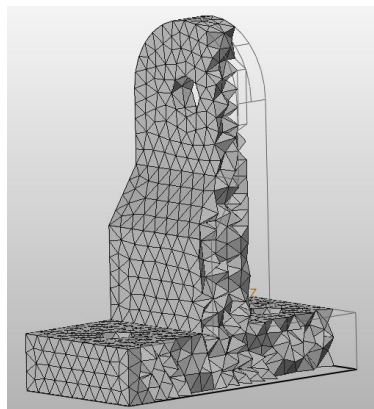
5.3.17 Mesh Generation for Single Part Supported

ZW3D Structural supports mesh division of a single part.



5.3.18 Newly Added Grid Display/Hide Function

The new version can be hidden or displayed by picking the grid, which is convenient for checking the grid division. The program supports 5 grid picking methods including frame selection and click selection. Grid picking also provides the basis for the grid editing function of the later version.



→ Where

Simulation >> Simulation tree >> Mesh >> Show/Hide Mesh

5.3.19 Support More Mesh Information Query

The mesh information function in the new version provides users with an overview of mesh information, including the total number of elements, nodes, and the number of various types of mesh. In addition, it supports selecting mesh nodes to generate detailed node information lists.

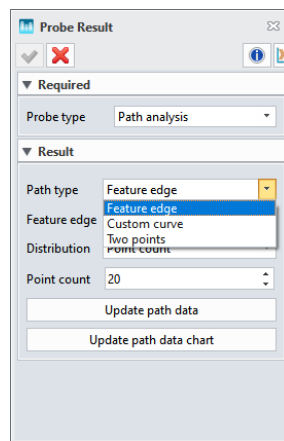
→ Where

Simulation >> Ribbon >> Mesh Information

5.4 Post-processing

5.4.1 Support the Display of Results Along the Path

The new version of the detection result can draw the result curve along the path. The path type can be a feature edge, a custom curve or a line segment established by two points. The interpolation position of the result can be determined according to the distribution form.



→ Where

Simulation >> Simulation tree >> Result >> Probe Result

5.4.2 Support Viewing Mass Participation Factor

The new version of modal analysis, linear transient dynamics analysis (modal superposition), response spectrum analysis and harmonic response analysis supports viewing mass participation factors. You can view the mass participation factor values at each frequency in the results.

→ **Where**

[Simulation](#) >> [Simulation tree](#) >> [Result](#) >> [Mass Participation Factor](#)

5.4.3 Support the Definition of Frequency Response Graph

The new version of modal analysis, linear transient dynamics analysis (modal superposition), response spectrum analysis and harmonic response analysis defines frequency response graph. The curve graph of modal order and frequency, the histogram of modal order and effective mass participation factor and the curve graph of modal order and cumulative effective mass participation factor can be drawn.

→ **Where**

[Simulation](#) >> [Simulation tree](#) >> [Result](#) >> [Frequency Response Graph](#)

5.4.4 Support Vector Sum

Support the vector sum to meet the needs of customers in the demand for force among the probe result.

5.4.5 Support Simplified Chinese and Traditional Chinese Display in Result Post-Processing

The post-processing results of the new version support simplified Chinese and traditional Chinese display in the Chinese environment of the software, which improves the user's post-processing experience.

5.4.6 The Results Support Equivalent Strain in Post-Processing

In post-processing, you can choose to view the equivalent strain of the model.

→ Where

[Simulation >> Result >> New Result Display >> Type-element result-EqTStrain](#)

5.5 Other

5.5.1 The Function New Expression Function Supports Cylindrical Coordinate System and Spherical Coordinate System

The new version of the new function expression function supports the definition of function variables in the cylindrical coordinate system or the spherical coordinate system, which makes the way of building function expressions more diverse.

→ Where

[Simulation >> Result >> New Result Display >> Type-Geometry](#)

5.5.2 Greatly Improve the Efficiency of Mesh Display

The new version of the software has optimized the mesh-related display function. Resource consumption has been effectively reduced. Mesh display efficiency and related operation response speed are improved, enabling the software to support smooth display and mesh picking of 3000w-level number of mesh elements. Tests based on big cases show that the memory is reduced by 86%, the memory usage is reduced by 60%, the time of mesh picking highlight is reduced by 80%, and the viewing angle switching and zooming time is maintained at the millisecond level.

The optimization of the mesh display function expands the performance boundary of the software and improves the user experience.

5.5.3 Supporting the Import and Export of BDF Format

The new version supports the import and export of BDF format. BDF format is one of the general finite element model formats.

→ Where

Simulation >> Ribbon >> Import/Export

5.5.4 Support Simplified Chinese and Traditional Chinese Display in Result Post-Processing

The post-processing results of the new version support simplified Chinese and traditional Chinese display in the Chinese environment of the software, which improves the user's post-processing experience.

5.5.5 Add Show Beam Section Function

Added the function of show beam section, which can intuitively display the 3D shape of the beam.

When using it, firstly mesh the beam. After dividing, you can choose to show beam section. The effect is shown in the figure below.



→ Where

Simulation >> Mesh >> Mesh the beam >> Simulation tree >> Show Beam Section