



ZW3D WHAT'S NEW

V 2023



ZWSOFT CO., LTD.(Guangzhou)





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ZW3D[™] V2023 What's New

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Highlight of ZW3D 2023

Basic:

<u>New Multi-Print</u> <u>Pick Interaction Improvement</u> <u>Datum Improvement</u> <u>Thickness Analysis Improvement</u> <u>Color Improvement</u>

CAD:

New Smart Constraint Inference Face Offset Supports G2 **Spiral Helix Improvement New Revolve Flange** Lofted Flange Improvement **New Smart Fastener New Envelop** 2D/3D Sketch External Reference Improvement Sketch Curve Continuity Inquire Improvement **3D Sketch Improvement** Loft Improvement New External Geom Copy **Blend Face Improvement** Pattern Feature **UDF** Improvement New Double Bend Flange **New Universal Joint New Pulley Assembly Feature** New Motor in Assembly Animation

Drawing Sheet:

<u>New 3D Crop</u> <u>View Projection Efficiency Improvement</u> <u>New Batch Sheet Format Attributes</u>

ZW3D



Mold Design:

New Engrave New Oil Groove New Load Asm-Tree in Mold Workpiece Improvement Layout Improvement Standard Part Mechanism Improvement

Piping:

<u>New Parallel Routing</u> <u>New Hanger</u>

CAM:

Tool Speed and Feed Overall Upgrade Tool Library Overall Upgrade New Multi Work Station New Reference Tool in 2 Axis Operation FMS Support 5-axis Index Milling Simulation FMS Support Auto On/Off RTCP Speed Up QM Rough Operation Calculation Support Taper Hole and Taper Boss Milling 2D Pocket Operation Supports External Lead-in Open Pocket New Follow Part Path Pattern in 2X Contourcut operation Profile Feature Curve Attribute Optimization Update User Customized Tool Functionality

Note: Important enhancements in this article are marked with \star



1 Basic

1.1 File Dialogue Improvement

1.1.1 Open/Import Panel Improvement

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

💯 Open						₽ 33
n n 🔻 🕆 💄	My Computer	Þ			Search	
						II. 🗖
📋 My Document	Name 🔺	Size	Туре			
	(C:)		Drive			
	(D:)		Drive			
My Computer 💼	(E:)		Drive			
					Preview	
					Object list	
		1111			•	
Local	conv				Quick filter:	• • • •
	сору					
File name:				*	All Files (*)	*
✓ The nume.						
Root object:				Search	Open	Cancel
				Search	Open	Cancel
				Search	Open	Cancel
Root object:	My Computer	b				₽ X
Root object:	▶ My Computer	Þ			Open	₽ X
Root object:			Tura			C X
Root object:	Name 4	1.53	Туре	Date Modified		C X
Root object:	Name 4	Size	Drive	Date Modified 2021/10/20 17:15		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name 4	Size	Drive	Date Modified 2021/10/20 17:15		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:)	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:) Document (H	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X
Root object:	Name C: Software (D:) Document (H	Size	Drive Drive	Date Modified 2021/10/20 17:15 2021/10/20 17:14		C X

→ Where it is

Any Environment >> File >> Open/Import



1.1.2 New File Name Search

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

💯 Open					\Box	23
	🕞 « C: 🕨 Users 🕨 Administrator 🕨 Desktop 🕨	\sim	\diamond	Search		

1.1.3 New Quick Filter

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

1.1.4 New Shortcut Folder Function

1.1.4.1 Working Folder

User can set folder as working folder by context menu.

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

Display	ZW3D file folder	C:\Users\Administrator\Documents\ZW3D	
Eiler	Save last open folder as next default directory		
Files	Working folder	C:\Users\Administrator\Documents\ZW3D	-

1.1.4.2 Custom Shortcut Folder Context Menu

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



Name 🌰	Size Type	
Project	File Folder	
Project 01	File Folder	
Project 02	File Folder	
Project 03	File Folder	
Project 04	File Folder	
Project 05	Open	
Project 06		
h Project 07	Cut Copy Delete	
	Rename	
	Set quick directory	
	Set working folder	

1.2 File Path Management Improvement

1.2.1 File Search in Configuration

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

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



2 Configuration		₽ ⊠
General Part 2D Color Background Display	Punch library Symbol library Clipboard library ☑ Session journal Export folder Drafting Paper size definitions DEF_SHEE	D:\0ZW3D\2700_Piping
Files CAM	BOM template	•
User PDM ECAD Routing	Name Image: Active folder Image: Active folder Image: Active & its sub-folders Image: Active & its sub-folders Image: Active folder's recursive directories Image: Active folder's recursive directories Image: Components folders Image: Standard parts folder	
	Search with pop-up box	
Reset Default		OK Cancel Apply

1.2.2 Search Path Expansion

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

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

Saved folder: The saved folder path in the file.

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

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

Working folder: The working folder that set by user.

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

Components folders: The folder that locates the components.

Standard parts folder: The folder of standard parts.





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

1.2.3 Search with Pop-up Box

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

Name								
Assembly 004.Z3ASM > Assembly 004	C:\Users\Z\	V3D\Documents\ZW3D						
ess "Yes" if you would like ZW3D to sea	arch your default	folders.	6					
			6					
ress "Yes" if you would like ZW3D to sea ress folder icon to specify new path for You can disable this prompt by switchir	each missing obj	ects.	~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~					

The popup windows improvements mainly include the following:

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



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

→ Where it is

Configuration >> File >> File search

1.2.4 Search Path Adjustment

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

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

💯 ZW3D Search Path	15		₽ 33
Auto Save	Auto Record	Save/use paths in Z3 file	
When "Auto Record'		ngs/assemblies/plans. ically added to this list when you open st are saved to the default configuratio	
	Reset OK	Cancel	

→ Where it is

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



1.3 Support Maximum Distance Measure

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

🔚 Distance		23	
🗸 🗙 🖪	Image: Sector Secto		
Required	Positive • Max distance •		
N	. 😵 💸 . 📢		
1st entity	F7 5	-	🦉 Measure Distance 🖓 🕅
2nd entity	F14	Image: Section of the sec	
▼ Result Cont	rol	_	Dist-X 380 [mm]
X distance			Dist-Y 240 [mm]
V distance			
Z distance			
Sign		•	OK Cancel
Min or max	Max distance	•	
Direction	Magnitude only	•	
Projection	None	•	
Methods	Between bounds	•	
Associative	Measure		
Settings			

→ Where it is

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

1.4 Echo Effect Improvement

1.4.1 Modeling Command Echo Improvement

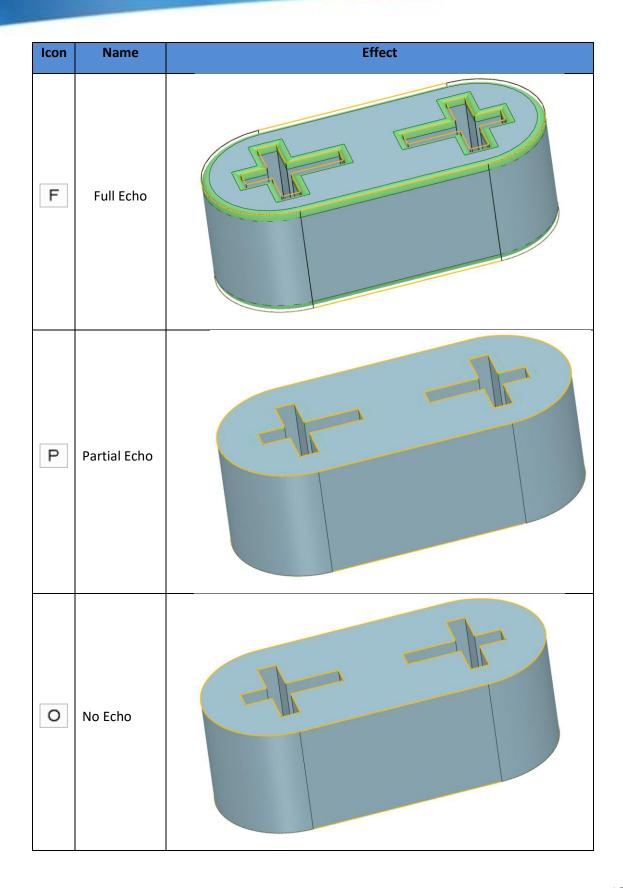
We added "Partial Echo" and "No Echo" effects.

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

1.4.1.1 Echo Effect Display

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









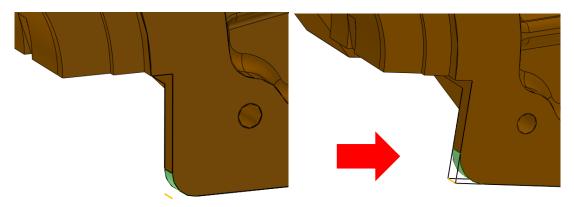
Part Environment >> Shape>> Engineering Feature >> Fillet/Chamfer/Draft

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

1.4.1.2 Full Echo Effect Improvement

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

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



1.4.1.3 New Auto Switch Partial Echo Checkbox

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

Compass Triad Display : ON Show Hidden : ON	🖉 Echo Type 🖓	23
✓ Echo	Type Quick Echo	Ŧ
Color Echo type : Quick Echo	Detail Image: Constraint of the second sec	•
Attribute Tag Display : ON Texture Map Display : ON	OK Cancel	Ŧ
Material Display - OFF		

1.5 **★**New Multi-Print

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



Sel	ect File	25		
•	1			
V	Includ	le files from sub-fol	ders	
	V		File name	
[V	l:\LZH\Case\CAD\Z	ZW3D\W05-D238.Z3	
Sel	act Dra	winds-		
Sel	ect Dra	awings	Obiect	File
Seli		-	Object W05-D238-4M	File H:\LZH\Case\C
		Sheet		
1		Sheet Zeichnungsblatt1	W05-D238-4M	H:\LZH\Case\C
1		Sheet Zeichnungsblatt1 Zeichnungsblatt1	W05-D238-4M W05-D238-4M W05-D238.00-4	H:\LZH\Case\C H:\LZH\Case\C H:\LZH\Case\C

Menu Bar >> Multi-Print

1.6 Equation and Variable Browser

1.6.1 New Global Control Function

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



😨 Configuratio	on								Ç	53
General		Max interval (secor	ids) 10		Cache life span	(days)	2			
		Default datums	CSYS 🔻		Datum scale		Defaul	t		
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2D		🗹 Allow dynamic drag o	f assembly compone	nt						
Color		Ignore constraint failu	re during regeneratio	on						
Background		Load link information	of reference objects							
Display		Enable assembly layer								
Files		Parallel import assem	bly file							
CAM		Save component files	during assembly imp	oort						
		Enable quick constrain	nt solver for complex	assembly	1					
User		Only published set ca	n be referenced							
PDM		Folders display settings								
ECAD		Solid Auto	*		Configuration	Auto	•			
Routing		Surface Auto	•		Expression	Auto	•			
		Wireframe Auto	•		Block	Auto	-			
		Pick Set Auto	•		Publish Set	Auto	-			
		Part name Display	/ part name	*	Entity name	With feat	ure nam	e *		
		Maximum entities listed un	der Folders		1000					
										-
Reset	Default					0	к	Cancel	Ар	ply

Part Environment >> Configuration >> Part

1.6.2 New Publish Variable Function

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





Equation Manager Expression List								₩ 23	👰 Variable Browser		₽ %
Filter All	•			ð				4	Browsing this file Part003.Z3PRT		
Name	Expressio	n Value N	lin Max	Unit	Туре	Public	Source	Descripti	Browsing this object		
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Reset					0	ĸ	Cancel	Apply	Imp	oort Cancel	

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

1.6.3 Equation Manager Function Adjustment

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

Filter	All 👻				2	õ			1	2 2 D		K
Nam	ie	Expression	Value	Min	Max	Unit	Туре	Publish	Descri	iption	State	
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	<u>External variables</u>	1										
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	Attr_Density_1	[Error of Hole	8.03e			kg/m	Num					
	Part_Attr_Area_1	[Error of Hole	3625			mm^2	Num					
	20 Part_Attr_Volume_1	[Error of Hole	8697			mm^3	Num					
	20 Part_Attr_Length_1	[Error of Hole	35			mm	Num					
	20 Part_Attr_Width_1	[Error of Hole	20			mm	Num					
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Part Environment >> Tools >> Equation Manager

1.6.4 Variable Browser Improvement

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

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

👰 Variable Browser					₽ %
Browsing this file					
Gear.Z3PRT					- 🔚
Browsing this object					
bronning this object					
Gear				•	
Objects and variables					
Name	Value	Unit		Name	
Search	Search	Search		Expression	
🗙 🗹 📲 Standard attribute					
✓ part_name			\checkmark	Value	
✓ part_number				Min	
✓ part_class				Max	
✓ part_designer			V	Unit	
✓ part_cost				Туре	
✓ part_supplier					
✓ part_description				Description	
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✓ part_manager				Customize menu	
✓ part_material			_		
✓ part_startdate					
✓ part_enddate					
✓ part_derived					-
Only published variables					
Only published variables					
		Import	Cance	el	

→ Where it is

Part Environment >> Tools >> Variable Browser



1.7 New Layer Rule

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

🖗 Layer Rule Edito	r				\Box	23
Layer Rule Editor						
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Active	Layer	Attribute	Condition value	Operator		
					Add	
					Delete	
					Submit E	dit
					Clear	
		ОК	Cancel			

→ Where it is

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

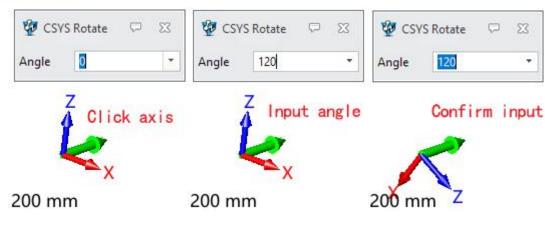
1.8 Ease of Use Improvement

1.8.1 3D Navigation Tag Supports Specifying Angle

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

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

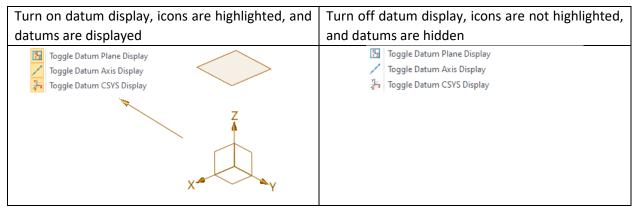




Part/Assembly Environment >> Drawing Region >> 3D Navigation Tag

1.8.2 New On/Off Datum Plane/Axis/CSYS

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



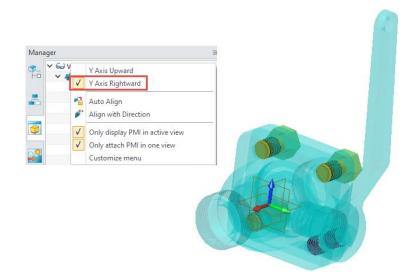
→ Where it is

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

1.8.3 New Y Axis Rightward

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

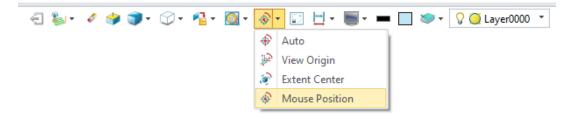




Part/Assembly Environment >> View Manager >> View Context Menu >> Y Axis Rightward

1.8.4 DA Toolbar Support Rotation Center

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



→ Where it is

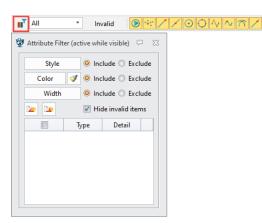
Part/Assembly Environment >> DA Toolbar >> Rotation Center

1.9 **★**Pick Interaction Improvement

1.9.1 Sketch/Drawing Sheet Environment Filter Improvement

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





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

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

Filter List	Pick Filter Panel			
All Invalid OPERATION	All Description of the second			

→ Where it is

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

1.9.2 Support More Intentional Selection Behaviors

1.9.2.1 Edges Intentional Selection Behavior

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





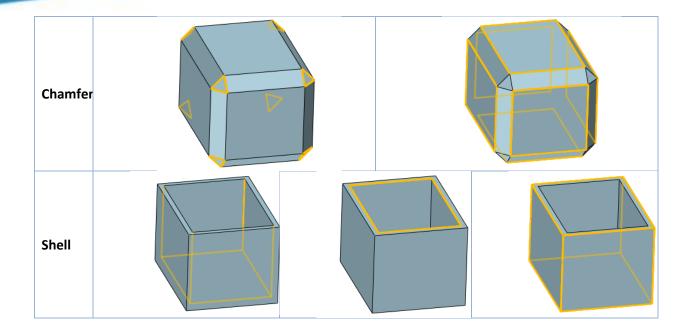
1.9.2.1.1 Feature Edges

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

Feature Feature Edges Picking Effect Extrude Revolve Sweep Fillet

"Feature Edges" Picking Effect Display





1.9.2.1.2 Vertex Edges

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



1.9.2.1.3 Face Edges

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



Edge	🝷 😻 Part Only	· ····	Sa Ba Ba Ba 🕅	: 🗮 📴 付	🍻 ၊ 🚯 斗 🔳 🖬 Face Edges	- 🔧 🦑
					🐲 Face Edges Settings	₽ X
					 All 	
					 Inner Outer 	

"Face Edges" Pick Effect Display

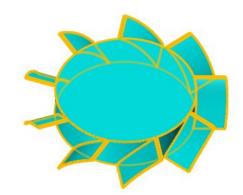
Rule	Pick Effect
All	
Inner	
Outer	

1.9.2.1.4 Shape Edges

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

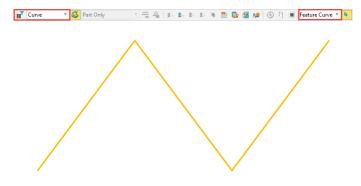






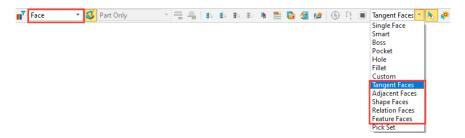
1.9.2.2 Intentional Behavior on Curve

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



1.9.2.3 Face Intentional Selection Behaviors

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

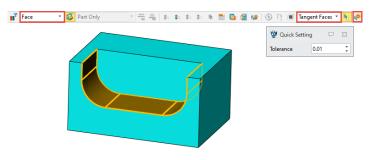






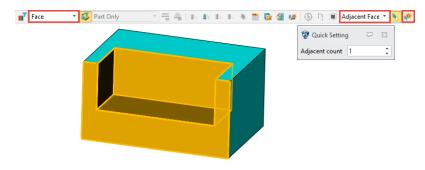
1.9.2.3.1 Tangent Faces

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



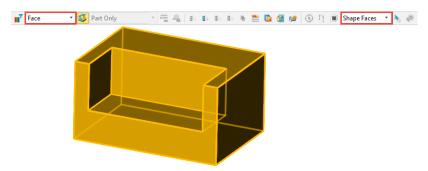
1.9.2.3.2 Adjacent Faces

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



1.9.2.3.3 Shape Faces

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





1.9.2.3.4 Relation Faces

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

• Coaxial

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

Tangent

Find faces that are tangent with the feed face.

Coplanar

Find faces that are coplanar with the feed face.

Coplanar axes

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

• Equal radius

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

Symmetric

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

• Offset

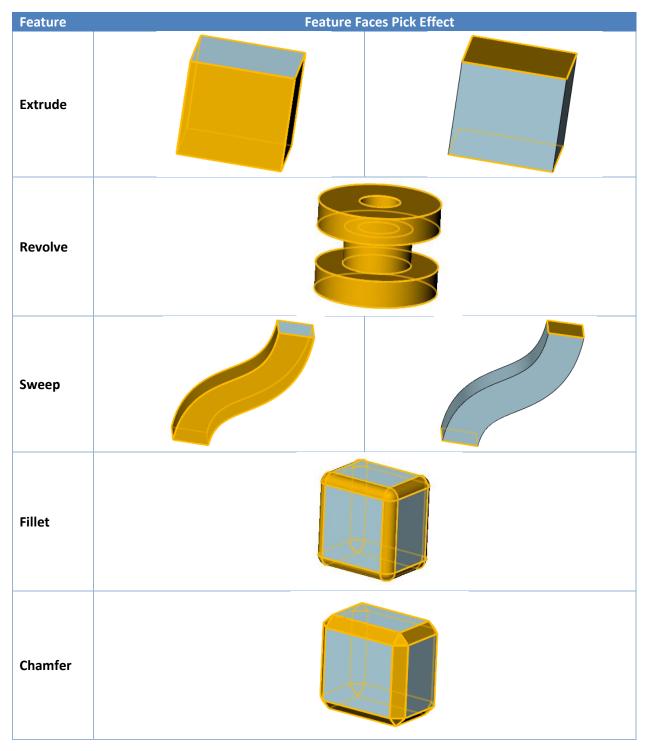
Find faces that have partial area overlapping and opposite vector with the feed face. There is only one offset face on the feed face.

1.9.2.3.5 Feature Faces

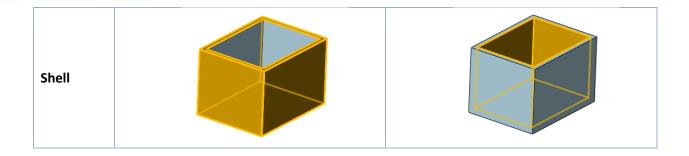
When "**Feature Faces**" are selected as pick rule, user can execute the selection behavior by leftclicking the mouse on a face. The system will find the corresponding faces based on the feature.



"Feature Faces" Pick Effect Display







1.9.2.4 Shape Intentional Behaviors

1.9.2.4.1 Feature Shapes

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

1.9.2.4.2 Distinguish between Solid and Surface in Shape Selection

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

1.9.3Pick Interaction Behavior Improvement

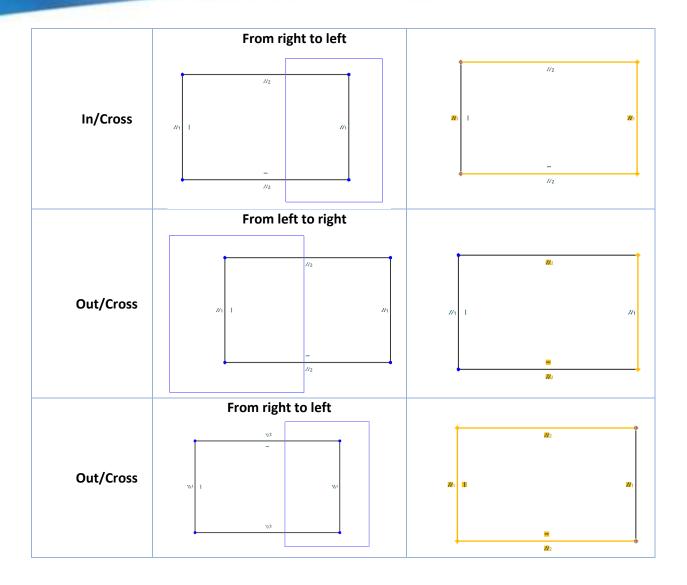
1.9.3.1 Windows Pick Behavior Improvement

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

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

Pick Mode	Windows F	Pick Direction	Pick Result	
	From le	ft to right		
			A	
		//2	//2	Ī
In/Cross				
	<i>W</i> 1 I	//1	<i>W</i> 1	//1
		-	-	
		//2	//2	





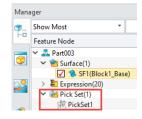
1.9.3.2 Pick List Supports Quick Switch

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

1.9.4New Pick Set

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





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

1.10 **★**Datum Improvement

1.10.1 "Datum CSYS"

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

The improvements show as below:

Method	Main Improvement	lcon
Geometry	With the improvement, user can select up to three reference geometry objects at once. It will automatically create a Datum CSYS and support switching axes. The reference geometry includes points, lines, edges, faces and CSYS (User can set offset or rotation first when CSYS is referenced).	Required Ist axis X 2nd axis Y Vert solution Orientation Offset first Orientation Orientation
3 points	(1) Icon updated.(2) Support switching axes and reverse their directions (Specify X and Y as default axes)	▼ Required ↓ <th< td=""></th<>



3 faces	The new method allows users to select three reference planes to create a Datum CSYS, switch axes and reverse the axis direction.	Required Image: Constraint of the second se
Point and 2 directions	The new method allows users to select an origin and 2 directions to create a Datum CSYS, switch axes and reverse the axis direction.	▼ Required Image: Construction 1 X Image: Construction 1 Direction 2 Y
Plane, point and direction	The new method supports creating a Datum CSYS where the selected plane is defined as Z-axis plane (XY plane), the projections of the point and the direction on the plane are defined as the origin and X-axis and supports switching axes and reversing the axis direction.	▼ Required Image: Constraint of the system Face Z Point Signature Direction X
View plane	Only icon is updated.	Required Image: Constraint of the second sec
Dynamic	Only icon is updated	▼ Required Image: Constraint of the system Position X axis Y axis Y axis Z axis

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

1.10.2 Datum Plane Improvement

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

The improvements show as below:



Method	Main Improvement	lcon
Geometry	The method supports user selecting at most three reference geometric objects at one time. The system will automatically analyze the "Constraint" relationship between geometry and the creating datum face and highlight the constraint icon. The supporting geometries include points, lines, edges, axes and faces.	Datum Plane Image: Constraint of the second seco
Offset face	The new method supports user creating a datum plane by specifying a plane or datum plane to offset and supports to reverse the normal of datum plane.	Note Note Image: Constraint of the state
Angle to face	The new method supports user specifying reference plane, rotation axis and angle to create a datum plane which is at an angle to the reference plane and supports to reverse the normal of datum plane.	Datum Plane Required Face F2 Axis E5 Angle 45 deg 2
3 point plane	The upgraded method supports user to specify at most three points to create a datum plane. Next solution is available for single point and the default normal of the created datum plane can be along with the three axes. Support flipping the normal of the datum plane.	Datum Plane Required Required Image: Second



On curve	The new method supports user specifying reference curve/edge to create datum plane and supports to control the curve position via Percent and Distance. The direction type refers to the relation between datum plane and the reference curve, which includes Perpendicular, Tangent, Perpendicular to curve and Tangent to curve. Support flipping the normal of the datum plane.	Datum Plane
View plane	The icon is updated and flipping direction is supported.	Datum Plane Required Provide the second se
Dynamic	Only the icon is updated.	Datum Plane Required Required Position X axis Y axis <td< td=""></td<>

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

1.10.3 Datum Axis Improvement

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

The improvements show below:

ZW3D

Method	Main Improvement	lcon
Geometry	The upgraded method supports user picking at most 2 reference geometry objects and the system will automatically analyze the "Constraint" relationship between the picked geometry and the creating datum axis and highlight the corresponding constraint icon. The reference geometry includes points, lines, edges, axes, and surfaces.	✓ Datum Axis ⊠ ✓ X ☑ ✓ Required ☑ ✓ Geometry 1 picked Length 10 The second secon
Center axis	The upgraded method can create a datum axis through plane curve axis.	✓ Datum Axis Similar ✓ X Image: Constraint of the second
2 points	Only the icon is updated.	✓ Datum Axis ⊠ ✓ X ☑ ✓ Required ✓ Ist point ✓ * 2nd point ✓ *
Point and direction	The upgraded method adds the direction types including Parallel and Perpendicular. User can specify point and direction and create a datum axis according to the specified direction type.	✓ Datum Axis ⊠ ✓ X Image: Constraint of the system ✓ Required ✓ Image: Constraint of the system Ørigin
Intersected faces	Keep unchanged.	Datum Axis Required Ist face 2nd face Length



Angular bisector	The new method supports user specifying two intersected lines. The intersected point is as the start of datum axis while the angular bisector or supplementary bisector is as the direction of the datum axis.	✓ Datum Axis ⊠ ✓ X II IIII ✓ Required ✓ ✓ IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII
On curve	The new method supports user creating a datum axis by specifying reference curve/edge and controlling the curve position by Percent and Distance. The direction type refers to the relationship between datum axis and reference curve including Tangent, Perpendicular, Parallel to curve and Perpendicular to curve.	Datum Axis Required Required Percent Distance Percent Distance Percent S0 Served Direction type Tangent Length

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

1.10.4 New Auto Size in Datum Plane

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

Option	Definition	Sample
Default	Use default size to display datum plane	



Auto size	Automatically use the plane that datum referred as the size of datum plane	
Reference	User can define random plane as the reference size for datum plane	
Size	User can customize the width and height for the datum plane	Auto size Width 15 Height 10

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

1.11 UI Efficiency Optimization

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

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



1.12 **★**Thickness Analysis Improvement

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

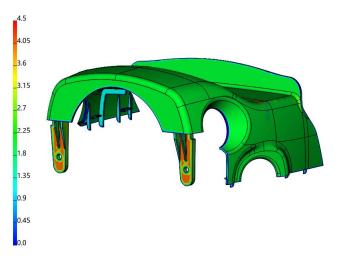
1) New thickness analysis method: Ball rolling

Analysis Settings				
Method	Ball rolling 🔹			
Resolution	2.516			

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

▼ Analysis Settings				
Method Ball rolling				
Resolution	2.516			

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



→ Where it is

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



1.13 Assembly Tree Manager Improvement

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

Assembly Tree Attributes Manager				Φ Σ
All Attributes Type Assembly Vasembly Num Standard Uniq Physical Com File Upda User Layer Color Display Mode	*	Displayed Name Assembly Node Constraint Status Part Configuration	Assembly Assembly Assembly	Width 200 100 100
	Defa	Width 100 🛟	ОК	Cancel

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

→ Where it is

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

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

1.14 Display Improvement

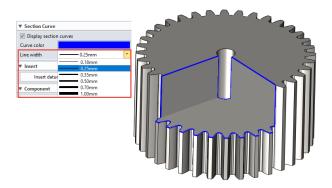
1.14.1 Section Improvement

1.14.1.1 New Line Width to Control Section Curve

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

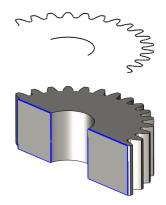






1.14.1.2 New Clip Wire

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



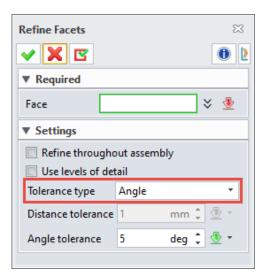
→ Where it is

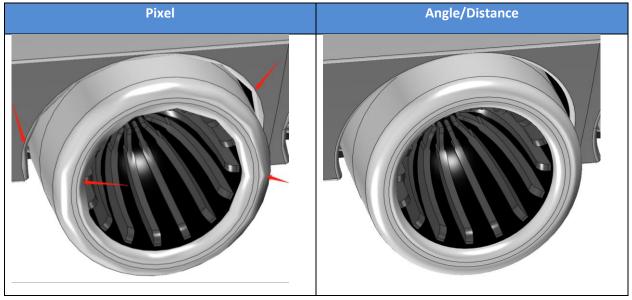
Part Environment >> Inquire >> Inspect Model >> Section

1.14.2 New Angle and Distance Tolerance Type

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

ZW3D





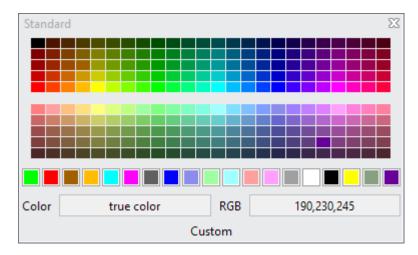
→ Where it is

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

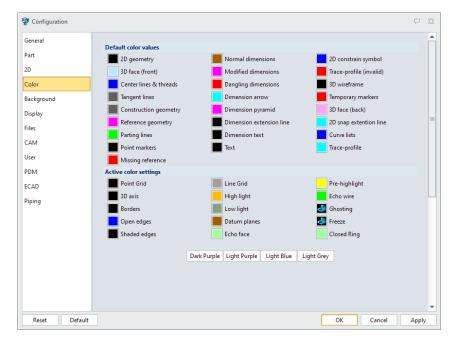
1.14.3 **★**Color Improvement

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





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



1.15 Usability Improvement

1.15.1 New Mouse Gesture and Shortcut Bar

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



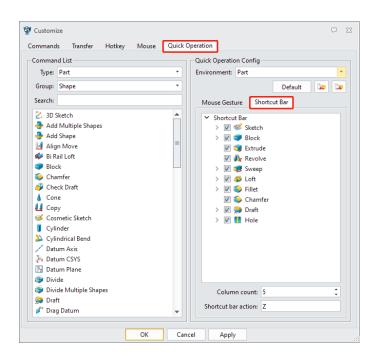


In Menu >> Tools>> Customize, open the Customize panel as below figure. User can drag the command icons by left-clicking the mouse to add or move commands from "Mouse Gesture".

😨 Custom Command		Hotkey	Mouse	Quick (Operation			Ģ	23
Commar Type: Group: Search: Search: Add Add Add Add Add Add Add Cor Che Cos Cos Cos Cos Cos Cos Cos Cos Cos Cos	d List Part Part Shape Sketch d Multiple Shapes d Shape d Multiple Shapes d Shape d Multiple Shapes d Shape d Multiple Shap d Multiple Shap d Multiple Shapes d Multiple Shape		Mouse	Queck €	Quick Operation Environment: P	srt Short	Default cut Bar		
			ОК	Can	cel Apply				

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





Mouse Gesture: Shift + Right Mouse Click

Shortcut Bar: Press shortcut key Z

1.15.2 New Activated Method Zoom All

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

1.16 Attributes Improvement

1.16.1 Support Adjusting User Properties Order

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

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



F	Properties							Ċ
St	tandard User	-	File Preview					
	Property name	Туре	Sub-type		(pression		Value	Unit/Format
1	A	String *		AA		AA		
2	с	String *		CC	<i>₫</i> ∑	сс	🧳 Delete	Item
3	В	String *		BB	<u>A</u>	BB	Insert	ltem
1	<add a="" item="" new=""></add>						Тор	
							Move	· ·
							Move	
							Botto	m
_								
								🔹 📴 📴

Part Environment >> Attributes >> Properties >> User

1.16.2 Batch Attribute Edit Improvement

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

🦉 Batch Attribu	te Edit						\Box	23
Load Config	1k\2611x64\supp\Te	mplate.z3preset	V			Pick objects to edit		
🗹 Include	Attribute	Value			ID	Name		
	Number	ZWSoft123	-					
	Class	Part	-					
	Designer	Mark	-					
	Cost	12.99	-					
	Supplier	ZWSoft	-					
	Manager	Jason	-					
	CustomAttribut1	User Attrib1	-					
Material			-	🗹 Re	main curren	t list		
Density		kg/m^3	*	Num	ber * =	Т	arget	Q
Part size	L x W x H					Rename picked shapes		
	Auto calculate			Cle	ear listed obj	ects after update		
Stock size				🔲 Hi	de listed obj	ects after update		
	Auto calculate		6					
		ОК	Ca	ncel	Apply			



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

1.16.3 Attributes Setting Improvement

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

Standard User	File Preview			Drawing001.Z3DRW •
Name	Drawing001			Browsing this object
Number		Designer		
Manager		Supplier		Drawing001
Keywords	Each keyword should be separated by	/ a "," comma		Objects and variables
Description	At most 95 characters			part_material
Create time	2021/3/18		Short date	part_startdate part_enddate
Last modified	2013/1/29		Show time	part_derived part_density part_filepath
				part_filename
				part_sheetcode part_SMT_Thickness
				part_SMT_Kfactor
				part_SMT_Length
				part_SMT_Width
				prop_name prop_number
				prop_designer
				prop_manager
				prop_supplier
				prop_keyword prop_description
				<le>left-click> to browse/select</le>

→ Where it is

Menu Bar >> Attributes >> Properties

1.17Configuration Improvement

1.17.1New Layer On/Off Option

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

ZWED

Configuration				C	- 33	
General	Window restoration	Standard *	55 Wile differences		•	
Part	3D input device	Spaceball	20 mile thickness			
2D	Pick aperture size	Large 🔻	Construction line	•		
	Zoom reference point	Cursor -	Hidden line style			
Color	Rotation center	Auto 🔻	Hidden line fade	80		
Background	Rotation center display mode	Show on motion	U isolines	1		
Display	Anti-Alias	Wire -	V isolines	1		
Files	Highlight mode	Color -	Interpolation point size	0.001		
CAM	Inactive field highlight mode	Edge 🔻				
	Display mode	Shade 🔻				
User	Display OutofScope parts	Transparent *				
PDM	Cursor size	Default *				l
ECAD	Tangent edge display	Visible 🔻				
Routing	Tolerance type	Pixel 🔻				
	Layer on/off	Ask users 🔻]			
			- -			
					=	
					-	1
Reset Default			ОК	Cancel Ap	ply	

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

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

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

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



→ Where it is

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





1.17.2 New Default Template

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

eneral	Basic					
rt	Object template file D:	:\2700\languages\	en_US\resourd	ce\	-	
)	Default Part template				-	
olor	Default Assembly template				-	
ckground	Default Sheet Metal template				-	
splay	Default Sheet template				-	
es	Default CAM template				-	
	Naming for new file Do	efault localization	•			
AM	Attribute bundles file Bu	undles.Z3			-	
er	Default session name .S	ession				
M	Temporary folder ata	a\Roaming\ZWSO	FT\ZW3D\270	0\output\temp	-	
AD	Internet browser ies	xplore.exe			-	
outing	ZW3D file folder C:	:\Users\Administra	ator\Documen	nts\ZW3D	-	
-	Save last open folder as next default directory					
	Working folder C:	\Users\Administra	ator\Documen	nts\ZW3D		
	ZW3D backup folder C:	:\Users\Administra	ator\Documen	nts\ZW3D_bak	-	
	Save backup files in the same location as the original					
Reset Default Configuration		[ОК	Cancel	Apr	ply
		[ОК	Cancel		oly
Configuration	Basic]	ОК	Cancel		oly
Configuration		[ОК	Cancel		ply
Configuration eneral	Basic]	ОК	Cancel		ply
Configuration ineral int	Basic Default ECAD template]	ОК	Cancel		oly
Configuration Ineral Int	Basic Default ECAD template Board		ОК	Cancel		oly
Configuration eneral rt polor ckground	Basic Default ECAD template Board Board color Component		ОК	Cancel		oly
Configuration eneral rt b) blor ckground splay	Basic Default ECAD template Board Board color Component Component color		ОК	Cancel		oly
Configuration eneral rt plor cckground splay es AM	Basic Default ECAD template Board Board color Component	CAD	ОК	Cancel		oly
Configuration eneral rt olor cckground splay es MM er	Basic Default ECAD template Board Board color Component Component color	CAD	ОК	Cancel		oly
Configuration eneral rt olor cckground splay es AM er DM	Basic Default ECAD template Board Board Board color Component Component color Component library C:\Users\Administrator\Desktop\EG Area		OK	Cancel		oly
Configuration eneral rt blor cckground splay es sM M er M	Basic Default ECAD template Board Board Board color Component Component color Component library C:\Users\Administrator\Desktop\E0	CAD Keep-out	OK	Cancel		oly
Configuration eneral rt olor cckground splay es AM er DM	Basic Default ECAD template Board Board Board color Component Component color Component library C:\Users\Administrator\Desktop\EG Area		OK	Cancel		
Configuration eneral rt blor cckground splay es sM M er M	Basic Default ECAD template Board Board Board color Component Component color Component library C:\Users\Administrator\Desktop\EG Area		OK	Cancel		oly
Configuration eneral rt blor cckground splay es sM M er M	Basic Default ECAD template Board Board Board color Component Component color Component library C:\Users\Administrator\Desktop\EG Area		OK	Cancel		oly
Configuration eneral rt blor cckground splay es sM M er M	Basic Default ECAD template Board Board Board color Component Component color Component library C:\Users\Administrator\Desktop\EG Area		OK	Cancel		oly

→ Where it is

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

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



1.17.3 Ref Point & Pick from List Timer Improvement

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

💯 Configuration		23
General		
Part	General	
2D	Merge shapes before cut	
Color	Animate assembly alignment	
	Calculate mass properties on demand	
Background	Calculate mass properties with open shape	
Display	Enable "Snap Pick"	
Files	Display import/export status window	
CAM	Auto check topology error after import	
User	✓ New assembly mode	
	Enable multi-shell explicit shape features	
PDM	Only exact match in history regeneration	
ECAD	Ignore regeneration errors and keep history playing	
Piping	Save display data with part states	
	Backup geometry data of each feature	
	Backup data for history quick rollback	
	🔲 Backup data for feature quick highlight	
	Ref point & pick from list timer 0.1 🗘	-
Reset Default	OK Cancel App	ly

→ Where it is

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

1.17.4 Opening Efficiency of Big File Optimization

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

Lightweight loading		
Enable lightweight cache		
Include curves in cache	Include edges in cache	
Use custom folder, otherwise same as open file	C:\Users\zw3d\Documents\ZW3D\GR	

If "Enable lightweight cache" is checked, it will:

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



- 2) Load cache file if there is cache file when open file.
- 3) Automatically generate cache file when save file.

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

→ Where it is

Configuration >> Files >> General >> Lightweight loading

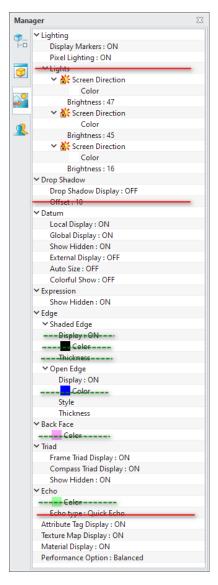
1.17.5 New "Visual Manager" Configuration

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

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

ZW3D





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



Mana	ager		e XX	
•	✓ Lighting			
Fo	Display Markers : 0	ON		
	Pixel Lighting : Of	1		
	✓ Lights			
~	🗸 😽 Screen Dire	ction		
	Color			
•	Brightness : 4			
	🗸 💥 Screen Dire	ction		
2	Color			
-	Brightness : 4			
	✓ X Screen Dire	ction		
	Color	Set Shaded Edge Color		Σ
	Brightness : 1	° 🗸 🗹		
	✓ Drop Shadow			
	Drop Shadow Disp Offset : 10	Color		
	✓ Datum	RGB O HSL		
	Local Display : ON	R 0 1		-
	Global Display : O	· · · ·		_
	Show Hidden : ON			
	External Display :	B 0 ‡		*
	Auto Size : OFF	Edge color from face		
	Colorful Show : O	-		
	✓ Expression Set	Open Edge Color		23
	Show Hidden			
	🕆 Edge 🛛 💙			
	✓ Shaded Edge C	olor		
	Display : O	RGB O HSL		
	Color	R 0 1	_	
	✓ Open Edge			
	Display : O	G 0 ‡	_	
	Color	B 255 ‡		
	Style	Edge color from face		
	Thickness			
	✓ Back Set Back Face	Color	23	
	🗸 🗙 🔽	2	0	
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1.17.6 Background Improvement

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

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

ZW3D

🖗 Configuration		₽ %
General	Environment	
Part		
2D	Part/Assembly Apply to all environments Part/Assembly	
Color	s 2D Drawing CAM	=
Background	Color	
Display	Gradient background	
Files	Enable gradient background Upper left Upper right	
САМ	Lower left Lower right	
User	Background image	
PDM	Enable background image	
ECAD	Display method Center 🔹 (mm) 0 🛟	
Piping	Image filename	
	Dark Purple Light Purple Light Blue Light Grey	
	Background display as config	-
Reset Default	OK Cancel	Apply

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

→ Where it is

Configuration >> Background

1.17.7 New Background Modification Function

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



🐲 Background	\overline{a}	23
Solid background		
Color		
Gradient background		
Enable gradient background		
Upper left 📃 Upper right		
Lower left 📃 📕 Lower right		
Background image		
Enable background image		
Display method Center 💌 (mm) 0	÷	:
lmage filename		-
Apply config to file		
OK Cancel		

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

→ Where it is

All Environments >> Tools >> Settings >> Background

1.18 Other

1.18.1 New Function Expression in "Function List"

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

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

The function has the following characteristics:

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





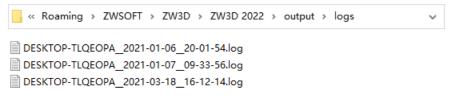
- If both X and Y are negative, the absolute value rounded down away from 0.
- If X is negative and Y is positive, the absolute value rounded up away from 0.
 Sample:

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

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

1.18.2 New Logs Saved Number

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



→ Where it is

User Directory >> output >> logs

1.18.3 ZW3D 2023 Will Give Up Supporting 32-bit Version

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



2 Translator

2.1 Input Format Update

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

Format Postfix		Version Supported 4.1.9 – 4.2.4		
Catia V4 .model, .exp, .session				
Catia V5/V6	.CATPart, .CATProduct, .CGR	V5R8V5/V6R2021		
Catia_2D	.CATDrawing	V5R8V5/V6R2021		
NX(UG)	.prt	11– NX 1980		
Creo(Pro/E)	.prt, .prt.*, .asm, .asm.*	16 – Creo 7.0		
SolidWorks	.sldprt, .sldasm	98– 2021(only 64-bit supported)		
SolidWorks_2D	.slddrw	2013-2021(only 64-bit supported)		
SolidEdge	.par, .asm, .psm	V18 – SE2021		
		.ipt (V6 – V2022)		
Inventor	.ipt, .iam	.iam (V11 – V2022)		
ACIS	.sat, .sab, .asat, .asab	R1 – 2021 1.0		
DWG	.dwg	R11 -		
DXF	.dxf	R11 -		
IGES	.ige, .iges			
STEP	.stp, .step, .stpz	203, 214, 242		
Parasolid	.x_t, .x_b, .xmt_txt, .xmt_bin	Up to 30.0		
VDA	.vda			
Image File	.bmp, .gif, .jpg, .jpeg, .tif, .tiff			
Neutral File	.z3n, .v3n			
PartSolutions	.ps2, .ps3			
STL	.stl			
3DXML	.3dxml	4.0 - 4.3		
JT	jt	6.4-10.4		
OBJ	.obj			



2.2 Lightweight Model Support Batch Loading

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

			- × 🔏 🔪	asm4			
> 🔽 🚣 🎙 (-)_asm2			> 🔽 🕌	🛓 🎙 ()_asm2			
> 🔽 🛃 🎙 (-)_asm3			> 🔽 🕌				
✓) (−)_model2				🄰 🎙 (–)_model2	_		
✓ ● \ (-)_mode ¹² ✓ ● \ (-)_mode ¹⁷	Auto Regen	1		D 🐧 (-)_model2	To	Auto Regen	
> ✓ 🚑 🎙 (−)_asm1 🐴	Configure Component			🔰 🎙 (-)_model1 🛓 🎙 (-)_asm1_2		Configure Component	
			* ¥		15	Group as Sub-assembly	
4	Assembly Folder				4	Assembly Folder	
42 24	2 C				22	Move	
1	Suppress					Suppress	
	Blank					Blank	
	Show Only				4	Show Only	
21	Pack				±1	Pack	
					-	Merge	
	3				٩	Fix	
5	Unblank External Datum				5	Unblank External Datum	
3					\$	Copy/Move to Layer	
					23	Make Virtual	
•	Display >					Display	
	, spidy	-					_
-	Face Attributes					Face Attributes	
4	Inherit Attributes				4	Inherit Attributes	
*	Cut				8	Cut	
le le	Сору				Þ	Сору	
<u> </u>	Paste				Ê	Paste	
4	Erase				ø	Erase	
se 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 1990 - 199	Load All Models				٩	Load All Models	
8	Load Current Model					Load Current Model	
8	Export				8	Export	
	Customize menu					Customize menu	

→ Where it is

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

2.3 New "Specify part or assembly" in Step File Import

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

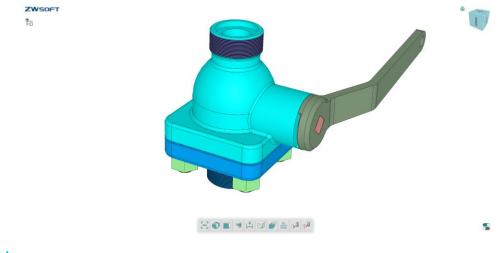


🖗 Step Fil	le Import 🖓 🖾			
Step				
▼ Impo	ort from			
File	File H:\LZH\Case\CAD\STEP\006.stp			
▼ Impo	ort to			
	rent object v object v file			
	Save to H:\LZH\Case\CAD Assembly			
▼ Gene	eral			
C Auto	✓ Auto sew geometry ✓ Auto activate part △ Auto create sub-part ✓ Break trim edges □ Force close □ Sew unique parts □ Read hidden entities □ Keep origin face direction			
	Default OK Cancel			

File >> Import >> Import >> STEP

2.4 Export Html Format Improvement

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



→ Where it is

File >> Import >> Html



3 CAD

3.1 Sketch Design

3.1.1 Quick Dimension Supports Perimeter Dimension of Curves

We added perimeter dimension of curves to Quick Dimension.



→ Where it is

Shape >> Sketch >> Dimension >> Quick Dimension

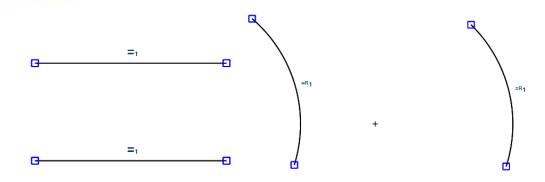
3.1.2 Equal Length/Radius Improvement

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

File Equal Length	প)⇒) Equal Radius 😂
• 🗙 💿 👌	• 🗙 💿 👌
▼ Required	▼ Required
Curve 👲	Arc/Circle
Curves 🛛 🕹 💆	Arcs/Circles 🛛 🗧 💆
Group selection	Group selection

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



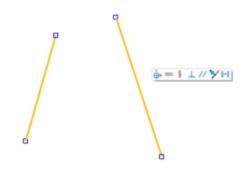


Sketch Environment >> Constraint >> Equal Length

Sketch Environment >> Constraint >> Equal Radius

3.1.3 **★**New Smart Constraint Inference

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



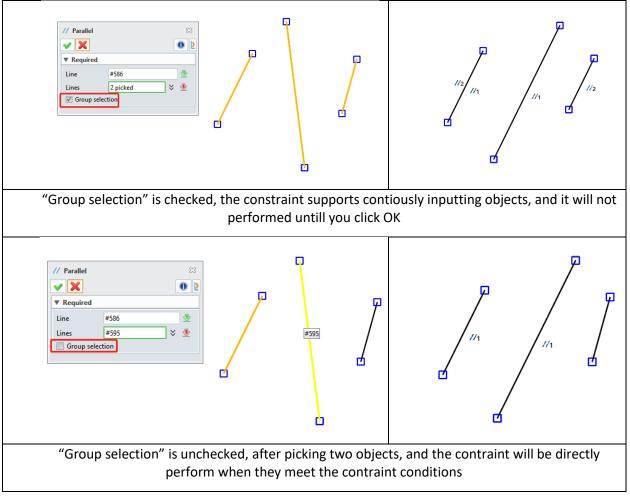
→ Where it is

Sketch Environment >> Picked Objects >> Constraint Toolbar

3.1.4 Constraints Support Group Selection

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





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

3.1.5 New Edit Week Dimensions

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



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▼ Required	
Weak dims	* 👲
Onvert to driving ○ Delete	

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

3.1.6 New Section On/Off in Sketch

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

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

→ Where it is

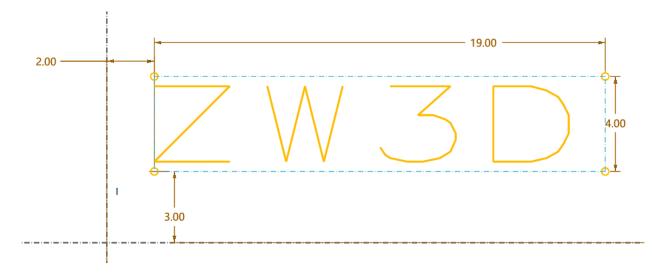
Sketch Environment >> DA Toolbar >> Section on/off

3.1.7 Sketch Box Text Supports Dimensions

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



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

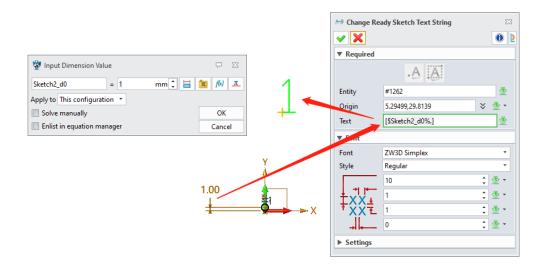


→ Where it is

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

3.1.8 ReadySketch Improvement

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



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



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Symmetry		
Second direction	on	
Variable spacin	ig Sho	w table
Base point		
▼ Variable Patter	n	
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Parameter	Sketch2_d0	₫
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Orientation		
Boolean		
▼ Setting		
Exclude Base a	s instance	

Sketch Environment >> Drawing >> ReadySketch

3.1.9 + 2D/3D Sketch External Reference Improvement

3. 1. 9. 12D Sketch "External Reference" Improvement

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

3.1.9.1.1 Projection Mode

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

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

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





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

📂 Reference	23
✓ X	0
▼ Required	
🧕 Single 🔘 Chain 🔘 Loop	os 🔘 Point
Entity	*
Construction geometry	

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

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

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

🟓 Refe	erence	23			
~	K	0			
▼ Requ	iired				
O Sing	Single O Chain Coops O Point				
Mode	Chain pick	•			
	Entity Chain between picks				
Construction geometry					

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



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

All: Input all loops onto the faces.

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

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

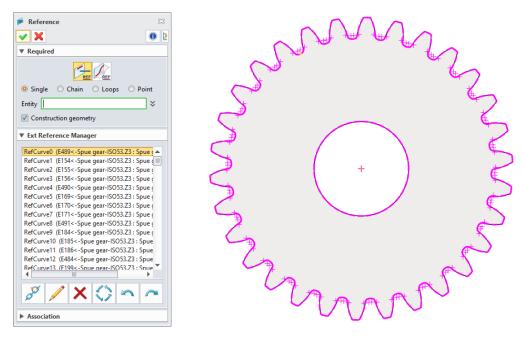
Inner: Input the inner loops of the picked face.

Outer: Input the outer loops of the picked face.

Open: Input the open boundary of the picked face.

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

Silhouette: Input the outer contour of the picked face.

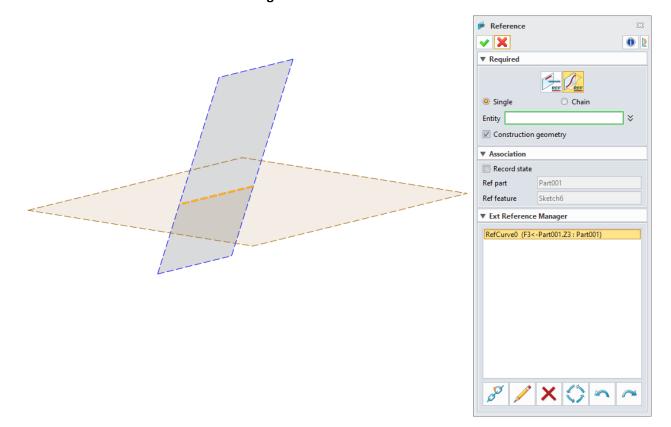


3.1.9.1.2 Intersection

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



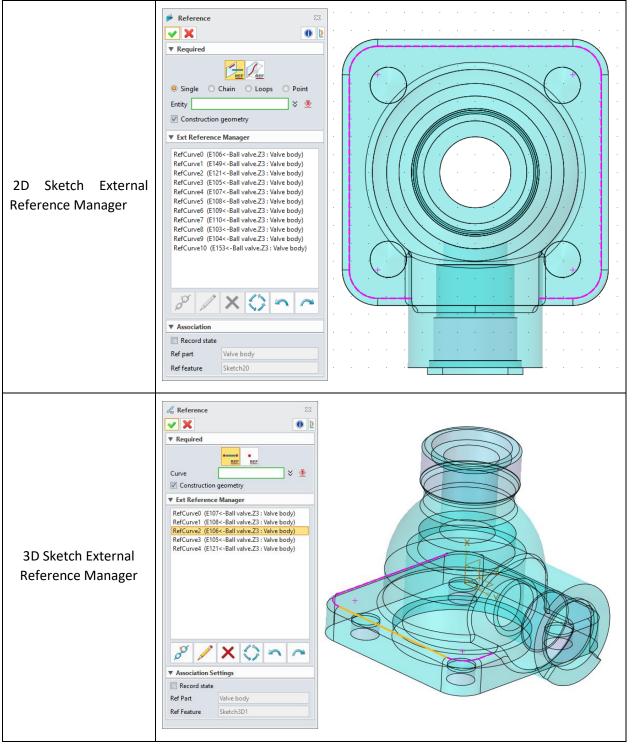
The input objects of intersection mode mainly include lines, faces, datum planes, and datum axes. There are two rules of intersection mode: **Single** and **Chain**.



3.1.9.2 Ext Reference Manager

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

ZW3D



→ Where it is

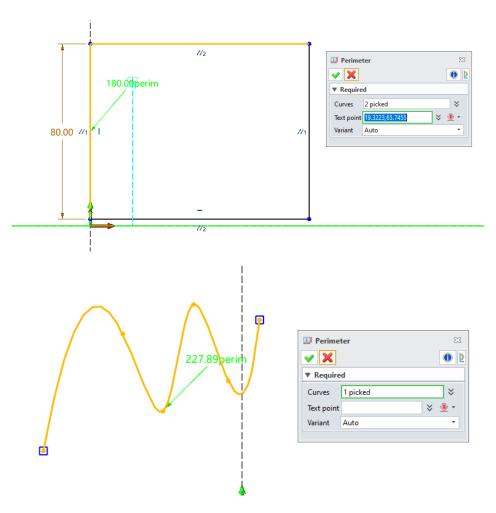
Sketch Environment >> 2D/3D Sketch >> Reference >> Reference

3.1.10 Dimension Improvement

3.1.10.1 New Perimeter Dimension

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

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



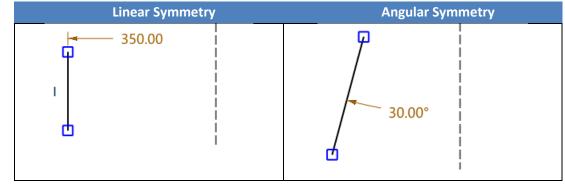
→ Where it is

Sketch Environment >> Sketch >> Dimension >> Perimeter



3.1.10.2 New Symmetry Dimension

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

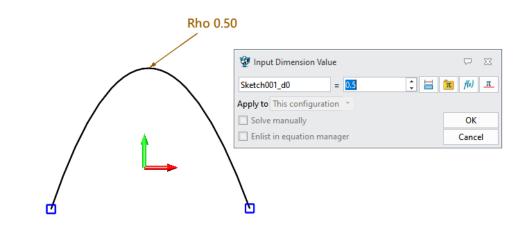


→ Where it is

Sketch Environment >> Sketch >> Dimension >> Symmetry

3.1.10.3 Support Adjusting Rho Value in 3 Point Conic

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



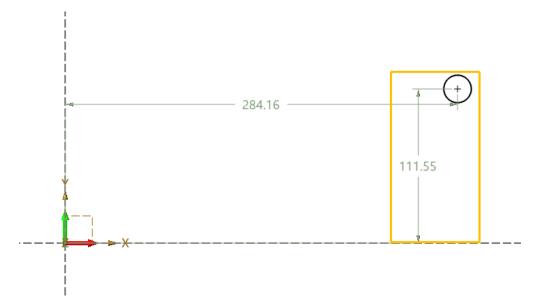
→ Where it is

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



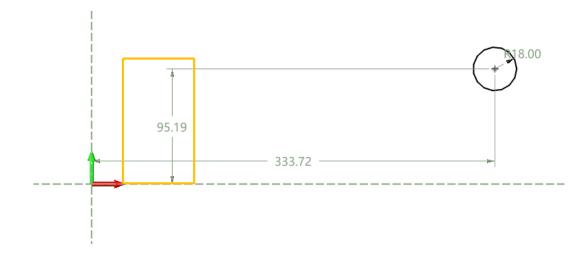
3.1.10.4 Weak Dimension Improvement

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



The improved weak dimension closes to the drawing figure.

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



→ Where it is

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

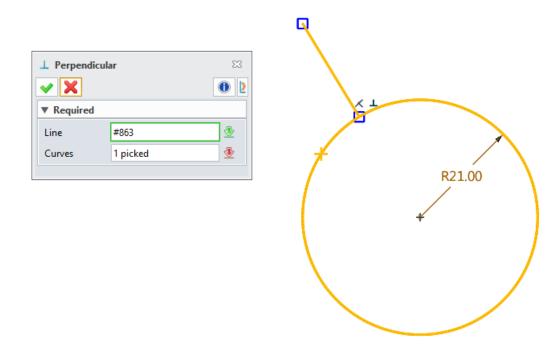




3.1.11 Constraint Improvement

3.1.11.1 New Vertical Constraint Between Line and Circle (Arc)

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



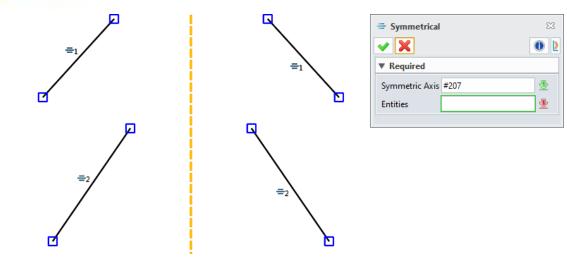
→ Where it is

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

3.1.11.2 Symmetrical Constraint Improvement

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

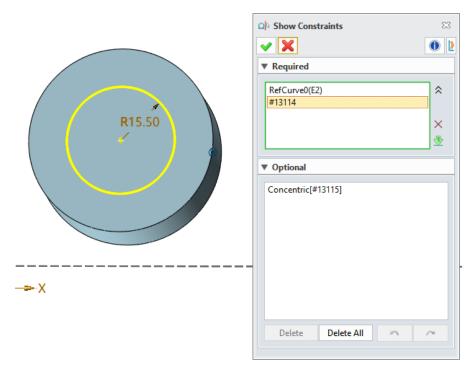




Sketch Environment >> Sketch >> Constraint >> Symmetrical

3.1.11.3 Quickly Search Constrained Object

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

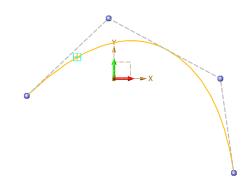




3.1.12 Sketch Spline Improvement

3.1.12.1 New Through Point During Spline Drawing

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



3.1.12.2 Combination of Through Point and Control Point into Spline

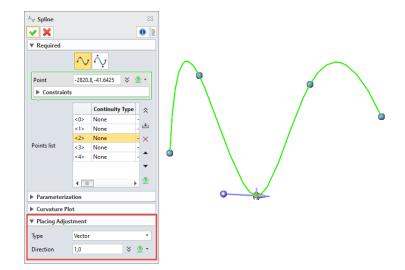
Through Point and Control Point splines are frequently used in the sketch environment. User often needs to switch the spline type during the drawing. Thus, we combined Through Point and Control Point into a new command "Spline" to better switch the types and reduce the repeated action.

⊷ Spline		23
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Placing Adjust	tment	
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3.1.12.3 New Placing Adjustment

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





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

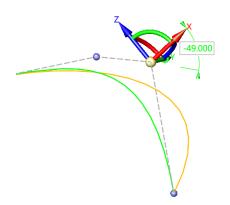
When through points and control points are adjusted, "Placing Adjustment" does not work but utilizes the rules of previous version by default.

2) Vector

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

3) WCS (3D sketch)

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



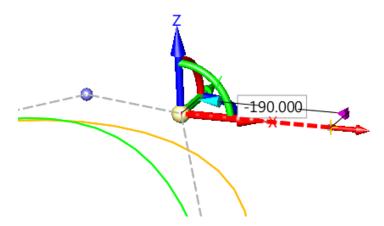
4) Plane (3D sketch)

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



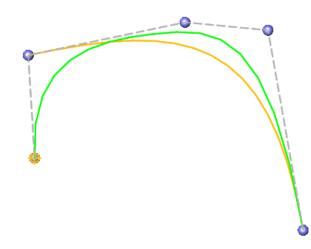


User can adjust the operator posture with dragging it in arc line. By checking "Project to plane", the relevant nodes can be moved to the projected point of the plane and the adjusted nodes are on the specified plane.



3.1.12.4 New Confirmed Points Spline Preview

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



3.1.12.5 Control Point Spline Degree Up to 20

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



∼ Spline		23
✓ X		0
▼ Required		
	$\sim \dot{\sim}$	
Point		¥ 👲 •
► Constraints		
Points list		≝ ¥
▼ Parameteriza	tion	
Degree	20	¢
Single segme	ent	
Curvature Plo	t	
Placing Adjust	tment	
Туре	None	•
Direction		🗧 🛬 ד
4	1111	•

3.1.12.6 Spline Points List Optimization

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

y Spline					0
Required					
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Point				* ₫	<u>k</u> -
► Constraints					
		Continuity Type	Point	Tangent direction	â
	<0>	None	-72.3278,2.3753,0	-	
	<1>	None	-51.9002,28.5036,0		⊻
	<2>	None	10.3325,47.981,0		X
Points list	<3>	None	44.5368,9.73872,0		
					Ŧ
					3
Parameterization					
Degree	20				÷
Single segment					
Curvature Plot					
Placing Adjustment					
Туре	None				•

→ Where it is

Sketch Environment >> Sketch >> Curve >> Spline



3.1.12.7 New Convert to Through Point Curve

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

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Curve Cv4560 Point		<u>∛</u>	
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Align plane		-	0 0
Use orientation tool			
Modification			6
Convert to Throug	h Point Curve		
Control points			
Add	Delete		
Modify position	Modify weight		
Reduce	Refit		
Modify the curve globally			
▼ Curvature Plot			
Display			ł
Display curvature comb			
Density 150.000 💲		-	
Show hints for modification	ons		
Display inflection points			

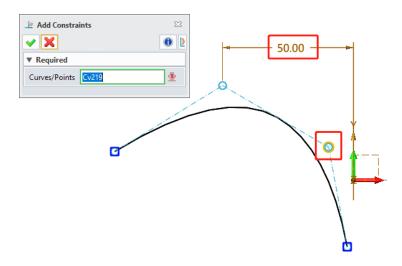
→ Where it is

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

3.1.12.8 Dimension and Constraint Support Picking Control Point

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

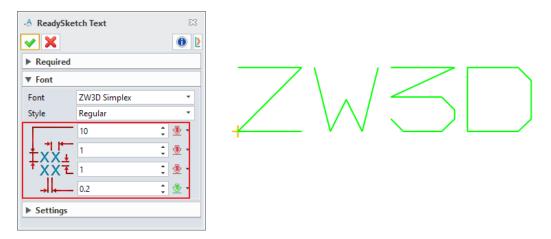




3.1.13 ReadySketch Text Improvement

3.1.13.1 New Font Style Setting

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



3.1.13.2 New Keep Text Shape

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





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

3.1.14 Ellipse Improvement

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

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1st point 79.12	264,21.2573	🗧 💆 🕶				
2nd point 116.9	61,21.2573	՝ 🕹 ד		/		
3nd point 100.4	169,34.7003	՝ 🕹 ד	26.886 -			
▼ Dimensions						
Width	37.83449 n	nm 🗘 🔐		\mathbf{X}		
Height	26.88604 n	nm 🗘 🔐	<u> </u>			
Start Angle	0 deg	g 🗘 垫 👻			— 37.83 45 	
End Angle	0 dec	g 🗘 👲 🗝				

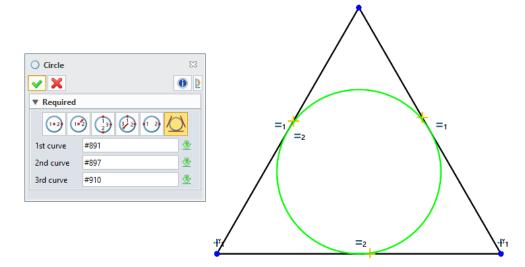
→ Where it is

Sketch Environment >> Sketch >> Drawing >> Ellipse

3.1.15 New 3 Tangent Circle

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





Sketch Environment >> Drawing >> Circle

3.1.16 Support Creating Direction in Center Arc Slot

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

🔇 Slot		×	+
▼ Required	d		
	S & S S		
Center	0,0	× 🕭 -	
1st center	0,26.9917	× 🕭 -	
2nd center	13.2868,-23.4949	× 🕭 -	
O Clockw	ise 🧿 Counter		
Radius	🔘 Diameter		-5
Radius	5 mm	‡ ₫ •	

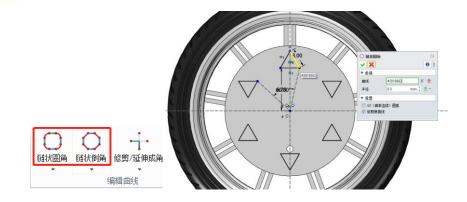
→ Where it is

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

3.1.17 Fillet Chain/Chamfer Chain Support Picking End

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

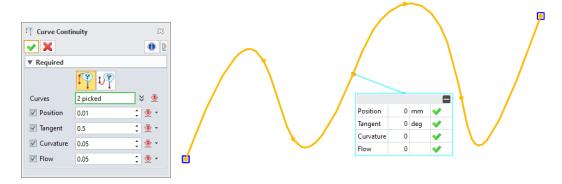




2D/3D Sketch Environment >> Sketch >>Edit Curve >> Fillet Chain/Chamfer Chain Part Environment >> Wireframe >>Edit Curve >> Fillet Chain/Chamfer Chain

3.1.18 ★Sketch Curve Continuity Inquire Improvement

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



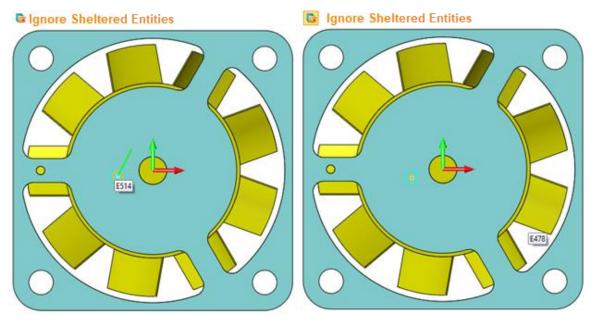
→ Where it is

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

3.1.19 New Ignore Sheltered Entities

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



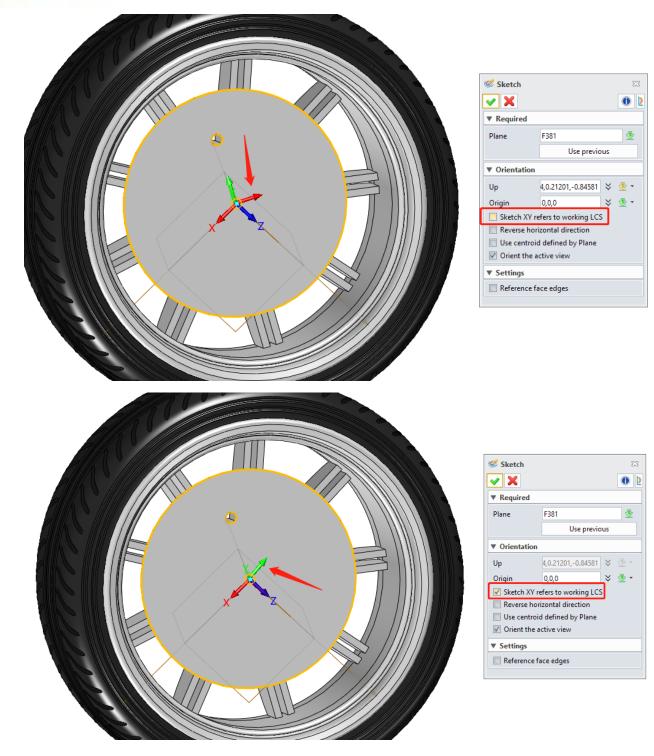


Sketch Environment >> DA Toolbar >> Ignore Sheltered Entities

3.1.20 New Sketch XY Refers to Working LCS

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



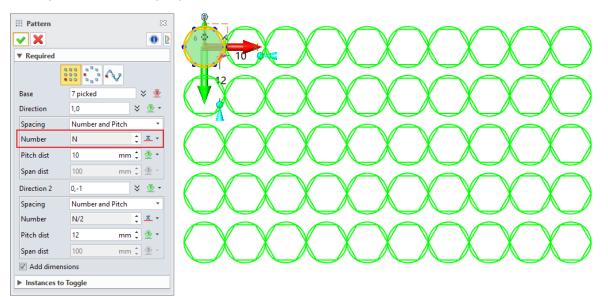


Part Environment >> Shape >> Basic Shape >> Sketch



3.1.21 Sketch Pattern Improvement

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



→ Where it is

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

3.1.22 Dimension Editor Improvement

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

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

ZW3D

rfi.	🛱 Dimension Editor 🛛					
V	X			0 2		
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Dii	mensior	n list <mark>4 picked</mark>		₫		
•	Optiona	ıl				
	Туре	Name	Value			
1	S	Sketch1_d2	21.68032			
2	\$	Sketch1_d3	115.95410			
3	<u> </u>	Sketch1_d0	30.32150			
4	<u> </u>	Sketch1_d1	57.32320			
	Increment 0 mm ↓ ★ ▼					
	Regene	erate				

→ Where it is

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

3.1.23 Power Trim Improvement

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

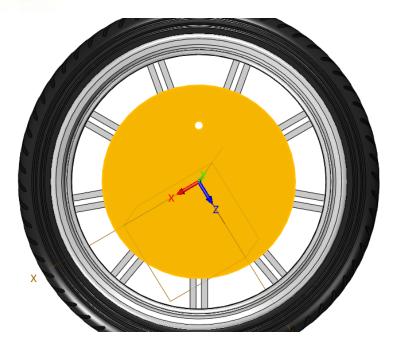
→ Where it is

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

3.1.24 Sketch Relocate Default Activation Update to "Plane"

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





Relocate					
▼ Required					
Plane	F381		- 🕹		
Up	0.48954,0.21201,-0.84581	$\stackrel{\scriptstyle \sim}{}$	-		
Origin		$\stackrel{\scriptstyle \sim}{}$	• 🛃		
Reverse horizontal direction					
Use centroid defined by Plane					
Orien	t the active view				

Sketch Environment >> Sketch >> Settings >> Relocate

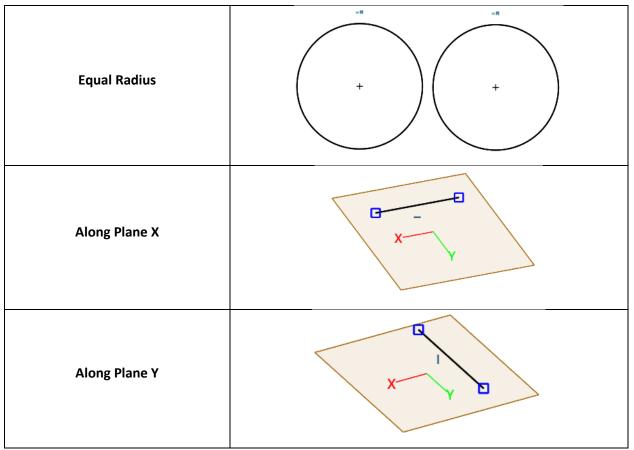
3.1.25 **★**3D Sketch Improvement

3.1.25.1 New Constraint Type

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

Constraint	Effect
Equal Length	

ZWBD



→ Where it is

3D Sketch Environment >> Sketch >> Constraint

3.1.25.2 Support Automatically Creating Constraint

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

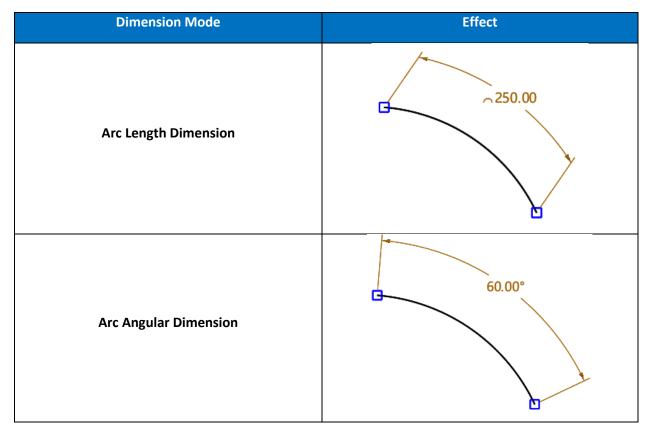


O Circle	
✓ X	
Center 1 picked > ✓ ● Radius ○ Diameter	R50.00
Radius 50 mm ‡ ∰ ▼ ▼ Settings	
Align plane Plane0 🔮	

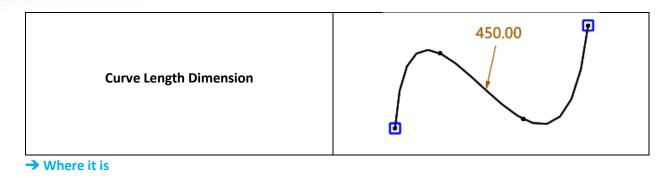
3D Sketch Environment >> Sketch >> Drawing

3.1.25.3 New Dim Modes

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



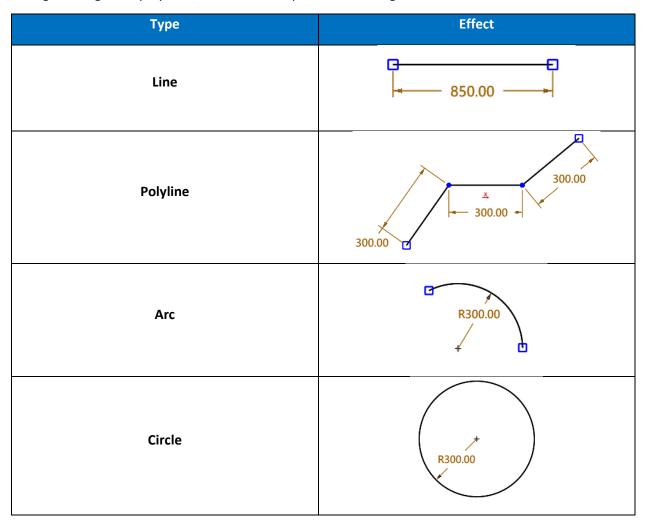
ZW3D



3D Sketch Environment >> Sketch >> Dimension

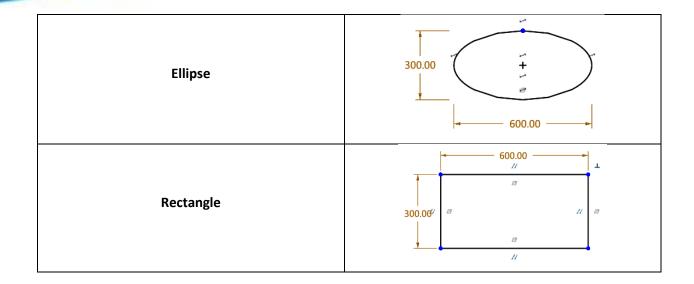
3.1.25.4 Support Automatically Creating Dimension

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



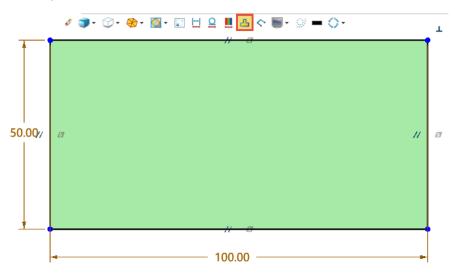
All-in-one, affordable CAD/CAM





3.1.25.5 New Closed Rings On/Off

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



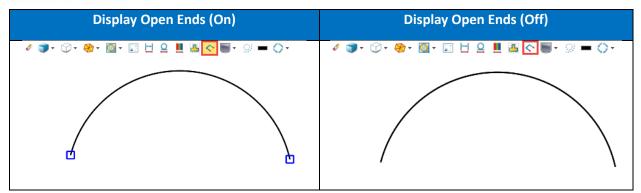
→ Where it is

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

3.1.25.6 New Display Open Ends On/Off

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





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

3.1.25.7 New Drag

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

→ Where it is

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

3.1.25.8 3D Sketch Datum Plane Participates in Constraint

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

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Required	ĺ.			Z
Line	Flat 0		₫	
Lines	Default CSYS_XY	\×	٠	14

The key points as below:

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



 Auto constrain new geometry Auto dimension new geometry Automatically fix the datum plane 	🔽 Enable co	nstraint solve	r
	Auto con	strain new geo	metry
Automatically fix the datum plane	🗹 Auto dim	ension new ge	eometry
Material and the autom plane	🔽 Automati	cally fix the da	atum plane
	Reset	OK	Cancel

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

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

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



→ Where it is

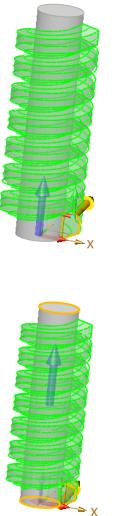
3D Sketch Environment >> Constraint

3.2 Shape Design

3.2.1 Spiral Sweep and Thread Improvement

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





鼬 Spiral Sweep	•		23	
🗸 🗙			0	
▼ Required				
Profile P	Sketch1		₫	
Axis A	0,0,1	\approx	• 🔄	
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Distance D	12	mm ‡	• 🗄	
Boolean				
▶ Lead				
▶ Offset				
▼ Settings				
Taper	0	deg 🇘	• 🗄	
Revolve clockwise				
Reverse spiral direction				
Auto Reduce				
► Tolerance				
🜺 Thread			23	
🗸 🗶 🖪			0	
▼ Required				
Face F	F3		₫	
Profile P	Sketch1		₫	
Turns T	7	* *	- 🗄	
Distance D	12	mm ‡	- 🗄	
Boolean				
Lead in/out				
▼ Settings				
Revolve clo	ckwise			
Reverse spiral direction				
► Auto Reduce				

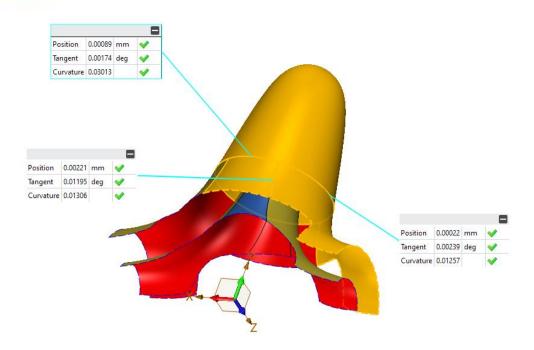
Part Environment >> Shape >> Basic Shape >> Spiral Sweep

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

3.2.2 ★Face Offset Supports G2

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

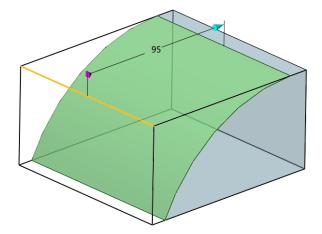




Part Environment >> Shape >> Edit Shape >> Face Offset

3.2.3 Fillet Improvement

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



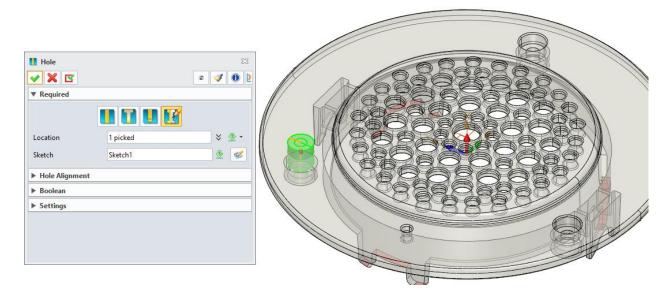
→ Where it is

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



3.2.4 New Profile Hole

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



→ Where it is

Part Environment >> Shape >> Hole

3.2.5 Pattern Improvement

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

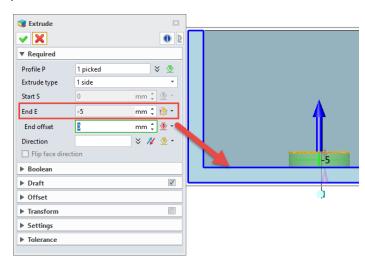


Pattern Feature Required Required Base Direction Number Spacing Symmetry	1 picked 1,0,0 4 50	23 F • 2 • 2 • 2 • 2 • 2 • • • • • • • • • •		
Symmetry Second direction Direction D Number N Spacing S Symmetry Variable spacing	0,0,1 5 28		50.00	
🔮 Spacing Table		2 2		
Spacing	Spacing S			
1 20	20		20.00	
2 30	50			
3 50	30			
4 Reset	20 OK Cance	el Apply	≈ 20.00 ≈ ≈ 30.00 - ∞ ≈	50.00

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

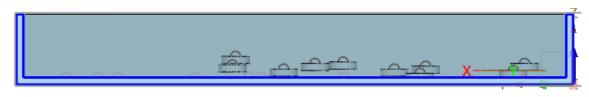
3.2.6 Extrude Point to Point Pattern Improvement

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

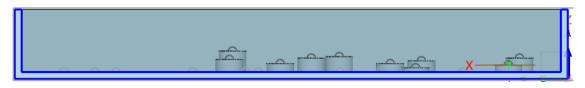








Before Improvement



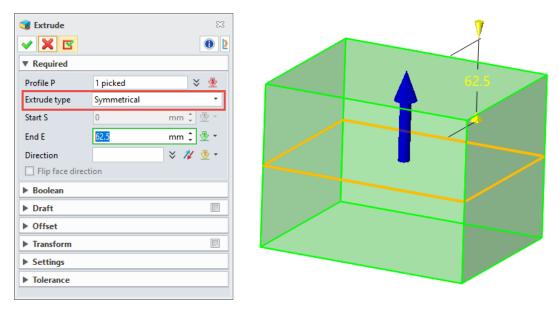
After Improvement

→ Where it is

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

3.2.7 Extrude Improvement

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



→ Where it is

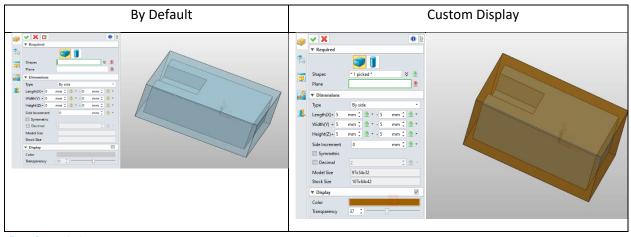
Part Environment >> Shape >> Basic Shape >> Extrude



3.2.8 Support Display Stock Color and Transparency

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

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



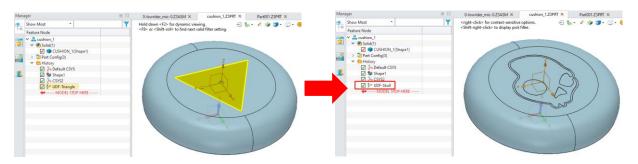
→ Where it is

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

3.2.9 UDF Improvement

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

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







Context Menu >> UDF Replace

3.2.10 📩 🛨 Loft Improvement

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

Required Loft type Profiles	
Profile 1 picked 1 🖉 Start point 🔮 Profiles 2 picked	
 ▶ Boolean ▼ Boundary Constraints Both ends Start End Continuity Flow ▼ Direction Perpendicular ▼ Weight 0.000 ↓ 	
Scale 1 Connection Lines Settings Auto Reduce Tolerance	

→ Where it is

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



3.2.11 Extrude/Revolve Workflow Improvement

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

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

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

Profile P Attribute Filter Result 🔶 🗕 🖽 • 🔿 🖬 📶 - 🕰 🗸 🗶 🖸 (▼ Required Profile P × - 🕀 - 🔿 🖬 Face - 😻 ÷ 🗸 🗙 🖸 (▼ Required × Profile P 🕂 🗕 🗄 • 🔿 🖬 📶 - 🕰 🗸 🗙 🖪 0 Required × Profile P 🗕 🗄 🔹 🚫 📊 Datum Plane 🔹 🚳 🗸 🗙 🖸 Required ⇒ Profile P

Here take the extrude feature as examples.

→ Where it is

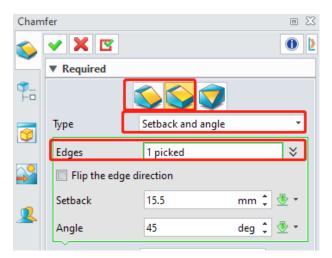
Part Environment >> Shape >> Basic Shape >> Extrude

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



3.2.12 New Memory Input Object in Chamfer

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

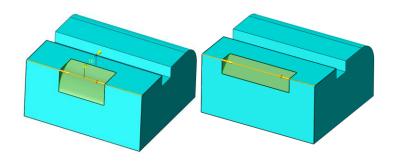


→ Where it is

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

3.2.13 New Stop Short of Corner in Fillet/Chamfer

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



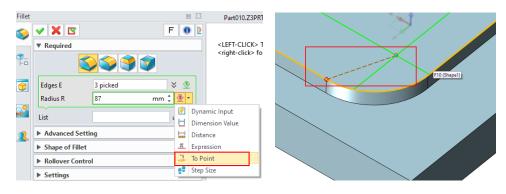
→ Where it is

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



3.2.14 New Dotted Line in To Point

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

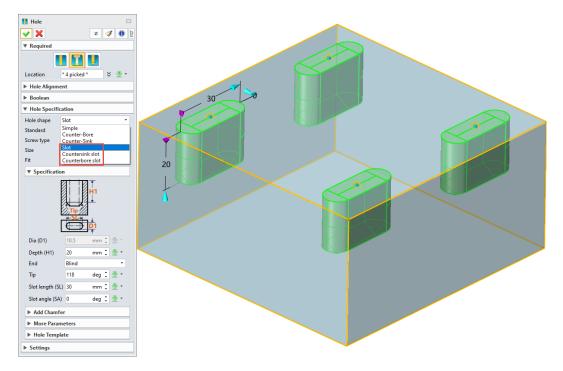


→ Where it is

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

3.2.15 New Slot Hole

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



→ Where it is



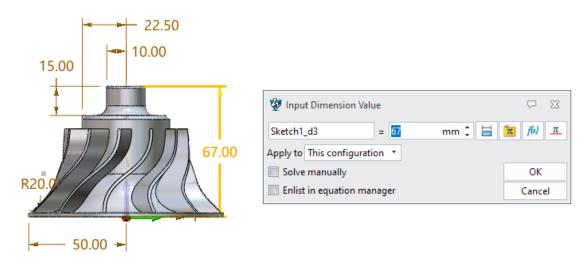


Part Environment >> Shape >> Engineering Feature >> Hole

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

3.2.16 Show Dimensions Improvement

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



→ Where it is

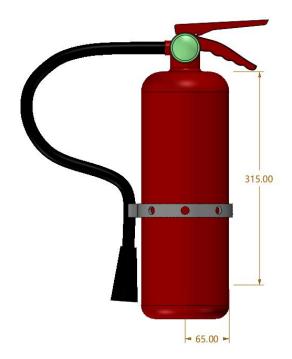
Part Environment >> Feature/Sketch Context Menu >> Show Dimensions

3.2.17 Dimensions On/Off Improvement

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

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



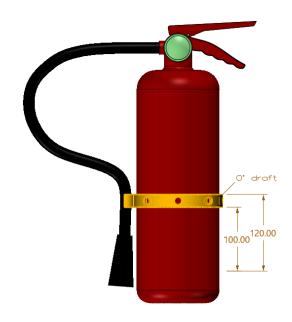


2) When user does not choose any feature and enable the Dimensions On/Off, the default show/hide all features dimensions.



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





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

3.2.18 Datum Axis Improvement

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

/ Datum Axis		23	A	
🗸 🗶 🖪		0		
▼ Required				
😢 🏚 🖌	<u>∡ Ľ Ľ / 1⁄</u>	2		
Face	F3			
Length	700 mm	‡ 🕸 🔹		
 Orientation 				
Flip direction				
▼ Datum Attribu	ıtes			
Custom attrik	outes			
Color				
Style		-		
Width		-		
Datum format	Arrow	•		

→ Where it is

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



3.2.19 **★**New External Geom Copy

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

😻 External Geom Copy 🛛 🔀	😻 External Geom Copy 🛛 🖄
 ▼ Required File/Part Part004.Z3PRT Preview Placement Default CSVS Geom to copy Copy all valid from pick file ▼ Association Settings ✓ Associative copy ④ Auto update Manual update Record state Ref feature Ref feature Ref part 	▼ Required Image: Second conductions Type Picked entities Geom to copy Image: Second conductive copy Image: Associative copy
117.961 mm	
117.961 mm	

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



- Support the disconnection and association of reference object as well as the manual and automatic update.
- 4) Support the widget picking the reference object.
- 5) Support referring to the feature with time stamp.
- → Where it is

Part Environment >> Data Exchange >> External Geom Copy

3.2.20 New Publish Set

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

😻 External Ge	com Copy 🛛	
🗸 🗙	0 2	
▼ Required		
File/Part Part009.Z3P	RT - Can	
Placement	Default datum 🔹	
Туре	Publish set	
Geom to copy	PublishSet1 💆	
	Settings	
Associative	сору	
	te 🔘 Manual update	
Record stat	e	
Ref feature		
Ref part		

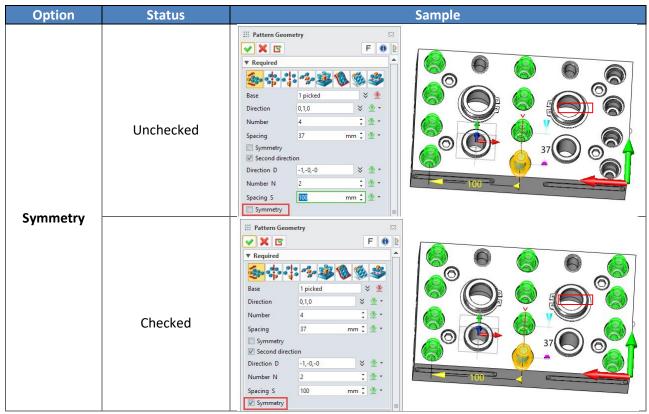
→ Where it is

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

3.2.21 Pattern Improvement

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





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

Pattern Geo	metry	83		S	
🗸 🗙 🖻		F 🕕 🖢		0	
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Base	4 picked	× 👲	1		
To points	2 picked	🛛 👲 +			
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Base point		> 👲 ▪			
▼ Instances to	Toggle				
Toggle		× 👲		2 - 1	
▼ Orientation				2 6 🥨	
Alignment		2.			
	e1.0	1.0			3
On face					
▼ Boolean					
	🧠 🤳 🍓	Ş			
Boolean shape		* 👲			
▼ Settings					
Associative			S		

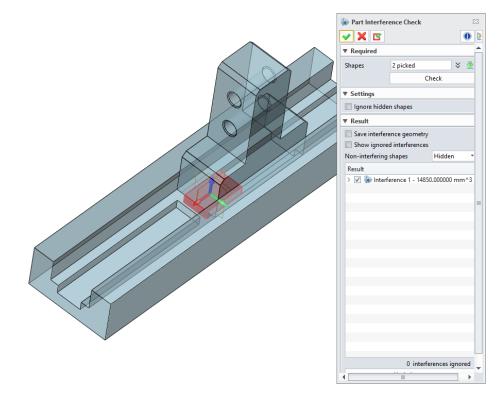
→ Where it is

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



3.2.22 New Interference Check

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



→ Where it is

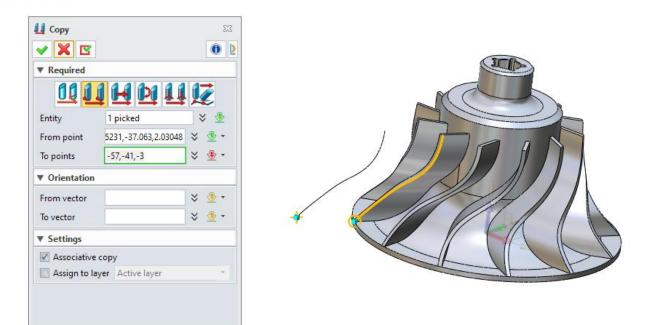
Part Environment >> Inquire >> Inspect Model >> Part Interference Check

3.2.23 New Copy Edge

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

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





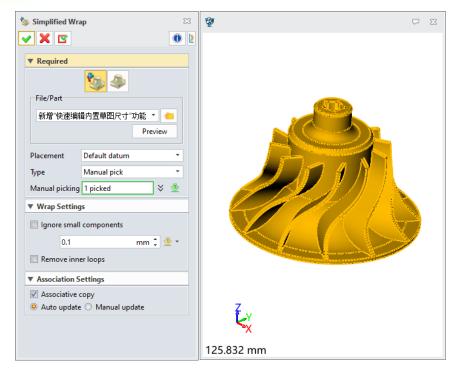
Part Environment >> Shape >> Basic Editing >> Copy

3.2.24 New Simplified Wrap

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

• Support extracting wrap shapes from the small windows.

ZŴBD



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

→ Where it is

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

3.3 Free Form Design

3.3.1 Hend Face Improvement

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



	23	
🗸 🗙 🖸	• 2	
▼ Required		
Start edge	E7 🛷 💆	
End edge	E3 🛷 💆	
▼ Start Constra	aints	
Continuity	Flow *	
Direction	Perpendicular 🔹	
Weight	0.000 ‡	
Side	Toggle Face	
▼ End Constrai	nts	
Continuity	Flow 🔹	
Direction	Perpendicular 🔹	
Weight	0.000 ‡	
Side	Toggle Face	
▼ Sew		
Sew shapes	*	

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

3.3.2Extend Shape Improvement

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

🕼 Extend Shape	×
▼ Required Edges 5 picked > Distance 24 mm 2	
Settings Surface extension Linear Edge extension Orthogonal Method None Create new faces	
Loft extend with new faces	The edge of new surface is orthogonal to the extended edge.

→ Where it is

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



3.3.3 New Disjoint Entities as New U/V

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

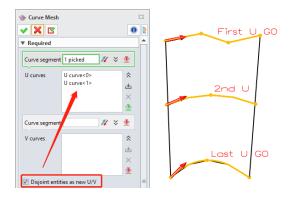


Figure A Check "Disjoint entities as new U/V"

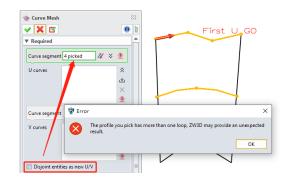


Figure B Uncheck "Disjoint entities as new U/V"

→ Where it is

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

3.4 Wireframe Design

3.4.1 **★**Spiral Helix Improvement

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

Constant radius Line	ear radius	3 degree radius	Custom radius
----------------------	------------	-----------------	---------------



	MM	MMM	
	\sim		
			-
Linear pitch	Custom pitch	Variable radius and	Law curves
Linear pitch	Custom pitch	variable radius and	Law curves
•	•		
		nitch	
		pitch	

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

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

3.4.2 New Keep Original Curve in Curve to Arcs/Lines

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

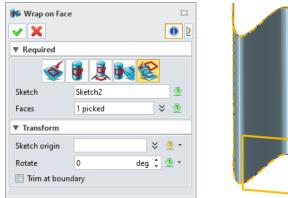


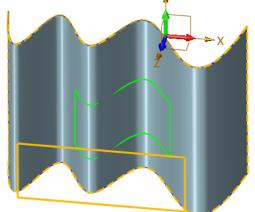
Convert to A	arcs/Lines	× ()
▼ Required		
Curves		¥ 👲
▼ Settings		
Tolerance	0.5	mm 🗘 垫 👻
🔲 Keep origina	l curve	

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

3.4.3 New Wrap Curve on Developable Face

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





→ Where it is

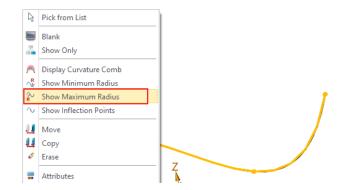
Part Environment >> Wireframe >> Curve>> Wrap on Face

3.4.4Show Maximum Radius Improvement

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







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

3.5 History Feature

3.5.1 Modify Tolerance Improvement

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

Modify Tol	erance	23
🗸 🗙 🖪		0
▼ Required		
Feature list		≈
New Tol	0.01	mm 🗘 垫 👻

→ Where it is

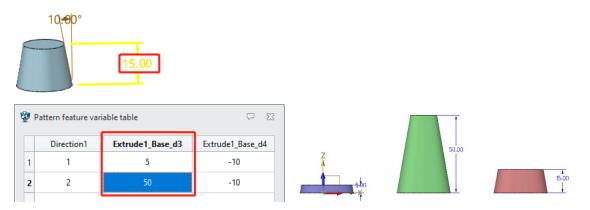
Part Environment >> Edit >> History Operation >> Modify Tolerance

3.5.2 **★** Pattern Feature

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

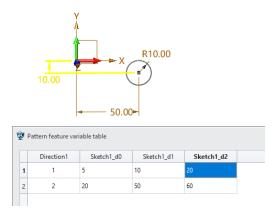
▼ Variable Pattern				
Туре	Instance parameter table	-		
Increment	None Parameter increment list Parameter increment table			
Instances to Tellnstance parameter table				

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



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

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



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

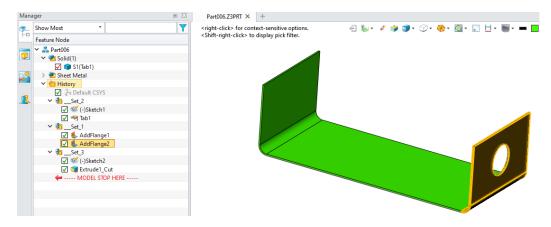




Part Environment >> Shape/Free Form/Wireframe/Weldments >> Basic Editing >> Pattern Feature

3.5.3 Location History Node Improvement

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



3.5.4★ UDF Improvement

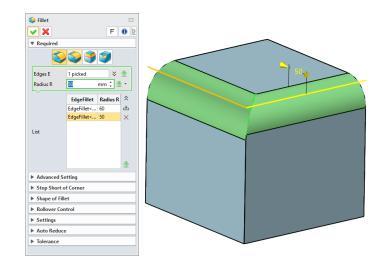
3.5.4.1 UDF Supports List

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

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







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

3.5.4.2 UDF Supports More Commands

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

3.5.4.3 UDF Supports Custom Color

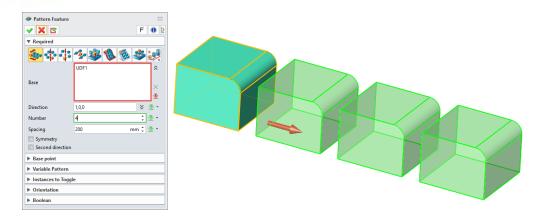
We added "Custom color" in the UDF panel, which is used to generate the color of UDF feature.

🖗 User Defined	Feature Wizard				\Box	23
Definitions	Features in Part		Features	in User Define	ed Feature	
Features	Custom color		🔲 Add (Children Featu	ires	
Expressions	✓ ▲ UDF → Default CSYS	_	📇 U	DF		
References	Block1_Base		-			
Summary						
						
		Back	Next	Finish	Cancel	

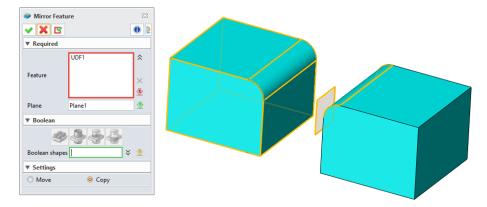
3.5.4.4 "UDF" Pattern Feature/Mirror Feature Support Selecting UDF

The command "Pattern Feature' supports selecting UDF as input objects.





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



3.5.4.5 UDF Placing Method Improvement

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

UDF2				
✓ X				
▼ Reference				
Base < Punch1	●			
Location < Punch1	🗧 🕹 🕇			
Frame < Punch1	<u>.</u>			
1st entity				
2nd entity	*			
$($	Q 0 // L Z			
Offset	0 🗘 🔹 -			
Angle	0 deg 🗘 🔹 🔹			
	Flip direction			
List < Punch1	*			
Open O < Punch1				
▼ Settings				
Repeat				



🐓 UDF6 🗸 🗙 0 Reference 1 Face < CSYS1 F2 🗧 🛬 ד Point < CSYS1 66.3024,-22.0061,0.5 े 垫 🝷 Direction < CSYS1 1,0,0 \approx Open O < Punch1 🛛 🕹 👲 Settings Repeat

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

3.5.4.6 New User Editable Expressions

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

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

🐲 User Defined I	Feature Wizard		₽ X
Definitions	Available Option References	User Edit References	
Features	 ✓ ♣ Part009 ✓ ॐ Sketch1 	Plane < Sketch1	
Expressions References	Origin Up V W Sketch2	Plane < Sketch2	
🔲 Summary	Origin Up		
		Share input Unshare input New Prompt	:
		Back Next Finish	Cancel

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





UDF0	23
✓ X	0
▼ Reference	
Plane < Sketch1	
▼ Settings	
Repeat	

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

₽ UDF1 ▼ X	× (1)
▼ Reference	
Plane < Sketch1	
▼ Settings	
🔲 Repeat	

→ Where it is

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

3.6 Sheet Metal Design

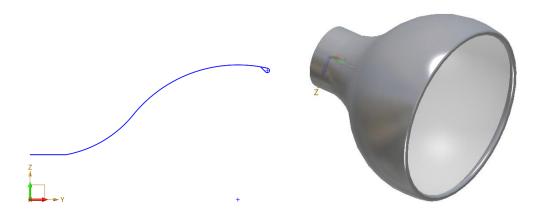
3.6.1 ★New Revolve Flange

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

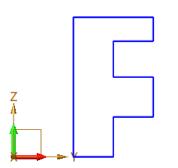
Revolve Flange command supports picking an open profile as well as a closed profile. The picked profile object will create a sheet metal solid by revolving an axis. User can set the rotation angle, thickness, bend radius, and K-factor of the sheet metal. An opposite direction is added to the thickness.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

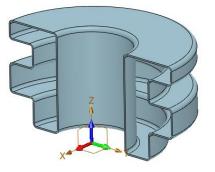
ZW3D

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Open Profile





Closed Profile





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

3.6.2 **★** Lofted Flange Improvement

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

Improvement	Main improvement	Case
Non-parallel profile lofting	Support the sheet metal lofting design of the non-parallel profile	Image: second
Complicated curve lofting	Support the complex curve sheet metal lofting such as splines	Image: state
Closed curve lofting	Support the sheet metal lofting of the closed curves	I laftad Hange V Registrad V Regestrad

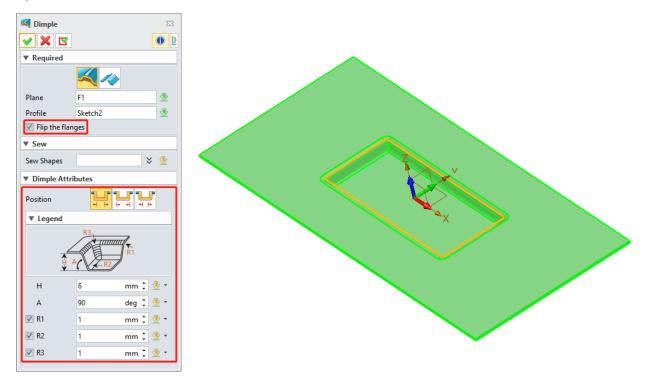
→ Where it is

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



3.6.3 Dimple Improvement

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



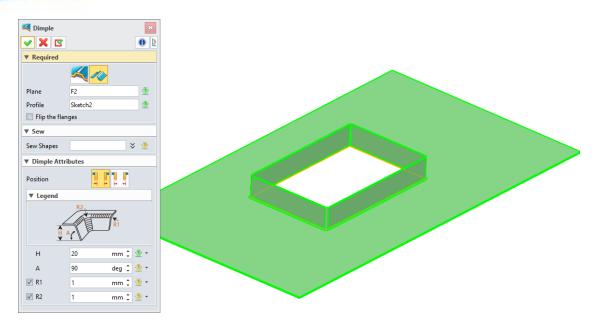
→ Where it is

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

3.6.4 Flared Hole Improvement

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

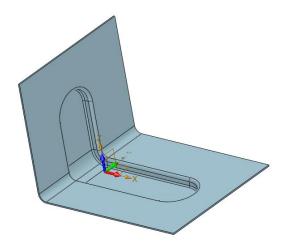




Part Environment >> Sheet Metal >> Form >> Dimple

3.6.5 Fold by Line Improvement

Fold by Line command supports folding sheet metal with dimple.



→ Where it is

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



3.6.6 Flatten Improvement

Flatten command supports ignoring tangent edges from flat pattern.



→ Where it is

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

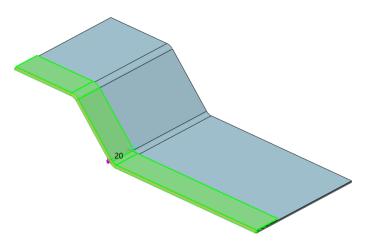
3.6.7★New Double Bend Flange

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

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



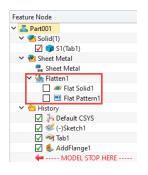




Part Environment >> Sheet Meal >> Flange >> Full Flange/Partial Flange/Hem Flange

3.6.8 Automatically Add Flatten Attribute

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



→ Where it is

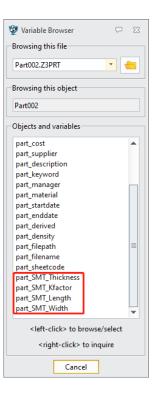
Part Environment >> Manager >> Sheet Metal >> Flatten

3.6.9 New Sheet Metal Variables

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

ZW3D





→ Where it is

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

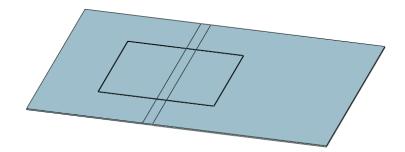
3.6.10 New Curves on Sheet Metal in Unfold/Fold

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

Unfold 🛛	
✓ X I	
▼ Required	
Shape 🛛 💆	
▼ Settings	
Stationary 👲	
Bend faces 🛛 🕹 👲	
Collect all bends	
Curves on sheet metal 8 picked 🛛 🕹 👲	



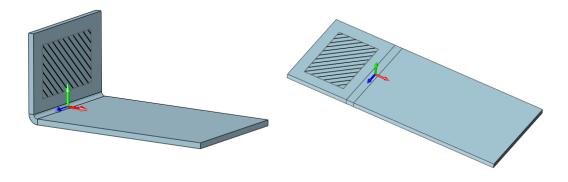




Part Environment >> Sheet Metal >> Bend >> Unfold/Fold

3.6.11 Cosmetic Sketch Fold as Sheet Metal

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



→ Where it is

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

3.6.12 Sheet Metal Flange Improvement

3.6.12.1 Flange with Profile Supports Pattern Feature/Mirror Feature

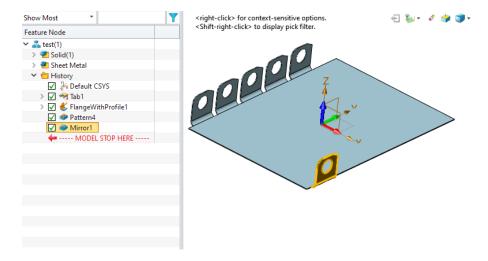
ZW3D 2023 supports pattern feature and mirror feature of "Flange with Profile". So far, there are three flange commands: **Full Flange**, **Partial Flange** and **Flange with Profile** that support pattern feature/mirror feature.

Flange with Profile supports pattern feature during sheet metal design.



Show Most *	middle-click> to		- A] 🗞 - 🥒 🎲 🕻	يا 🗊 😒 🗞 😪 🖓 -
Feature Node	< Shift-right-click?	 to display pick filter. 		~	
🖌 🚣 test(1)		_			
> 🔁 Solid(1)					
> 🕘 Sheet Metal					
✓			_		
✓ ½ Default CSYS ✓ ≪ Tab1			<u>_</u>		
> V 🐇 FlangeWithProfile1				1	
MODEL STOP HERE			The second se		
				\sim	
	<			<u> </u>	
	Pattern Feat	ure	23		
	✓ X		F 🕕	2	
	▼ Required				
	<u> -</u>	\$ * 🐝 🔇	s 🍪 🤹 🧈		
	Base	1 picked	¥ 👲		
	Direction	0,1,0	🗧 🛬 🗝		
	Number	5	‡ 🕭 •		
	Spacing	50	mm 🗘 速 *		
	Symmetry				
	Second dire				

Flange with Profile supports mirror feature during sheet metal design.



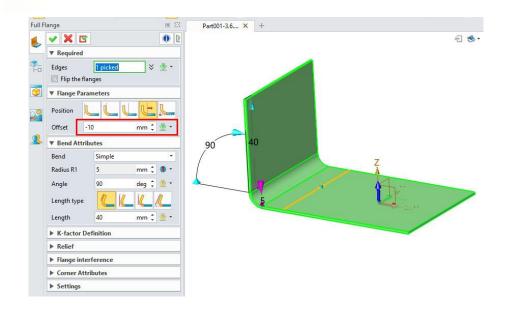
→ Where it is

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

3.6.12.2 Offset Supports Negative Value

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





3.6.12.3 New Bend Deduction

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

💺 Full Flange 🛛				
🗸 🗙 🖪	0			
▼ Required				
Edges Edges Flip the flanges	× 💁 -			
▼ Flange Parameters				
Position				
▼ Bend Attributes				
Bend	Simple *			
Radius R1	5 mm 🗘 💆 🔹			
Angle	90 deg 靠 垫 🕶			
Length type	<u>(</u> i(_ i(_ /(_			
Length	40 mm 🗘 💆 👻			
▼ K-factor Definit	ion			
Туре	Custom -			
K-Factor	Custom From material			
▶ Relief	Bend table Bend allowance table			
► Flange interfere				
Corner Attribut	Bend deduction			
Settings				

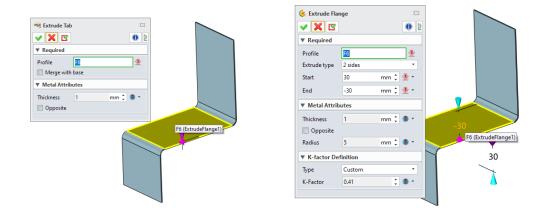
→ Where it is

Part Environment >> Sheet Metal >> Flange



3.6.13 Extrude Tab and Extrude Flange Improvement

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



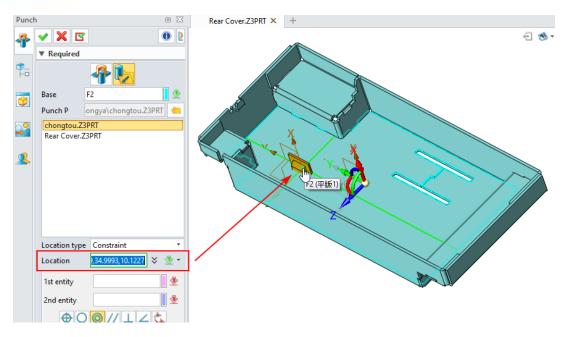
→ Where it is

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

3.6.14 Punch Improvement

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





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

3.6.15 K-factor Supports Greater Range

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





👰 Sheet Metal Attribu	ites	₩ 23	🐲 Sheet Met	al Attributes	₽ X
Standard Corner	Attributes		Standard	Corner Attributes	
► Global Value			► Global Va	lue	
Flange Paramete	rs		Flange Pa	arameters	
▼ K-factor			▼ K-factor		
Туре	Custom	•	Туре	Custom	•
Default value	-0.5	÷	Default valu	e 1.5	÷
▶ Relief			► Relief		
Import			Import		
Export			Export		
Add sheet metal fla	t automatically		Add sheet	metal flat automatically	
Accept	Reset	Cancel	Accept	t Reset	Cancel

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

3.7 Weldment Design

3.7.1 New Fillet Weldment

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

ZWJD

🖆 Fillet Weld	
✓ X I	
▼ Required	
Faces 1 1 picked 🛛 🕹	
Faces 2 1 picked 🛛 🕹	
Two sided	
▼ Cross Section	
Geometry type Surface 🔻	
Contour None *	
Dimension type Weld leg 🔹	
Weld leg(K) $mm \ddagger 5 - 1$	
Offset and Toggle	
► Intermittent	
Supplementary Symbol	
► PMI	

→ Where it is

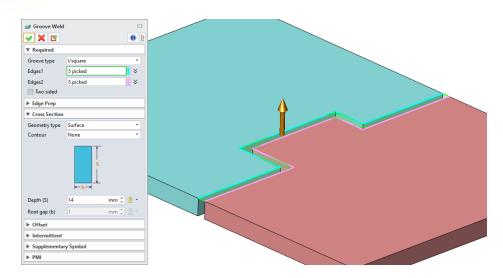
Weldments >> Fillet Weld

3.7.2New Groove Weldment

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

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





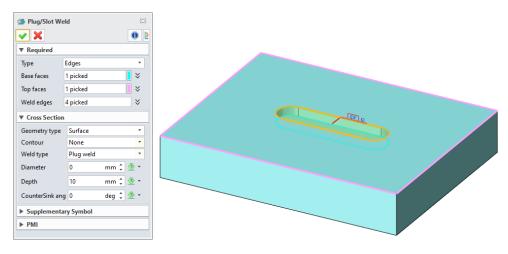


Weldments >> Groove Weld

3.7.3 New Plug/Slot Weld

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

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



→ Where it is

Weldments >> Plug/Slot Weld





3.7.4New Spot Weld

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

🍜 Spot Weld	4
	1
▼ Required	
Points 4 picked 😵 👲 🔻	
▼ Cross Section	•
Geometry type Surface 🔹	
Contour None 🔻	
Dimension type Diameter 🔹	
Diameter 1 mm 🗘 🏂 🔻	
▼ Supplementary Symbol	o
Finish None -	
Field weld	9
Tail	
Tail type 🚽 🔻	
Text	
▶ PMI	

→ Where it is

Weldments >> Spot Weld

3.7.5 Weldment Improvement

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

3.7.5.1 Support Solid Weldment

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

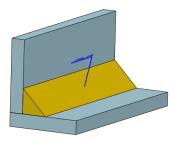
In Cross Section option, user can switch Solid option in Geometry type and create the solid weldment.

Cross Section	
Geometry type	Solid 👻
Contour	Surface Light
Dimension type	Solid

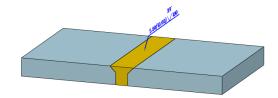
Solid type in Fillet Weldment:



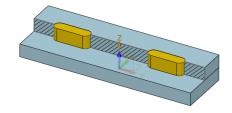




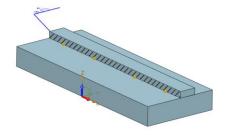
Solid type in Groove Weldment:



Solid type in Plug/Slot Weld



Solid type in Spot Weld:



→ Where it is

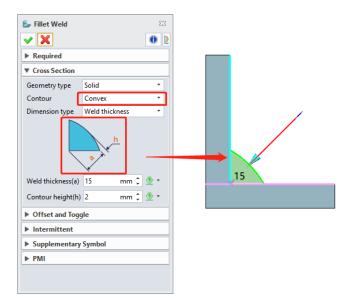
Part Environment >> Weldments >> Frame >> Fillet Weld/Groove Weld/Plug/Slot Weld/Spot Weld

3.7.5.2 Convex Concave Improvement

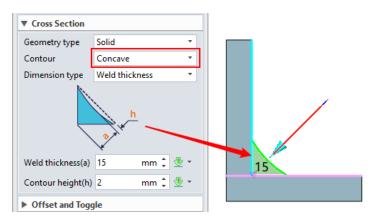
ZW3D 2023 supports creating the convex or concave type of weldment to generate solid with the actual graphic feature.



Convex in Fillet Weld:



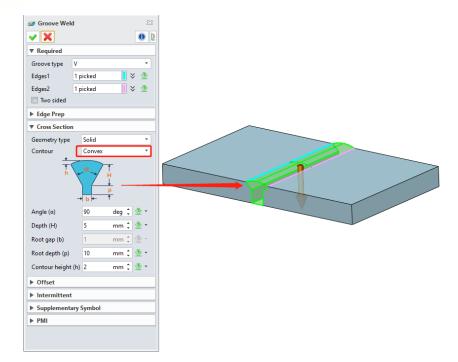
Concave in Fillet Weld:



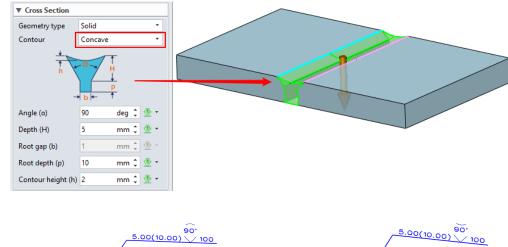
Convex of V type in Groove Weld:

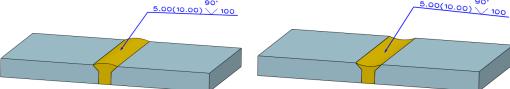
ZWBD





Concave of V type in Groove Weld:





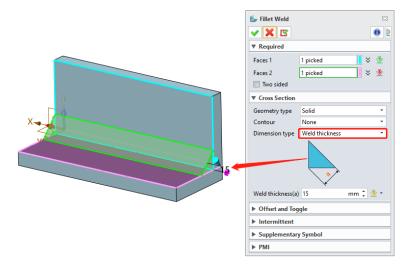
→ Where it is

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



3.7.5.3 Fillet Weld Improvement

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



→ Where it is

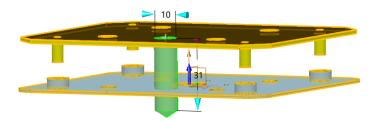
Part Environment >> Weldments >> Frame >> Fillet Weld

3.8 Assembly Design

3.8.1 Assembly Feature Improvement

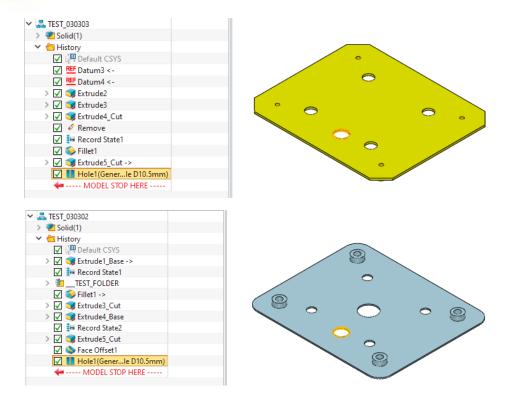
When creating assembly features, if "Propagate feature to components" checked, the assembly features can be transmitted synchronously to the corresponding parts/sub-parts of the assembly components.

Case: add assembluy "Hole" feature to the assembly.



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





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

3.8.2 **★**New Smart Fastener

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

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



Smart Fastener	
Required	
Location faces	
Multi insert	
Result	
Count Holes Fasteners	
8 General_Simple 3.2 Hex head bolts ISO4014-M3x20	
Set different lengths as one item	
Constrain component	
Parameters	
Fastener Hex head bolts ISO4014.Z3	
🖉 Image 🖉 Refresh	
Auto size to hole diameter	
🖉 Auto update length	
Name Value	
Nominal Diameter d(m 3	
Nominal Length I(mm) 20 🔹	
▼ Top Stack	
🖉 Add top stack	
Plain normal washers A ISO7089.Z3	
Bottom Stack	

Assembly Environment >> Component >> Smart Fastener

3.8.3 Assembly Constraint Improvement

3.8.3.1 Assembly Constraint Order Adjustment

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

Mana	iger (E Σ3	Mana	iger	
1	Show All	Y	7 -	Show All	
1-0	Assembly Node		+o	Assembly Node	1
-	✓ ▲ Assembly 001		-	✓ X Assembly 001	
	✓ (=)12121			✓ (→)12121	
	🗹 🇊 (–)Part002			🗹 🌍 (-)Part002	
۲	🗹 🧊 (–)Part003		1	🗹 🌍 (-)Part003	
	Before adjusting the ord	or		After adjusting the o	rdor
-	✓			✓	luei
2	🖌 💠 Coincident 1 (12121, 平面1)			🖌 💠 Coincident 1 (12121, 平面1)	
	Coincident 2 (Part003, 12121)			Coincident 2 (Part003, 12121)	
	Coincident 3 (Part002, 12121)			Coincident 3 (Part002, 12121)	
	Tangent 2 (Part002, 12121)			Tangent 2 (Part002, 12121)	

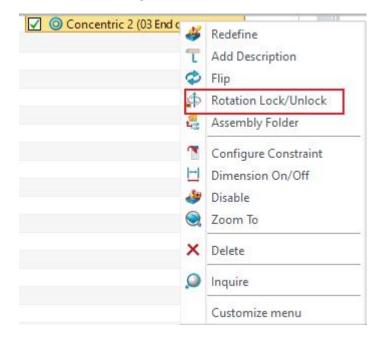
[→] Where it is

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



3.8.3.2 Concentric Constraint Improvement

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



→ Where it is

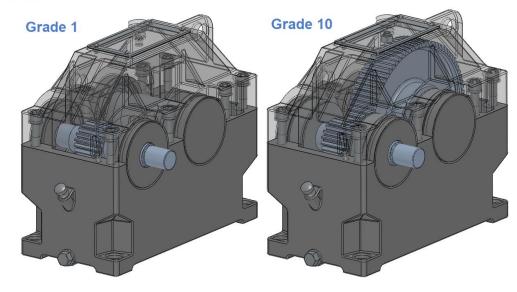
Assembly Environment >> Assembly Manager >> Constraint Context Menu >> Rotation Lock/Unlock

3.8.4 Simplified Wrap Improvement

3.8.4.1 New Sensitivity Control

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







Part Environment >> Data Exchange >> Simplified Wrap

3.8.5 **★**New Envelop

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

The main function is applied to the following cases:

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

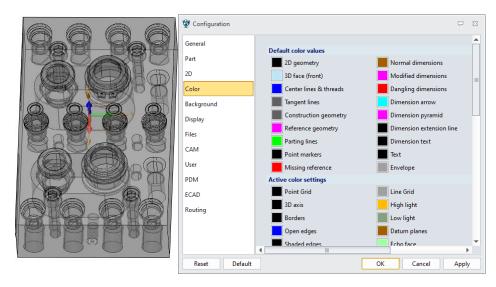
🕹 Insert		23	🕹 New to Inse	ert		Σ
✓ X		0	✓ X			0
Copy entire a	issembly part		▼ Required			
Copy associa	ted 2D layout		Name	Part003	.Z3PRT/Standard	•
Part Name	Part002		Template	[Default]		•
Regen	None	-	▼ Placement			
Auto delete insta	anced part		Туре	Default fram	me	-
▼ Settings			E Fix compon	ent		
Show dynamic p	preview		▼ Settings			
Pocket	Pocket color from part		🗹 Auto activat	ted		
	V Envelope		🔲 Virtual	🗹 Envelope	e	
Insert to layer	Active layer	• L	Insert to layer	Active layer	r	Ŧ



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

Assembly Node		Assembly Node		
✓ ¾ Assembly001	🍯 <u> 🔄 🛃 🎸 </u> 👘 1	 Assembly001 		🗧 🔮 🗳 🖉 📬
	Change Component Rename Part Display		〇日 間	Regen Change Component Rename Part Display
	 Configure Component Group as Sub-assembly Assembly Folder Suppress Blank Show Only Zoom To Pack Merge Fix Unblank External Datum Make Virtual 		3	Configure Component Group as Sub-assembly Assembly Folder Suppress Blank Show Only Zoom To Pack Merge Fix Unblank External Datum Make Virtual
	Make Envelope		1	Make Envelope

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



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

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



Standard 🖾 🗸	0	•		•	
▼ Required					S
File/Part					Y
3.7.1.Z3ASM 👻 📹					
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View TOP - Location × 🔮 -	Ø	Ý			Ő
▼ Settings		e e e a			
Optional Advanced				10 10 10 10 10 10 10 10 10 10 10 10 10 1	
Style <from standard=""> T</from>					
				14 - 14 - 14 - 14 - 14 - 14 - 14 - 14 -	
General Label Lines Model			• • •	• • •	• • •
Object select mode Exclude		1. 1. <i>1. 1</i> . 1	N N 17		
X2431_K85583150					
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Location 🛛 🕹 🝷	a 4 a a			a a a	
Location × ⊕ - ▼ Settings	176th	<u>iita</u>		<u>e</u> la	
▼ Settings					
▼ Settings					
Settings Optional Advanced					
Settings Optional Advanced Style <from standard=""> T General Label Lines Model</from>					
Settings Optional Advanced Style <from standard=""> General Label Lines Model</from>					
Settings Optional Advanced Style <from standard=""> General Label Lines Model Object select mode Exclude</from>				*	

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

→ Where it is

Assembly Environment >> Component >> Insert



3.8.6 Assembly Mirror Improvement

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

3.8.6.1 Self-symmetric Optimization

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

Mirror 🖾							
🗸 🗶 🔽				0			
▼ Required							
Components	I			¥ 👲			
Plane				<u>.</u>			
▼ Settings							
		36 0					
Erase source	geometry						
Instanced as	components						
🔲 As a whole t	o mirror						
Instances' at	tributes as par	ent					
с. I.с	XZ	YZ	XY	С			
Self-symmetric	XcZc	YcZc	XcYc	None			
Datum plane							
▼ Center of							
Bounding be	ox O M	ass	🔘 Frame				

3.8.6.2 Sub-assembly Mirror Optimization

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

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

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



🔶 Mirror				23	≁ <mark>⊳</mark> Mir	ror			23
🗸 🗙 🗹				0	· · · · · · · · · · · · · · · · · · ·	Κ 🖪			0
▼ Required					▼ Rec	quired			
Components	1 pick	ed		¥ 👲	Comp	onents	1 picked		😆 🔮
Plane	XY			<u>*</u>	Plane		XY		
▼ Settings					▼ Set	tings			
Erase source g Instanced as co As a whole to Instances' attri	omponents mirror	• • •			✓ Ke	ase source geom ep geometry ass ep position asso mbly nodes Ca	etry sociative ciative	& duplica	te
Assembly nodes	Category		-symmetric			EXHAUST Du	/	ii-symmetric	
✓ 10 EXHAUST_M		XZ	-,			10-02 EXHACo			
10-02 EXHAI		XZ				10-01 EXHADu			
10-01 EXHAI		XZ				11-02 PS_P De	lete		
11-02 PS_PU	Сору	ХZ			Categ	jory 💿 Co	py 🔘 Duj	plicate 🔘 D	elete
Self-symmetric	XZ	YZ	XY	С	Self-s	ymmetric	Z YZ	XY	С
seir-symmetric	XcZc	YcZc	XcYc	None	Jen 3	X	zZc YcZc	XcYc	None
Datum plane				<u>*</u>	Datur	n plane			
▼ Center of					▼ Cen	nter of			
Bounding box	🔘 Mas	s	O Frame		● Bo	ounding box	Mass	🔘 Frame	

3.8.6.3 Full Mirror Optimization

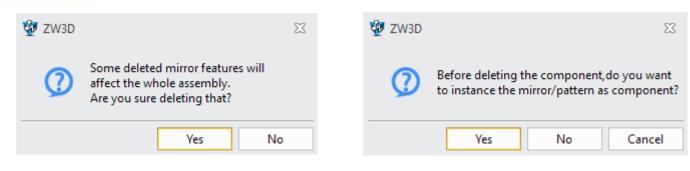
When completely mirroring components, user can check the option "Erase source geometry" to keep the source component objects.

Mirror 🖾							
▼ Required							
Components	1 picked	¥ 👲					
Plane	XY						
▼ Settings							
Erase source geometry Instanced as components							
As a whole to mirror							
Instances' attributes	as parent						

3.8.6.4 Deleting Source File Warning

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





Deleting mirror feature warning

Deleting mirror source file warning

→ Where it is

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

3.8.7 Support Dragging Overall Component

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

Man	ager	IE 23	Man	ager	E 23
-	Show All	Y	-	Show All	Y
FO	Assembly Node		Fo	Assembly Node	
4	✓ X Assembly001 ✓ X Assembly002		-	✓ X Assembly001 ✓ V Assembly002	
3			7	 ✓ (-)12121 ✓ (-)Part003 x 2 ✓ (-)12121 ✓ (-)12121 	
-	🔽 🇊 (-)Part003 x 2			🔽 🇊 (-)Part002 x 2	

→ Where it is

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

3.8.8 Copy Component Improvement

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



Mani	ager			Man	ager
-	Show All				Show All
100	Assembly Node	Man	ager	Fo	Assembly Node
	ZAssembly001 Original assembly (-)12121 Original assembly	10	Show All		~ Assembly001
•		*	Assembly Node Assembly001 Copy assembly G (-)Part003 (-)Part003 (-)Part003 (-)Part002	9	Z
2	✓ Φ Coincident 3 (Part002, 12121) ✓ Tangent 2 (Part002, 12121) ✓ Φ Coincident 2 (Part002, 12121) ✓ Φ Coincident 1 (Part012, Part012) ✓ Φ Coincident 1 (Part002, Part002)	2	(-)12121 (-)Part002	2	

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

→ Where it is

```
Assembly Environment >> Editing >> Copy + Paste
```

```
Assembly Environment >> Ctrl C + Ctrl V
```

3.8.9 Interference Check Result Support Order

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



Interference	Check	23
🗸 🗶 🖾		0
▼ Required		
Components	4 picked	* 👲
	Check	
▼ Settings		
Scope	Only among the picked	•
Check amon	g shapes n shapes and components	
Save interfere		Ŧ
Result Interfe Interfe Interfe	erence 1 - 65449.853845 mm^3 erence 2 - 29594.564964 mm^3 erence 3 - 1176.848662 mm^3	Big Small
	0 interferences i	gnored
	Undo ignores	

Assembly Environment >> Inquire >> Interference Check

3.8.103D BOM Support Ignoring Virtual Components

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



3D BOM															Ģ
dented	-	2 C	12		× 😰	ID		•		8	Y b	Column	Row Settings		
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												Index	ID		
												Name	Name	V	
												Number	Number	V	
												Class	Class		
												Designer	Designer		
												Cost	Cost		
												Supplier	Supplier		
												Material	Material		
												Quantity	Quantity	V	[
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														ate	_
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													Add user attrib	ute	

Assembly Environment >> Inquire >> 3D BOM

3.8.11Assembly Attributes Manager Improvement

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

ų	Ass	embly Tree Attributes Manager				Π Σ			
[All A	ttributes		Displayed					
	Туре	User 🔹		Name	Туре	Width			
	AA	A		Assembly Node	Assembly	304	Man	iger	e X
							7 -	Show All *	T
							FU	Assembly Node	AAA
			-				-	✓ ▲ Assembly001	
			4					🔽 🧊 (–)Part001	12345
							_		
							9		
				Width 304 🛟		♦			
ſ	Def	ault		ОК	Cancel	Apply	2		
							_		

→ Where it is

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

3.8.12New Auto Activated in New to Insert

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



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

🕹 New to Insert			23						
✓ X) 🔰						
▼ Required									
Name	Part002	.Z3PRT/Standard	•						
Template	Template [Default]								
▼ Placement									
Туре	Default frame		•						
Fix component									
▼ Settings									
Auto activated	☑ Auto activated								
Virtual									
Insert to layer	Active layer		•						

→ Where it is

Assembly Environment >> Component >> New to Insert

3.8.13Auto Regen Mark Improvement

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



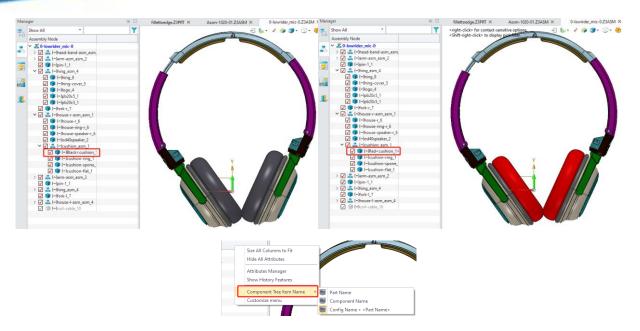
→ Where it is

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

3.8.14New Component Tree Item Name

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

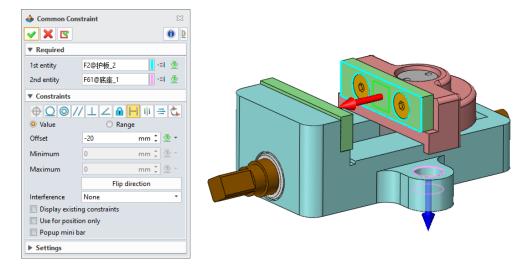




Assembly Environment >> Assembly Tree Manager >> Context Menu

3.8.15 Distance Constraint Improvement

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



→ Where it is

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



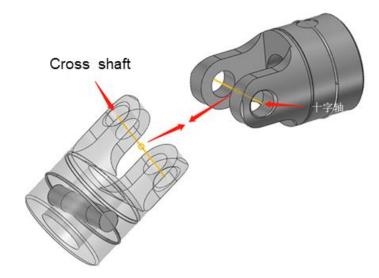
3.8.16 **★**New Universal Joint

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

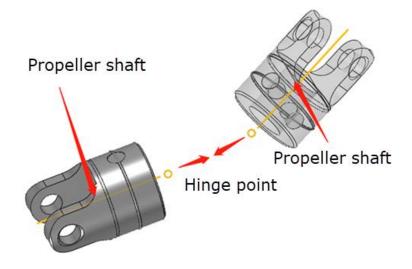
🖑 Mechanical Constraint	23	🗳 Mechanical Constraint 🛛 🖾
✓ X	0	
▼ Required		▼ Required
1st cross shaft	_ ₫	1st propeller shaft
1st hinge point 🛛 🕹	<u>⊪</u> -	🗹 1st hinge point 🛛 💝 👻 👻
2nd cross shaft		2nd propeller shaft
2nd hinge point 🛛 🕹	₫ •	🗹 2nd hinge point 🛛 🗧 😤 👻
▼ Constraints		▼ Constraints
৻৻৻৵৻৻	20	ॐ∿≓∰₫∽♀₽
• Variable velocity		Flip Constant valasity Constant valasity
Display existing constraints		 Variable velocity Constant velocity Display existing constraints
Interference None	-	Interference None *

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





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



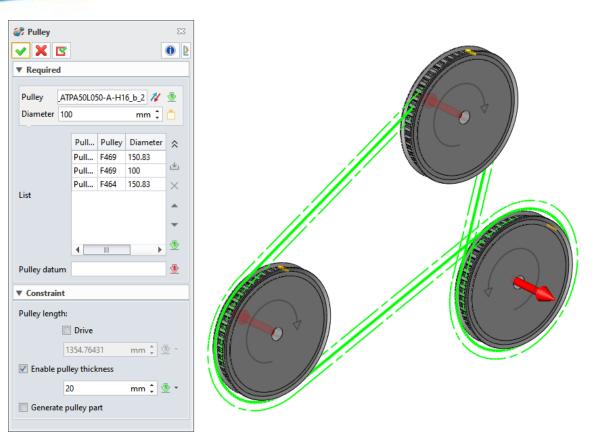
→ Where it is

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

3.8.17 **★**New Pulley Assembly Feature

Use this new command "Pulley" to design pulley components.

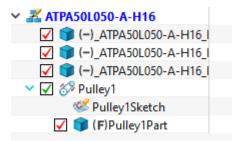




This command can be used to

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

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



Assembly Environment >> Assembly >> Constraint >> Pulley

3.8.18 Insert Improvement

3.8.18.1 New to Insert

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

🕹 New to Insert			23			
✓ X		0)]			
▼ Required						
Name	Part001	.Z3PRT/Standard	•			
Template	Template PartTemplate(MM)					
▼ Placement	PartTemplate(MI	M)				
Туре	Default frame	•	-			
Fix component	t					
▼ Settings						
Virtual						
Insert to layer	Active layer		•			

→ Where it is

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

3.8.18.2 Standard Part with Pocket Attribute

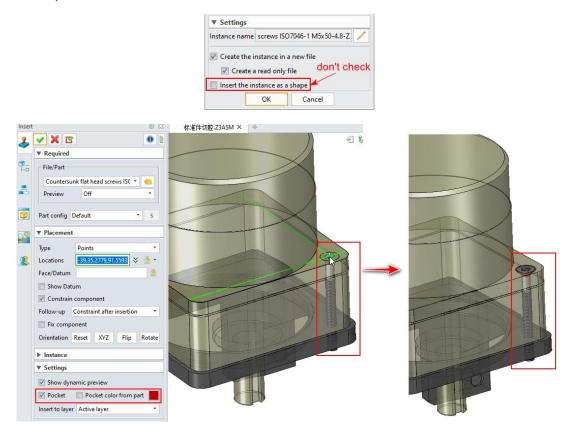
We added "Pocket" to the standard part "Insert" panel. When the standard part's pocket is defined pocket attribute, after inserting the standard part, the pocket feature will be inserted in the pocketed part



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

▼ Settings		
Show dynamic preview		
Pocket	Pocket color from part	
Insert to layer	Active layer	*

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



There exists several characteristics in standard part and pocket feature:

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

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

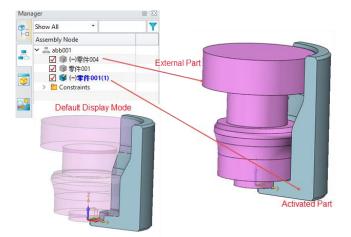
3)When the associated instance is deleted, the corresponding pocket feature should be deleted.



Assembly Environment >> Assembly >> Component >> Insert

3.8.19 New Shade Mode in Display OutofScope Parts

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



→ Where it is

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

3.8.20 **★**New Motor in Assembly Animation

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



Required Component Large gear shaft_1 Direction 0,1,-0 V V Type Uniform speed V Speed 60 V V End 0.05	🚖 Add Mot	tor	23				e
Component Large gear shaft 1 Direction 0,1,-0 Vipe Uniform speed Speed 60 1 0:00	Constant of		0 2	19			•
Component Large gear shaft_1	▼ Required	d 🔊					
Direction 0,1,-0 × • • // Type Uniform speed • Speed 60 • • • • • • • • • • • • • • • • • •	Componer				TE		
Type Uniform speed Speed 60 Start 0:00 Value (Construction)			14		1 The	CON P	
Speed 60 C Prom *	Type	Uniform speed 🔹			- FA		VASSA
Start 0:00	Speed	60 🌲 💆 🕇	rpm 🔻	6 Mall		0	
End 0:05 •	Start	0:00 -					<u>·</u>
	End	0:05 *			13-		
						Ĩ	

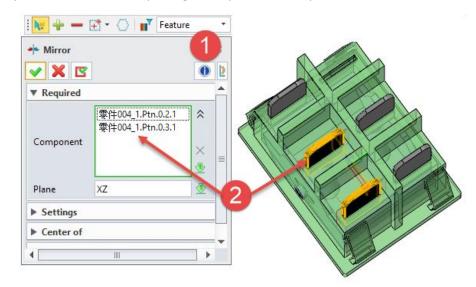
TTTT

→ Where it is

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

3.8.21 Assembly Pattern/Mirror Improvement

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



→ Where it is

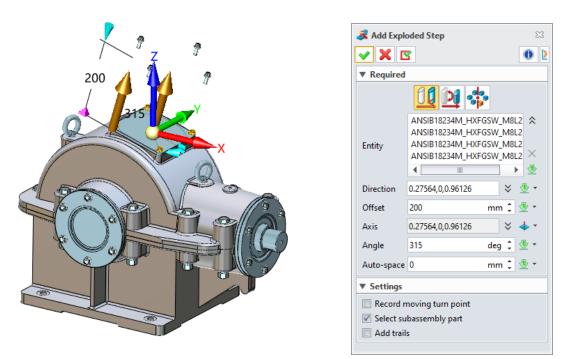
Assembly Environment >> Assembly >> Basic Editing >> Patten/Mirror



3.8.22 Exploded View Improvement

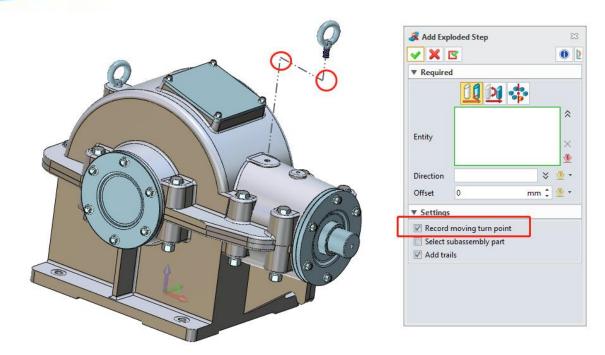
3.8.22.1 Add Exploded Step Improvement

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

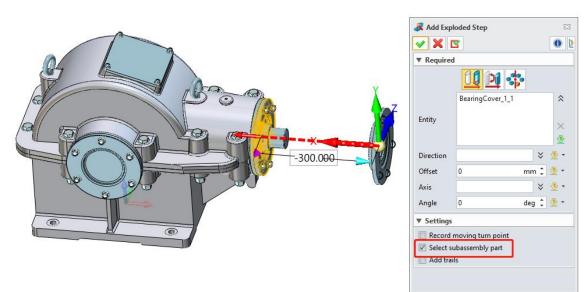


• Check "**Record moving turn point**" to record the current position, and then perform exploded steps in other directions. The final one will be a complex and continuously multiple exploded paths.

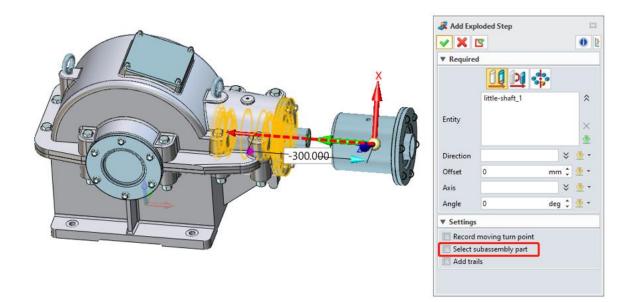




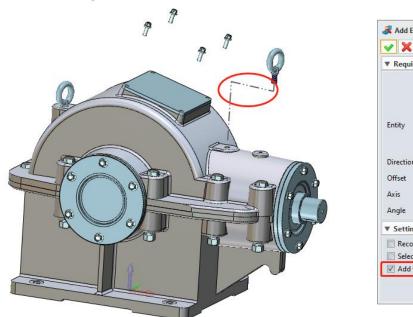
• Check "Select subassembly part" to explode the entire subassembly of the currently selected object, which can satisfy the request of the whole assembly explosion and finally get the designed explosion result.

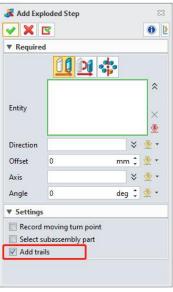






• Check "Add trails" option to record exploded trail with double dots line while creating an exploded view.





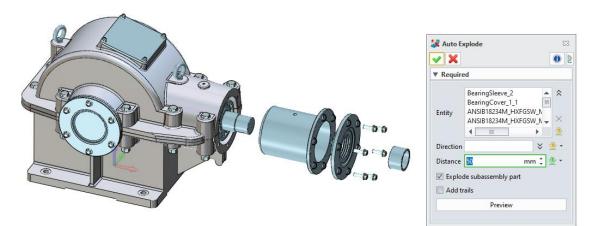
→ Where it is

Assembly Environment >> Exploded View >> Add Exploded Step



3.8.22.2 New Auto Explode

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

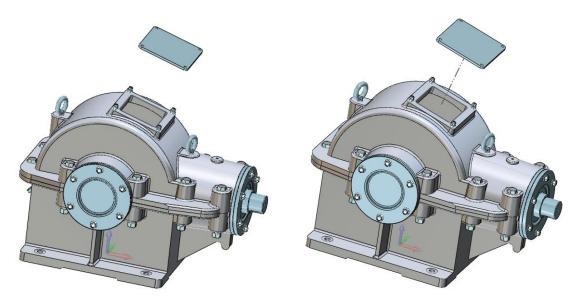


→ Where it is

Assembly Environment >> Exploded View >> Auto Explode

3.8.22.3 New Add Exploded Trail

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





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

Add Exploded Trail	×2 ••••••••••••••••••••••••••••••••••••
▼ Required	
1st entity	ः। 👲
2nd entity	-=1 👲
▼ Settings	
📝 Along XYZ	

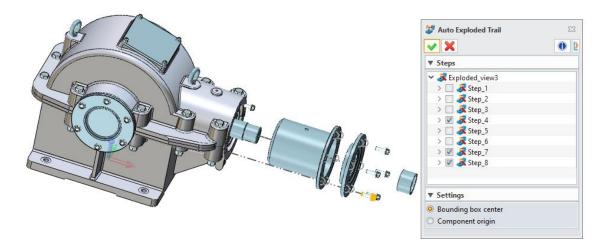
→ Where it is

Assembly Environment >> Exploded View >> Add Exploded Trail

3.8.22.4 New Auto Exploded Trail

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

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



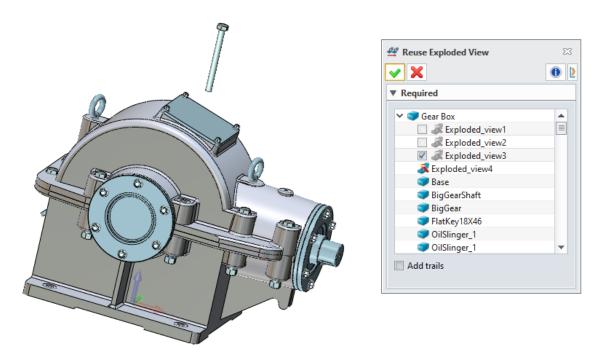
→ Where it is

Assembly Environment >> Exploded View >> Auto Exploded Trail

3.8.22.5 New Reuse Exploded View

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

• If the option "Add trails" is checked, the reused exploded view will carry with the exploded trails.



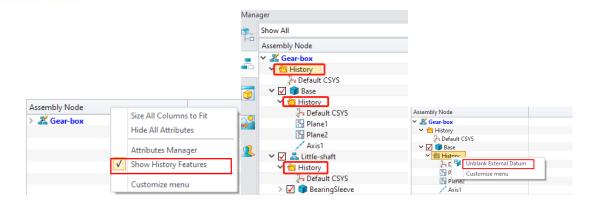
→ Where it is

Assembly Environment >> Exploded View >> Reuse Exploded View

3.8.23 New Show History Features

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





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

3.8.24 Pack Improvement

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

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

😨 Pacl	k File					X 🖵
	Name	Туре	Source directory		Target file	Destination directory
	Equation.xlsx	xlsx	C:\Users\Administrator\Des	ktop\12\	Equation.xlsx	C:\Users\Administrator\Documents\ZW3D
V	Part002.Z3PR	Part			Part002.Z3PRT	C:\Users\Administrator\Documents\ZW3D
•						•
🔽 Incl	lude componer	nts			🗹 Include (CAM Plan
🔽 Incl	lude Drawing S	heet			🗹 Include r	eferred files
🔲 Incl	lude componer	nts' Drav	ving Sheet/CAM Plan		Include s	suppressed components
🔲 Zip	the packed file	s				
Pre	fix				Suffix	
Folder	types All i	nto sing	le folder 🔹	1		
Pack to		nto sing p structi	le folder	٥\		
	Ren	nove em	pty folders	<u>h</u>		
			ure with common parent as assembly structure		Cancel	

→ Where it is

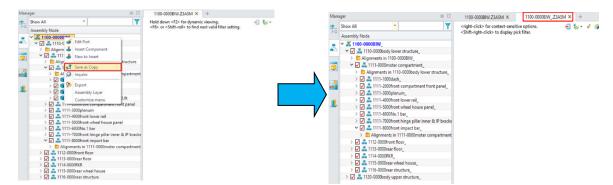
Assembly Environment >> File >> Pack



3.8.25 New Save as Copy

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

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



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

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

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

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

Use the small window **The second seco**



🐲 Save as Copy		Þ	XX 🐲		⊽ ⊠
Old object name ♥ ♥ N01-5301100-S01 > ♥ N01-530100-S01 > ♥ N01-5301040-S > ♥ N01-5301040-S > ♥ N01-5301050-S ♥ N01-5301050-S	Old file name N01-5301010-S.Z3ASM N01-5301100-S01.Z3ASM N01-5301030-S.Z3ASM N01-5301040-S.Z3ASM N01-5301050-S.Z3ASM	New obj A N01-5301 N01-5301 N01-5301 N01-5301 N01-5301			
Select all Rename Search Rename by rule Use template	Unselect all Search =	state.			
Use prefix Use suffix Ne Pack to folder C:\Users\Admi		el	1526.86 mm	HERE	

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

😴 Make Indepe	ndent Copy 🖂
✓ X	0
▼ Required	
Entity 1 picked	* 💆
▼ Settings	
Prefix	
Suffix	_1
Include child	components

→ Where it is

File >> Save as Copy

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

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



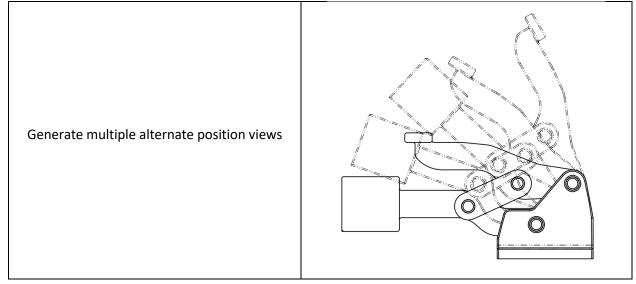
3.9 Drawing Sheet Design

3.9.1 New Alternate Position View

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

Insert alternate position view Select base view and part configuration	 Alternate Position View Alternate Position View Required Base view #57347 Part configuration pos1 Settings pos1 pos2 pos3 pos4
Generate alternate position view	

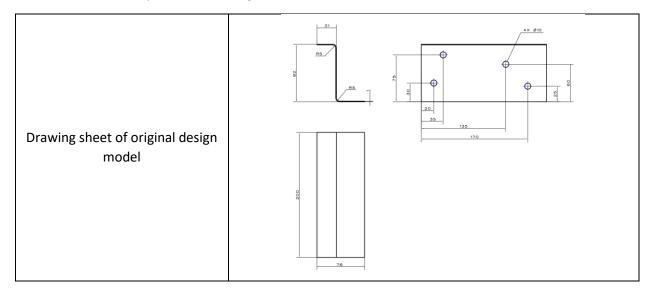




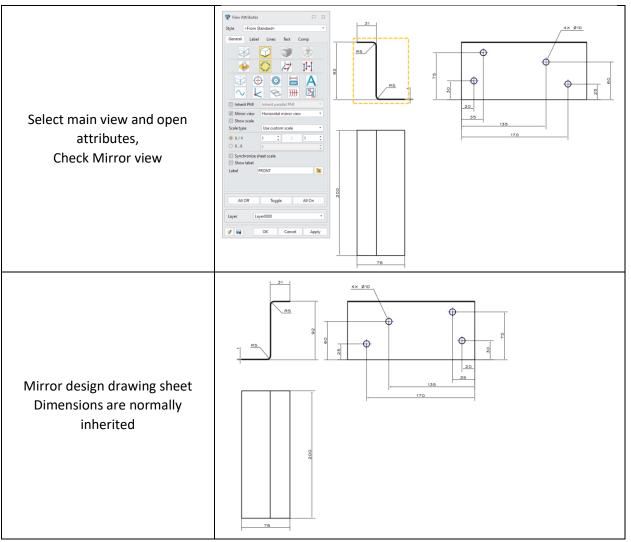
Drawing Sheet Environment >> Layout >> Alternate Position View

3.9.2 New Mirror View

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







Drawing Sheet Environment >> View Attributes >> Mirror View

3.9.3 ★New 3D Crop

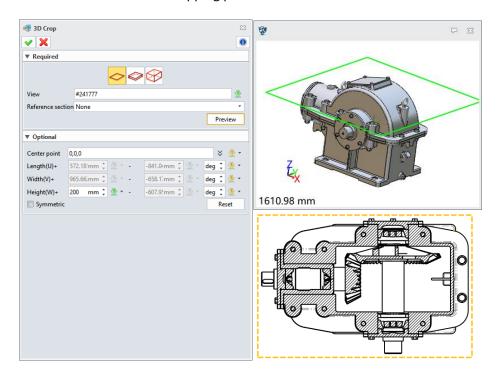
As for the engineering design of big assembly, it often needs to procure drawings by regions or by division of work or procure detailed layout sheet of partial space when design drawing sheet. Therefore, we added 3D Crop to limit the projection region of the drawing sheet and control. By changing the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

3.9.3.1 3D Crop

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



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

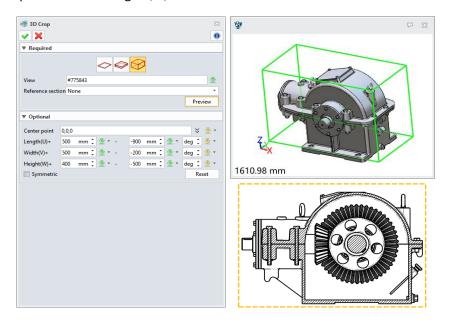


Section with slice: Use two planes to divide a piece of 3D space for projection view. Set positive and negative direction parameters of the current view and select the cropping section.

🧃 3D Crop		23	2	X 🖵
✓ X		0		
▼ Required			N	
View	#550392		0	
Reference sect	tion None	•	PLa C	
	Preview		CC CC	
▼ Optional			1000	
Center point	0,0,0 🛛 🕹 🔮	•	ZY See	J A
Length(U)+		-		
Width(V)+	875.32!mm 🗘 🕭 🐐 - 🛛 -607.9!mm 🗘 🗄 👻 deg 🗘 💆	-	1610.98 mm	
Height(W)+	200 mm 🗘 🕸 🔹200 mm 🗘 🕸 🕇 ideg 🛟 🕭	•		
Symmetric	Reset			
			0	
			Ŭ N	- and the and



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



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

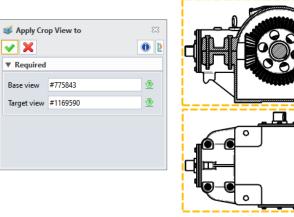
→ Where it is

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

3.9.3.2 3D Crop Apply Crop View to Specific View

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





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

3.9.3.3 Delete 3D Crop View

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

-					
	Lock Location	1			
	3D View 🕨		Projection	1	
	Zoom Limit	%	Auxiliary		
	Display mode	e	Full Section		
	Display Label	1	Aligned Section		
	Display Scale	<u>ð</u> e	3D Named Section		
	Display others	38	Bent Section		
	3D Measure		Broken Section		
	Component Visibility from Part	<u>6</u> 0	Detail		
\checkmark	View Orientation from Parent	-	Crop View		
			3D Crop	2	3D Crop
1	Erase	4	Break Line	1	Delete Crop View
	Blank	Г		1	Apply Crop View to
8	Entity Info			_	
"	Attributes				
	Move View to Sheet				
	Balloons link to BOM				
	Customize menu				

→ Where it is

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

3.9.4 **★**View Projection Efficiency Improvement

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

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

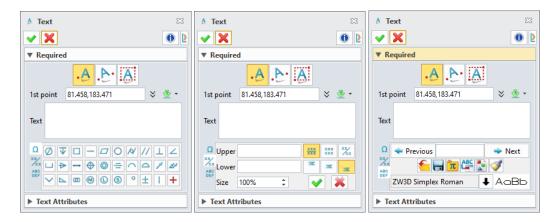
File size	101 MB	214 MB	385 MB	494 MB	850 MB	990 MB	2.11 GB
Speed Increase Rate	16.24%	17.85%	15.43%	20.84%	54.81%	95.52%	94.25%

→ Where it is

Drawing Sheet Environment >> Layout >> Projection View

3.9.5 Text Editor Improvement

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



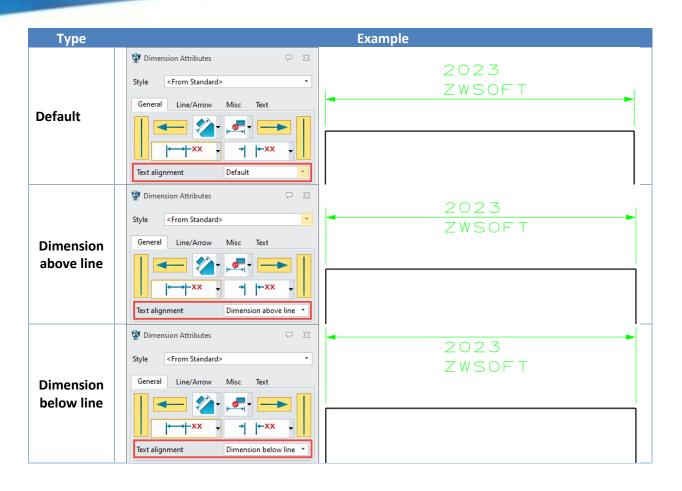
→ Where it is

Drawing Sheet Environment >> Drawing >> Text

3.9.6 New Text Alignment

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





Drawing Sheet Environment >> Dimension Context Menu >> Attributes

3.9.7 2D BOM Support Solid Attribute Filter

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

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



		🗸 🔚 P	art003		
			Solid(6)		
			Solid(0)		
			S2(Block2_Base)		
			S3(Block3_Base)		
			S4(Block4_Base)		
			S5(Block5_Base)		
			S6(Block6_Base)		
			History		
			V 🚰 Default CSYS		
			🗹 🧊 Block1_Base		
			🗹 🧊 Block2_Base		
			🗹 🧊 Block3_Base		
			🔽 🧊 Block4_Base		
			🔽 🧊 Block5_Base		
			🗹 🧊 Block6_Base		
			MODEL STOP HERE	-	
🖗 Shape Attributes					₽ 🛛
Standard User	Physical				
	-	1			
Property name	Туре	Sub-type	Data/Expression	Value	Unit/Format
1 cuboid	Number *	Constant	* 1 🗠 1		
2 <add a="" item="" new=""></add>	,				

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

🖉 BOM Filter	₽ %
BOM filter	
Attributes Operator Condition value	
cuboid • = • 1	Add
cuboid = 1	Delete
	Submit Edit
	Clear
Ok Cancel	

ID	Name	Co	⊃·s	t	1	۱L	ım	Ь	er	-	Q	U	a	n	ti	ŧу	Material
1	S2		÷		ľ.	÷								1			Aluminum
2	· .S4 ·				ŀ									1			Aluminum
3	S6		÷	÷	Ċ	÷	÷	÷		•	÷	÷		1		÷	Aluminum

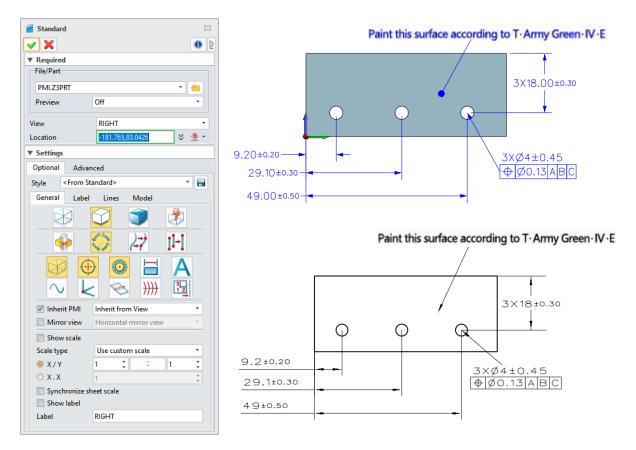
→ Where it is

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



3.9.8 New Inherit from View

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



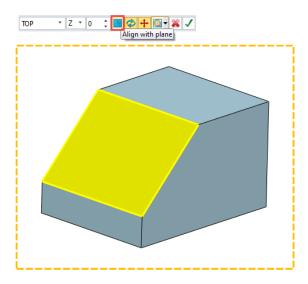
→ Where it is

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

3.9.9 New Align with Plane in Rotate View

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

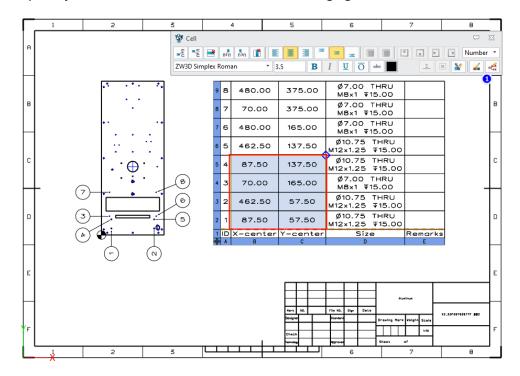




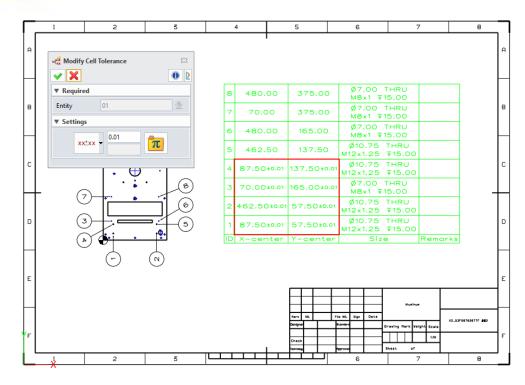
Drawing Sheet Environment >> Layout >> Edit View >> Rotate View

3.9.10Drawing Sheet Table Support Batch Modifying Tolerance

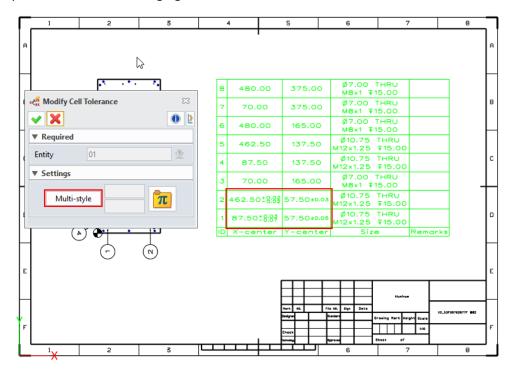
The hole table in the drawing sheet supported the setting of position tolerance in batch. User can box select multiple objects and set tolerance value as the following figure shown:







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



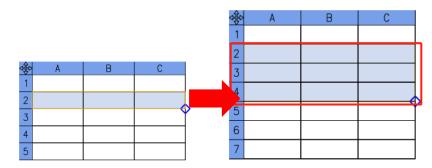
→ Where it is

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



3.9.11Drawing Sheet Table Support Inserting Row in Batch

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

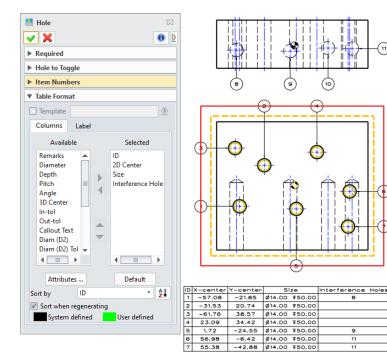


→ Where it is

Drawing Sheet Environment >> Table Context Menu >> Insert

3.9.12Hole Table Support Displaying Interference Hole

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







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

3.9.13Surface Finish Improvement

3.9.13.1 Drawing Sheet Surface Finish Support Configuration

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

✓ Surface Finish	\$3
✓ X	0 2
▼ Required	A
Ref. point	× 👲 -
▼ Settings	
4	f
Orientation 0	deg 🗘 💆 🔻
Lead. point	V 🕹 🕶
General Attribute	
Symbol Type	
Machining required	•
Grinding Symbol Layout	
Configuration Common Use	• • • ×
	-

→ Where it is

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

3.9.13.2 Drawing Sheet Surface Finish Remember Grinding Selection

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



✓ Surface Finish				23
✓ X			0	Þ
Required				A
Ref. point		×	• 🗄	
▼ Settings				
	4			
Orientation	0	deg ‡ ⇒	垫 👻	≡
Lead. point		\approx	垫 🝷	
General Attribu	ite			
Symbol Type				
JIS texture 2			•	
	$\nabla \nabla$			
🗹 Grinding				
-Symbol Layout				
	G			
25	- 25			
	/			
				-

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

3.9.14Additional List Block and Code List Block Improvement

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



⇔	А	В	С	D	E	E F		Н		
1		Tol Table								
2	v	Ø.5	ю	6	50	120	1000	2000		
3	<	5	6	30	120	400	2000	4000		
4	Tol	±0.2	±0.5	±Ø.8	±1.2	±2	±δ	±4		

→ Where it is

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

3.9.15 **★**New Batch Sheet Format Attributes

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

3.9.15.1 Sheets to Replace Page

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

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

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

		et Format Attributes						
hee	ts to R	leplace Sheet For	mat					
Sel	ect File	es						
P								
8	Name Name							
1	v C	:\Users\Administrato	or\Documents\Z	W3D\Drawing001.Z3				
	ect Ob							
Sel Filt	er	Show All						
Filt	er	Show All Name	Sheet	size File				
	er	Show All	Sheet					
Filt	er	Show All Name	Sheet	size File				
Filt	er	Show All Name	Sheet	size File				
Filt	er	Show All Name	Sheet	size File				
Filt	er	Show All Name	Sheet	size File				
Filt	er	Show All Name	Sheet	size File				
Filt	er	Show All Name	Sheet	size File				
Filt	er	Show All Name	Sheet	size File				
Filt	er	Show All Name	Sheet	size File				
Filt	er	Show All Name	Sheet	size File				

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



Filter	Show All 🔹	
	A4(H) (297.00 x 210.00 mm)	
	A4(V) (210.00 x 297.00 mm)	
	A4(H) (297.00 x 210.00 mm)	
	A3(V) (297.00 x 420.00 mm)	
	A3(H) (420.00 x 297.00 mm)	
	A2(V) (420.00 x 594.00 mm)	
	A2(H) (594.00 x 420.00 mm)	
	A1(V) (594.00 x 841.00 mm)	
	A1(H) (841.00 x 594.00 mm)	
	A0(V) (841.00 x 1189.00 mm)	
	A0(H) (1189.00 x 841.00 mm)	

3.9.15.2 Sheet Format Page

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

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

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

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

User can define whether replace the title block be checking/unchecking the checkbox.

• **Code list**: The behavior is the same as title block.

The file path that read by code list is \languages\en_US\resource\Templates_Code_List.Z3

• Additional list: The behavior is the same as title block.

The file path that ready by additional list is:

\languages\en_US\resource\Templates_Additional_List.Z3

- **Use border**: The behavior is the same as title block. When "None' is selected in source format, the sub-checkbox of Use border can read the content of Z3DRW.
- **Configuration:** Save the current configuration and the saved configuration can be re-read.



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

neets to Replace	Sheet Format				
Source format	A3_H(ANSI)				•
Sheet size	A3(H) (420.00 x 2	07.00			
Sheet size					
_	Width	420		Height	297
Title block	Title block(ANSI	_mm)			
Code list	<none></none>				
Additional list					
	Sheet Border				
	Bound				
Vse border	Trimming m	lark		Cente	ring mark
Use border	Partition				
	Horizontal	8	Length	52.500	
	Vertical	6	Length	49.500	
Configuration	<custom></custom>				
Preview					
FIEVIEW					
		Width: 4	20.0mm 297.0mm		

→ Where it is

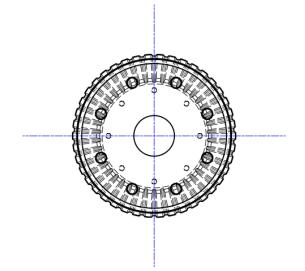
Drawing Sheet Environment >> Layout >> Sheet >> Batch Sheet Format Attributes

3.9.16 New Show Projected Datum

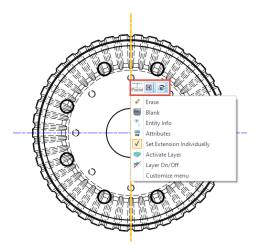
We added Show Projected Datum in the drawing sheet project, which can project the datum planes and datum axes of 3D environment to the drawing sheets. A line from datum projection is presented by a dotted line, whose attributes are the same as center line. User can modify the length of projected line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







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



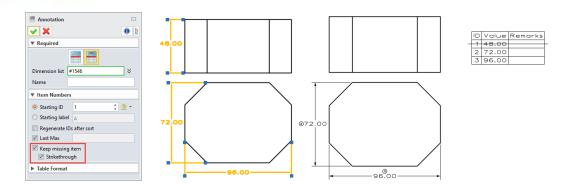
→ Where it is

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

3.9.17 New Keep Missing Item in Annotation

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

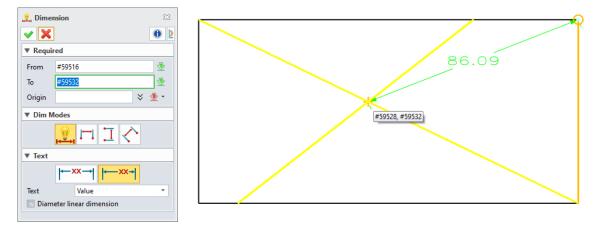




Drawing Sheet Environment >> Layout >> Table >> Annotation

3.9.18 Dimension Capture Improvement

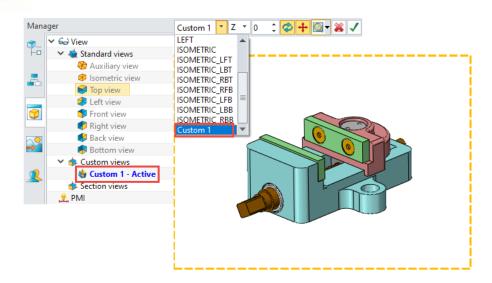
In the drawing sheet environment, the dimension supports capturing an interaction without manually using the filtering tool.



3.9.19 Support Custom View in Rotate View

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





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

3.9.20 Support Explode While Inserting Block

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

🧏 Insert		23
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File/Block		
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粗糙度3.2 五角星		
Preview	Off	•
Insertion point		× 👲 -
Explode		
▼ Scale		
Scale	0.5	* 🕭 *

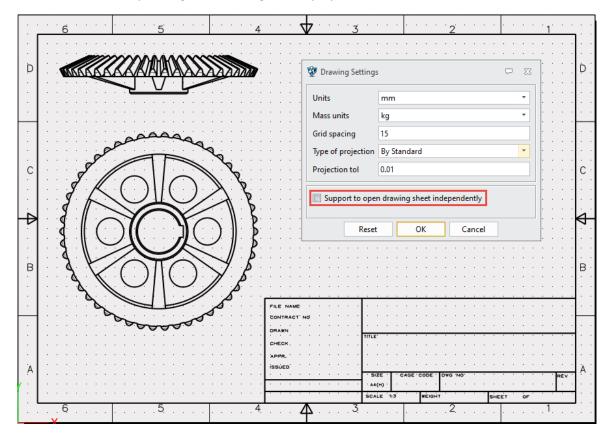
→ Where it is

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



3.9.21 New Support to Open Drawing Sheet Independently

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



→ Where it is

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

3.9.22 Weld in Drawing Sheet Supports Configuration

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

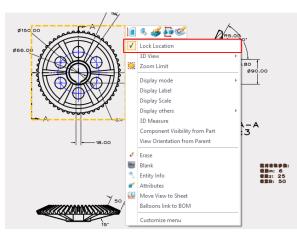


😓 Weld		83
✓ X		0
▼ Required		
Ref. point	1	¥ 👲 •
Location		× 👲 -
▼ Additional		
Multi jog points		¥ 👲 •
Multi leader points		՝ ջ .
▼ Settings		
Symbol Attribute		
Finish	None	
Contour	None *	
	Hone	
Groove angle		
6	- 12	
	· · _ · _ ^ 21(4)	
*		
	· ·	
Groove angle		
Contour	None -	
Finish	None -	
Include in weld table	Configuration ERW	a x
Include in weld table	Configuration ERW •	

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

3.9.23 New Lock Location in Drawing Sheet

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



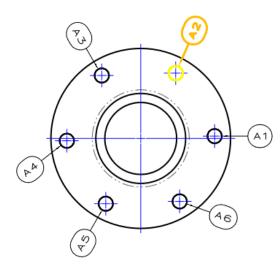
→ Where it is

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

3.9.24 Hole Table Improvement

3.9.24.1 New Highlight Hole Object

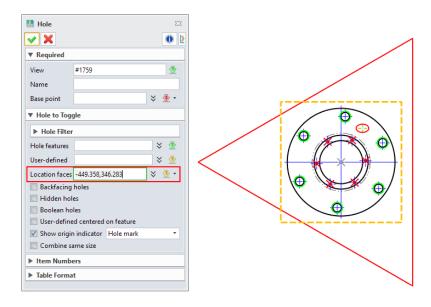
In the drawing sheet environment, user often needs to dimension holes among many hole parts and match up with hole table. The view now can highlight the hole while the hole dimension is selected so that user can locate hole position more clearly.



ID	X-center	Y-center	Size	
A1	41.15	101.22	Ø8.00	THRU
A2	19.46	136.22	Ø8.00	THRU
AЗ	-21.69	134.94	Ø8.00	THRU
A4	-41.15	98.66	Ø8.00	THRU
Α5	-19.46	63.66	Ø8.00	THRU
Α6	21.69	64.94	Ø8.00	THRU

3.9.24.2 New Location Faces in Hole

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





3.9.24.3 New Cell Tolerance

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

-		📑 nɨŋ 📩 👔		₩ ₩ x x x Number •	Modify Cell Tolerance
¢	A	В	С	D	
1	ID	X-center	Y-center	Size	▼ Required
2	A1	42.03+8:38	21.67	Ø8.00 THRU	Entity sheet 👲
3	Α2	7.28	46.07	Ø8.00 THRU	
4	AЗ	-31.23	28.16	Ø8.00 THRU	▼ Settings
5	Α4	-34.98	-14.14	Ø8.00 THRU	0.20
6	Α5	-0.22	-38.53	Ø8.00 THRU	xx±xx - 0.20 0.30
7	Α6	38.29	-20.63	Ø8.00 THRU	

→ Where it is

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

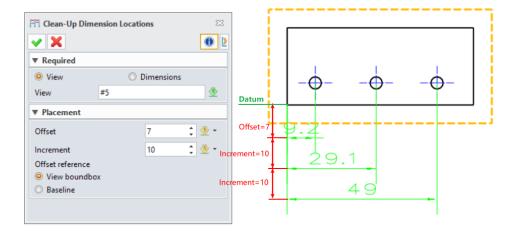
3.9.25 Clean-Up Dimension Locations Improvement

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

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

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





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

3.10 PMI

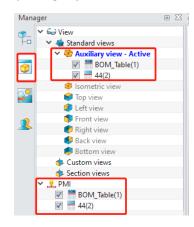
3.10.1 Support Inserting Table in PMI

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

	BOM 23 Image: Constraint of the second	
	Name	
	▼ Level Setting	
	Op-level only	
	○ Parts only	
- User Table	O By balloon only	• Insert Table
	○ Indented	••• Insert Table
	Max traverse depth	✓ X
▼ Required	▼ Settings	▼ Required
Name	Display configurations of the same part as one item	Point 🛛 🕹 🔹
▼ Settings	Derived parts as instances of source part	▼ Settings
Rows 3	Sync BOM table with part attributes	
· · · ·	Keep missing item	Origin Top-Left •
Columns 3	▼ Item Numbers	Plane 🔮
Attributes	Starting ID 1 🗘 💆 🔻	Rotate 0 deg 🗘 🖑 🔻
	Order Order by name	Attach to anchor point
	► Template	
	► Filter	



- User specifies the table name and sets table style. After clicking OK, an interface of Insert Table will prompt up, in which user can specify the place to insert table as well as set the plane to place table. The priority to place plane goes as: the plane specified in Plane widget >> PMI plane specified in View >> activated View Plane.
- 2) Insert Table is placed under the current view and PMI of Visual Manager, and supports the functions of editing, redefining, deletion, and placing adjustment.



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

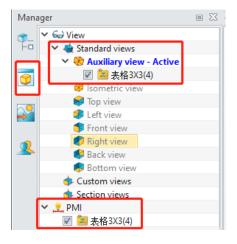
3.10.2 Insert Block in PMI

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

🧏 Insert	23
✓ X ⊑	0
▼ Required	
File/Block	
PMI-template.Z3DRW	• 📥
表格3X3	
表格3X3	
一般加工要求	
Preview Off	•
Insertion point -40.5,-10.5,8	× 🕹 -
Plane F2	₫
Rotate 0	deg 🗘 垫 👻
▼ Scale	
Scale 1	1 🕸 -
	• <u>•</u>



- 1) In the drawing sheet environment, the objects of tables, annotation, and symbols all can be selected to generate symbol blocks by block's command "Add".
- 2) The block can only be in the drawing sheet environment, and the new block is saved in the current drawing sheet.
- 3) When using inserting block in the modelling environment, user can select the corresponding drawing sheet filles and specify the symbol block to insert. User can complete the block inserting to the modeling environment by clicking Insert after setting placing point, placing plane and rotating angle.
- 4) The inserting Block is placed in the current view and PMI of the visual manager and supports the functions of editing, redefining, deletion, and placing adjustment.



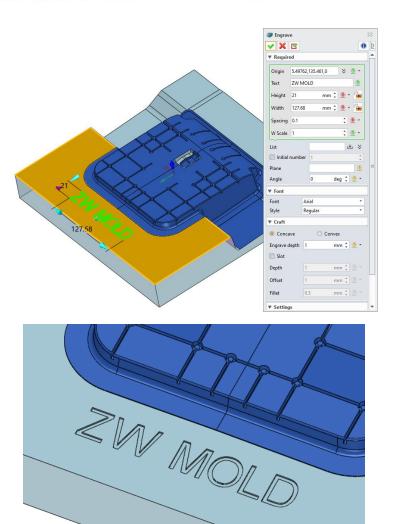
Part Environment >> PMI >> Annotation >> Insert

3.11Mold Design

3.11.1★New Engrave

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



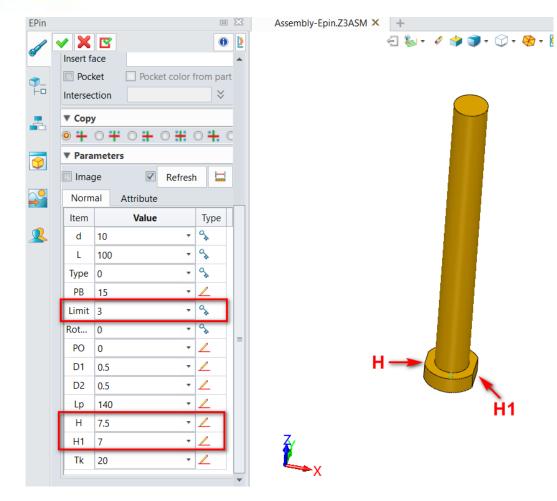


Part/Assembly Environment >> Mold >> Detail Design >> Engrave

3.11.2EPin Improvement

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



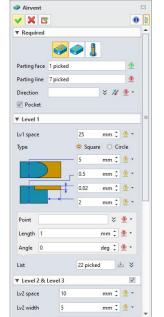


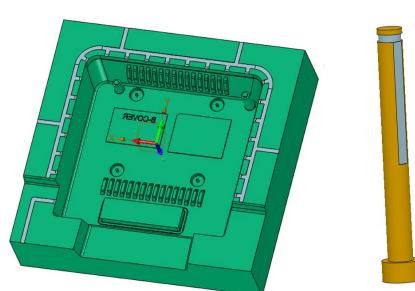
Mold >> Library >> Flat Epin

3.11.3New Airvent

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







Mold >> Detail Design >> Airvent

3.11.4 X New Oil Groove

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





 Oil Groove Carlo Grid wave Grid wave Plane Point Rotate Limited profile 	ve Cross Paralle	Eircle ntinod	grid	cross	Note
▼ Parameters					
Wave spacing	12	mm 🗘 💁 🔹			
Wave height	12	mm 🗘 💆 👻		$\bigcirc \bigcirc $	
Margin	3	mm 🗘 垫 🔹			
Min length	5	mm 🗘 💆 👻		00000	0 0 0 0 0
Section				wine or	hall
Square	🔘 Cir	cle	parallel	ring	ball
Width	1.5	mm 🌲 💆 *			
Depth	0.3	mm 🗘 💆 *			

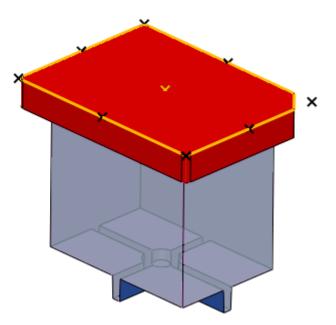
Mold >> Detail Design >> Oil Groove

3.11.5Electrode Datum Point Improvement

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



 [★] Electrode Datum Point [★] ★ [★] [★] ★ [★] 						
▼ Required						
Plane	F0	₫				
Point	P424	₫				
Assign to layer	Active layer	-				
New layer name						



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

3.11.6 Mold Workflow Improvement

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



→ Where it is

Part Environment >> Mold >> Product

3.11.7 Mold Assembly Optimization

3.11.7.1 Mold Project Mode Adjustment

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



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

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

💇 Mo	ld project	manage	er	₽ X			
▼ Rec	quired						
Name	2						
Mate	rial	NULL		•			
Shrin	k	1		÷			
▼ Cav	rity						
Cavity	у Туре		Single/Multi	*			
▼ Set	tings		Single/Multi Multi-Combin	ned			
File F	older			(
▼ Loa	d Asm-Tr	ee					
Load	Single/N	lulti		•			
File	MoldAsr	nTree\Si	ngle\ASM.Z3AS	M 😑			
		ОК	Cancel				

→ Where it is

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

3.11.7.2 **★**New Load Asm-Tree in Mold

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

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

The mold assembly configuration template supports being read, selected, and saved. Select the mold assembly template through path (select Z3 files for the multi-root template and select mold project top

assembly single-root files for the single-root template). Click 🔲 to save the selected template or click

to delete the selected and saved template. Select the template from the drop-down list in "Load", which can be used to create mold project.

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



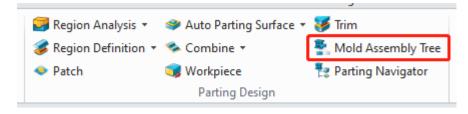
👰 Mold project	manage	r	\Box	23
▼ Required				
Name				
Material	NULL			•
Shrink	1			÷
▼ Cavity				
Cavity Type		Single/Multi		•
▼ Settings				
File Folder				-
▼ Load Asm-Tr	ee			
Load Single/M	1ulti		•	
File MoldAs	mTree\Sin	gle\ASM.Z3ASI	м	
		X		
	ОК	Cancel		

→ Where it is

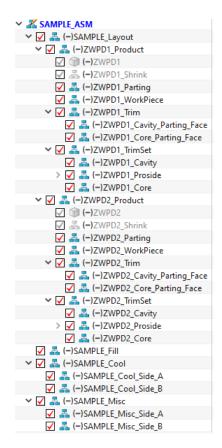
Part/Assembly Environment >> Mold >> Product >> Mold Project

3.11.7.3 New Mold Assembly Tree

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







→ Where it is

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

3.11.7.4 New Trim Node

Trim node is added in the product of assembly tree.

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



→ Where it is

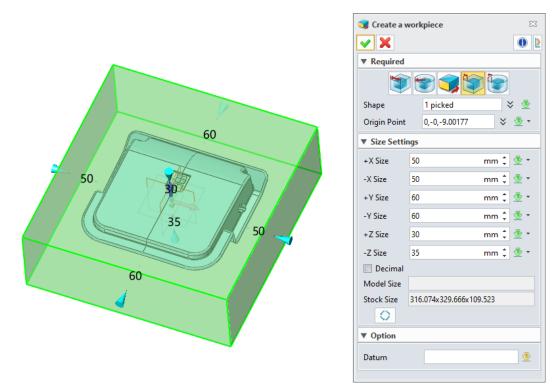
Assembly Environment >> Mold >> Parting Design >> Trim



3.11.8 **★** Workpiece Improvement

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

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



→ Where it is

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

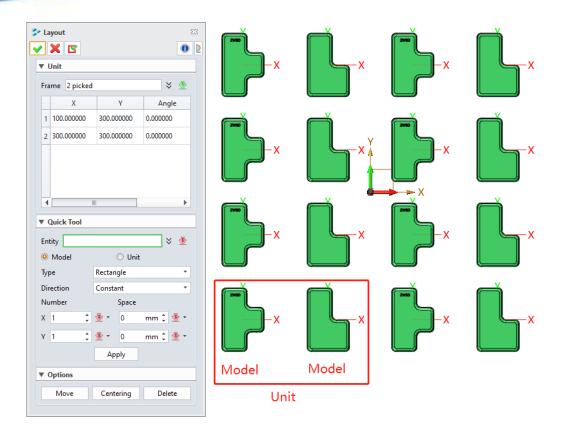
3.11.9 **★**Layout Improvement

1) Unit Layout/Overall Layout

The new layout brings more convenient functions.

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

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



2) Layout Tools

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

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

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	X							0
Ŧ	Unit							
Fra	ame	2 pic	ked				×	₫
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1	100	.00000	0	300.0	00000	0.000	000	
2	300	.00000	0	300.0	00000	0.000	000	
								•
▼ Quick Tool								
•	Quic	k Tool						
-	-	k Tool					~	-
En	tity				0 IIit		*	<u>₽</u>
En	tity Mo				◯ Unit		*	€
En © Ty	Mo pe	del		Rectan	gle		*	<u>ل</u>
En © Tyr Dir	tity <u>Mo</u> pe rectio	del on	[Rectan Consta	gle int	:	*	* *
En © Tyj Dir	Mo pe	del on	[gle		*	 <!--</td-->
En © Tyj Dir	tity <u>Mo</u> pe rectio	del on	[gle int	mm		- -
En © Tyj Dii Nu	ntity Mo pe rectio	del on	[Consta	gle int Space		€	- -
En © Tyr Dir Nu X	ntity Mo pe rection umbe	del on	[Consta	gle int Space 0	mm	€	- -
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3) Move, delete, centering

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

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

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		Х			γ	A	Angle		
1	100.	00000	0	300.0	00000	0.000	000		
2	300.	00000	0	300.0	00000	0.000	000		
•								•	
▼ (Quicl	c Tool							
Ent	tity						×		
0	Mod	del			🔘 Unit				
Тур	be			Rectar	ngle			•	
Dir	ectio	'n		Const	ant			•	
	imbe	r			Space				
Х	1		÷	- 🛃	0	mm	1	•	
Y	1		÷	- 🛃	0	mm	: 🛛	•	
				Ap	ply				
• (Optic	ons							
	M	ove		Cent	tering	D	elete		

→ Where it is

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

3.11.10 ★Standard Part Mechanism Improvement

Place type

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

Parent

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



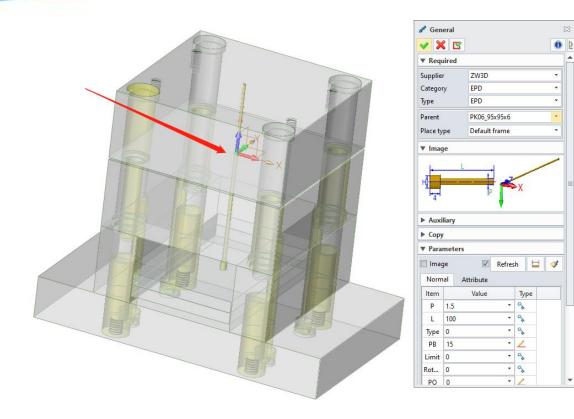
External Reference Read

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

13	##EXPRI	ESSIONS				
14	\$ZPlace=-	[&MB_ZWN	A:BP0H]-[&MB_ZWM:CP0H]+[&MB_ZWM	I:GAP0FP]	+[&MB_ZV	/M:FP0H]-0.5
15						
16	##PARE	T				
17	PK06_95	x95x6				

🖋 General			23
🗸 🗙 🖸			0
Required			
Supplier Category Type	ZW3D EPD EPD		•
Parent	Current		-
Place type	Default		-
Face			ا
Point		¥ 🔮	-
Round	0.01	Associate	
X align		¥ 🔮	-
Z align		¥ 🍕	-
Flip Z direct	ion		
▼ Image			
		χ	
Auxiliary			
Pocket Intersection	Pocket colo	r from part 🕺 💆	
▶ Сору			
Parameters			
🔲 Image	R	efresh 🔛	1
Normal A	ttribute		
ltem	Value	Туре	•





→ Where it is

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

3.12ZWStructure

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



→ Where it is

Part/Assembly Environment >> Simulation



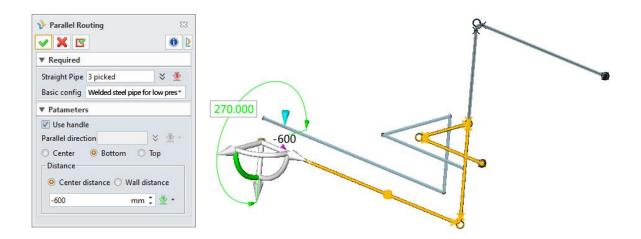
4 Pipeline

4.1 Piping/Tubing Design

4.1.1 ★New Parallel Routing

In the pipeline environment, we added the function of parallel routing to help user create parallel routing according to the existing pipeline which promotes the efficiency and quality of piping.

- Parallel direction supports handle rotation, angle input and selection of location.
- Alignment placement supports center alignment and wall alignment. The wall distance includes bottom alignment and top alignment.
- Parallel distance supports center distance and wall distance.



→ Where it is

Piping >> Route >> Parallel Routing

Routing >> Route >> Parallel Routing

4.1.2 **★**New Hanger

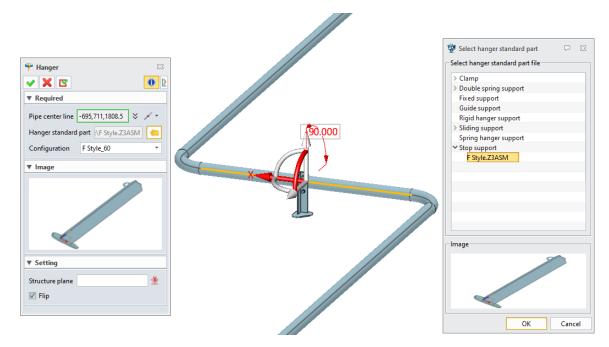
In the pipeline environment, we added a simple function of hanger to create a support hanger for a single pipe.

Note: Only a single link hanger is currently supported.

- Support previewing and selecting configuration.
- Support selecting structure plane for alignment.



- Support handle rotation, angle input and movement along pipeline.
- Support flipping.



→ Where it is

Piping >> Hanger >> Hanger

Routing >> Hanger >> Hanger

4.1.3 Piping Spool Drawing Optimization

We optimized the piping spool drawing to improve the customization of pipe drawing based on customer's different requirements on the piping spool.

- Add custom page number generation.
- Read template information such as dimension style and view.
- Support setting the location of view in sheet.



😵 Piping Spool Drawing 🖓 🛛
🖋 Spool 🚿 Sheet 🌗 View 🛗 Table
Size/Template Piping-A4_V(EN) Custom ping-A4_V(EN).Z3DRW
Custom page number generation 10
OK Cancel

→ Where it is

Piping >> Drawing >> Spool Drawing

Routing >> Drawing >> Spool Drawing

4.1.4 Routing BOM Optimization

We optimized the piping BOM to solve the piping BOM statistics information.

- Flange fastener statistics.
- Weight and length keep 3 decimal places.
- Optimize template.

Catalog	Type	Routeline Name	Instance Name	Standard	Specification	Material	Length/m	Number	Piece ∛eight/Kg	Total Weight/Kg
Fastener	Bo1t	Routing001	STUD	HG/T 20613	M16X90X0.09	35 Steel		32	0.144	4.608
Fastener	Nut	Routing001	hex nut	HG/T 20613	M16X90-I	25 Steel		64	0.029	1.856

→ Where it is

Piping >> BOM >> Routing BOM

Routing >> BOM >> Routing BOM



5 CAM

5.1 **★**Tool Speed and Feed Overall Upgrade

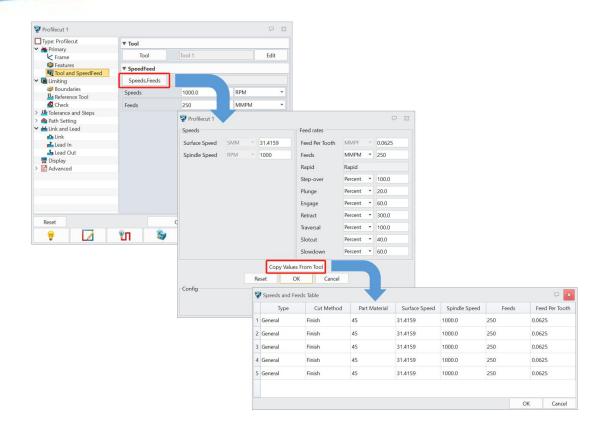
We had a comprehensive innovation to speed and feed function in ZW3D 2023 including:

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

Туре	Cut Method	l Pai	rt Material	Surface Speed	Spindle Speed	Feed	ls Feed Per Tooth
I General	Rough	45		31.4159	1000	250	0.0625
2 General	Finish	45		31.4159	1000	50	0.0125
3 General	Finish	AL		94.2478	3000	300	0.025
							Add Delete
Surface Speed Spindle Speed Attribute Type Cut Method Part Material	SMM 31.4159 RPM 1000 General Rough 45 1000		Feed Per Toot Feeds Rapid Step-over Plunge Engage Retract Traversal Slotcut Slowdown	MMPM * 250 Rapid Percent * 10 Percent * 20 Percent * 30 Percent * 11	00.0 0.0 0.0 00.0 00.0 00.0 0.0		

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

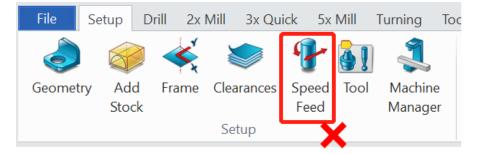




5.2 **★**Tool Library Overall Upgrade

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

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



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



(A	В	C	D	E	F	G	н	1	J	K	L	M	N	0	Р	Q	R	S	Т	U	V	W
#																						
#																						
	/ing fields are defined for "Drill	type.																				
	th - Tool length																					
	gth - Flute length																					
	ameter - Shank diameter																					
# Flutes	- Flutes																					
# Relief Ang	gle - Relief angle																					
# Tip Angle	- Tip angle																					
# Cutter Dia	ameter - Cutter diameter																					
# CamPlan	Unit - 1:mm unit,2:in unit																					
#																						
FORMAT	Name	Holder	Type	Sub Type I	Index			Flute Leng	Shank Dia	Flutes	Relief Ang	Tip Angle	Cutter D	ia Class	ID	D	н	Coolant	Spindle E	i Pre Reg O	Pre Reg Tcs	Juppli
SHP DATA	1 mm Twist Drill-Jobber		Drill		1	1	28.3	17	1	2	6	118	1	General				Flood	CLW	spot drill	6 mm NC ta	any
SHP DATA	1.25 mm Twist Drill-Jobber		Drill		2	1	35.3	22	1.25	2	6	118	1.25	General				Flood	CLW		6 mm NC 14	
SHP DATA	1.5 mm Twist Drill-Jobber		Drill		3	1			1.5	2			1.5	General				Flood	CLW		6 mm NC 1a	
SHP DATA	1.6 mm Twist Drill-Jobber		Drill	2	4	1	35.3	22	1.6	2	6	118	1.6	General				Flood	CLW	spot drill	6 mm NC 1a	inv
SHP DATA	1.75 mm Twist Drill-Jobber		Drill		Ś	1	38.3		1.75	2	6	118	1.75	General				Flood	CLW		6 mm NC 14	
SHP DATA	1.8 mm Twist Drill-Jobber		Drill	1	6				1.8	5			1.8	General				Flood	CLW		6 mm NC 1a	
SHP DATA	2 mm Twist Drill-Jobber		Drill		7		38.3	25	2	5		118	5	General				Flood	CLW		6 mm NC ta	
SHP DATA	2.05 mm Twist Drill-Jobber		Drill	1			41.3		2.05	5		118	2.05	General				Flood	CLW		6 mm NC 1a	
SHP DATA	2.5 mm Twist Drill-Jobber		Drill	-			40.95			5			2.5	General				Flood	CLW		6 mm NC 14	
SHP_DATA	2.7 mm Twist Drill-Jobber		Drill		10		50.95		2.5	2		118	2.5	General				Flood	CLW		6 mm NC ta	
	2.9 mm Twist Drill-Jobber		Drill		10	5				2			2.9									
SHP_DATA					11	1	50.95			5		118	3	General				Flood	CLW		6 mm NC 14	
SHP_DATA	3 mm Twist Drill-Jobber		Drill		12		56.95		3.3	2		118	3.3	General				Flood	CLW		6 mm NC ta	
SHP_DATA	3.3 mm Twist Drill-Jobber		Drill							2			3.3	General				Flood	CLW		6 mm NC 14	
SHP_DATA	3.5 mm Twist Drill-Jobber		Drill		14		56.95							General				Flood	CLW		6 mm NC ta	
SHP_DATA	3.7 mm Twist Drill-Jobber		Drill		15		56.95			2		118	3.7	General				Flood	CLW		6 mm NC ta	
SHP_DATA	4 mm Twist Drill-Jobber		Drill		16					2		118	4	General				Flood	CLW		6 mm NC ta	
SHP_DATA	4.2 mm Twist Drill-Jobber		Drill		17					2			4.2	General				Flood	CLW	spot_drill	6 mm NC Sa	iny
SHP_DATA	4.5 mm Twist Drill-Jobber		Drill		18		66.95		4.5	2		118	4.5	General				Flood	CLW	spot_drill	6 mm NC ta	iny
SHP_DATA	4.6 mm Twist Drill-Jobber		Drill		19		66.95			2			4.6	General				Flood	CLW	spot_drill	6 mm NC ta	iny
SHP_DATA	5 mm Twist Drill-Jobber		Drill		20					2		118	5	General				Flood	CLW	spot_drill	6 mm NC ta	any
SHP_DATA	5.5 mm Twist Drill-Jobber		Drill		21		75.95		5.5	2		118	5.5	General				Flood	CLW	spot_drill	12 mm NCa	iny
SHP_DATA	5.6 mm Twist Drill-Jobber		Drill		22		75.95			2		118	5.6	General				Flood	CLW	spot_drill	12 mm NCa	any
SHP_DATA	6 mm NC Spot Drill		Drill		23		76			2		90	6	General				Flood	CLW			iny
SHP DATA	6 mm Twist Drill-Jobber		Drill	1	24		76.6			2		118	6	General				Flood	CLW	spot_drill	12 mm NCa	any
SHP DATA	6.5 mm Twist Drill-Jobber		Drill	5	25	1	79.6	73	6.5	2	6	118	6.5	General				Flood	CLW	spot drill	12 mm NCa	any
SHP DATA	6.6 mm Twist Drill-Jobber		Drill		26	1	79.6	73	6.6	2	6	118	6.6	General				Flood	CLW		12 mm NCa	
SHP DATA	6.75 mm Twist Drill-Jobber		Drill		27	1	79.6	73	6.75	2	6	118	6.75	General				Flood	CLW		12 mm NCa	
SHP DATA	6.8 mm Twist Drill-Jobber		Drill		28		79.6		6.8	2			6.8	General				Flood	CLW		12 mm NCa	
	llipop Corner Round Wh	and in These				m Coun		Countersi		Center		-	4							-purm		,

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

Tool Shape Libra	ry																\sim
brary																	
D:\Distribute\2700	0_x64\I	anguages\	en_US\res	source\001	METRIC T	OOLS.xlsx											_
ool Type Name	Na	me	-						X	Y					Þ	Parameters	Display
Search		Name	Holder	Tool L	Cut Le	Flutes	Cutter	Shank	Radius	Class	ID	D	н	Coola	1	Name	\checkmark
F-Slot	1	12/1 m		35.5	22.5	4	12	12	1	General				Flood	2	Holder	
ollipop Corner Round	2	12/1.5		35.5	22.5	4	12	12	1.5	General				Flood	3	Tool Length	V
Wheel ThreadMill	3	12/2 m		35.5	22.5	4	12	12	2	General				Flood	4	Cut Length	\checkmark
Chamfer	4	12/2.5		35.5	22.5	4	12	12	2.5	General				Flood	5	Flutes	V
ap leam	5	12/3 m		35.5	22.5	4	12	12	3	General				Flood	6	Cutter Diameter	
Counterbore Countersink	6	13/3.5		35.5	22.5	4	13	12	3.5	General				Flood	7	Shank Diameter	
Drill	7	14/4 m		35.5	22.5	4	14	12	4	General				Flood	8	Radius	
Center Aill	8	16/4.5		35.5	21.5	4	16	16	4.5	General				Flood	9		
	9	16/5 m		35.5	21.5	4	16	16	5	General				Flood	= 4		-
		22/6 m		38.5	24.5	4	22	16	6	General				Flood	-		
		22/7 m		35	20.5	4	22	25	7	General				Flood			
		25/8 m		35	20.5	4	25	25	8	General				Flood			
		26/9 m		41.5	20.3	4	26	25	9	General				Flood			
		28/10		41.5	27	4	28	25	10	General				Flood			
		35/12		44.5	30	4	35	25	12	General				Flood			
		42/14		47.9	33	4	42	25	14	General				Flood			
		45/15		47.5	33	4	45	25	15	General				Flood			
		48/16		49.5	35	4	48	25	16	General				Flood			
		48.5/1		54	39.5	4	48.5	25	18	General				Flood	Ŧ		
	4			11													

• New duplication of name check mechanism

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

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

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

💯 ZW3D			⊽ ⊠
0	Tool [12.5/6 mm T-Slot C	Cutter] had already existed. Do you wa	nt to overwrite it or create a new one?
	Overwrite	Create	Cancel

5.3 ★New Multi Work Station

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



💱 Output Setting	I			⊽ %					
Select Machine									
Cr	eate		Edit						
Station Setting	g								
Station Information									
Sta	tion	Fran	ne	+					
	1	G54	1	*					
	2	G55	5						
	3	G56	5						
Output									
	e r								
Type of Outpu		lority		•					
Using Subrout	ines Yes			-					
Incrementa	l coordinates	Absolute	coordinat	es					
Setting									
Part Id	Spiralcut 1								
Programmer	Administrator								
Toolpath Space	Machine								
Relation Frame	Machine								
Tool Changes	Output			*					
Coolant	From Tool			•					
Comment									
Output File									
Spiralcut 1									
CL File	NC Code File	Operation List	Full Mac	hine Simulate					
				ОК					

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

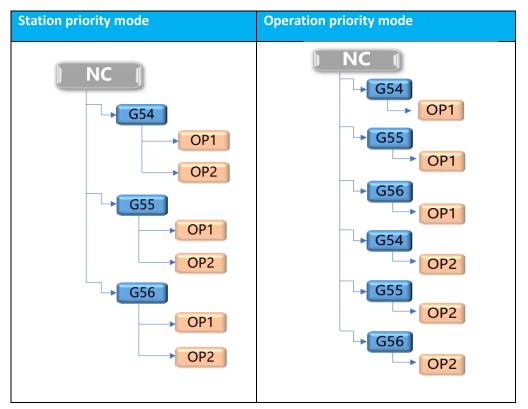


Station Setting					
Station Information					
Station		Frame	÷		
1		G59	×		
2		G54.1P1			
Output					
Type of Output	Station pr	iority	•		
Using Subroutines	Yes		-		
Incremental coo	rdinates	Absolute coordinates			

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

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

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

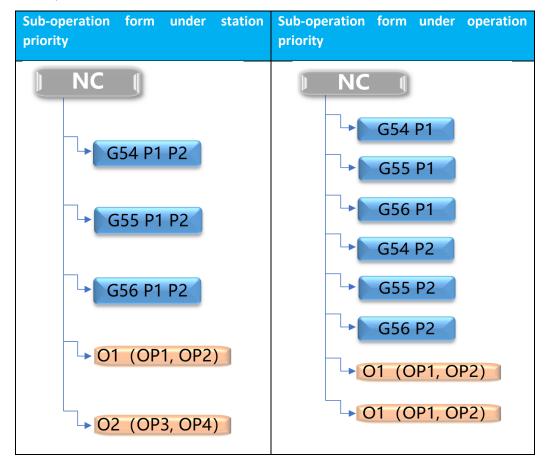


NC code structures under the two modes are as below:

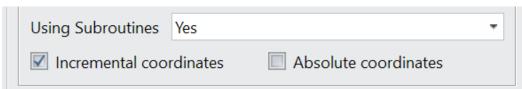


• Support output operation in the form of sub-operation

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



• Output form under subroutines



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

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



5.4 Analysis Toolpath Overall Upgrade

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

New UI

🗳 Analysis Toolpa ✔ 🔀		X
Analysis		
Analysis type	Collision	
Holder clearance	1	
Tool clearance	5 \$	
▼ Tool Path		
01 Rough Offs 02 Spiralcut 1 03 Spiralcut 2 04 Spiralcut 3 05 Spiralcut 4	Tool 1 Tool 1 Tool 1	
Part Pa	art003	
Stock Pa	art003_Stock.1	
▼ Display		
Style		
Width		
Color		
	Analyze	
OK	Reset Cancel	

• Support two modes: collision check and gouge check

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

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

• Newly support check multiple operations at the same time.

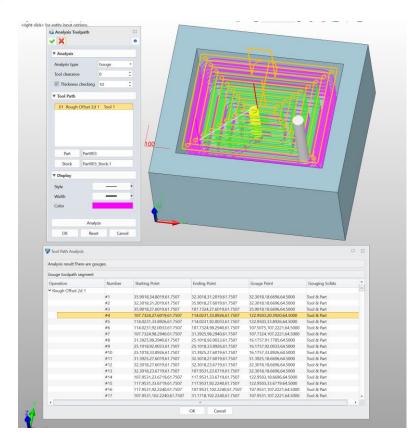
🎽 Analysis Toolpath				
✓ X		0		
▼ Analysis				
Analysis type	Collision	-		
Holder clearance	1	¢		
Tool clearance	5	÷		
▼ Tool Path				
01 Rough Off 02 Spiralcut 1 03 Spiralcut 2 04 Spiralcut 3 05 Spiralcut 4	Tool 1 Tool 1 Tool 1			
Part P	art003			
Stock P	art003_Stock.1			
▼ Display				
Style		•		
Width		•		
Color				
	Analyze			
ОК	Reset Cancel			

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

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

• Support displaying collision/gouge toolpath.

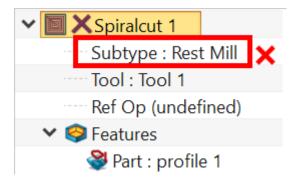




5.5 ★New Reference Tool in 2 Axis Operation

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

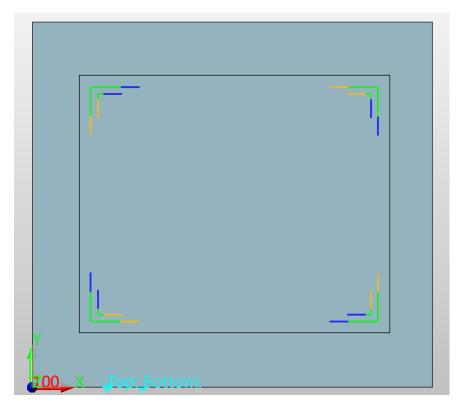
• Remove rest mill subtype in the operation.



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

💱 Contourcut 1					$\overline{\nabla}$	23
🖉 Type: Contourcut	▼ Refere	nce Tool				
💙 📥 Primary			[-
K Frame	Referer	nce Tool	Tool 2			
🤤 Features	Expand A	rea	2			
Tool and SpeedFeed						
🗸 📠 Limiting						
🗇 Boundaries						
Preference Tool						
🙋 Check						
Interance and Steps						
> 🔷 Path Setting						
💙 📥 Link and Lead						
💁 Link						
📥 Lead In						
📥 Lead Out						
冒 Display						
> 🛃 Advanced						
Reset		Ca	lculate	ОК	Canc	el
🦁 🗾	1		1.		1	

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





5.6 ★FMS Support 5-axis Index Milling Simulation

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

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

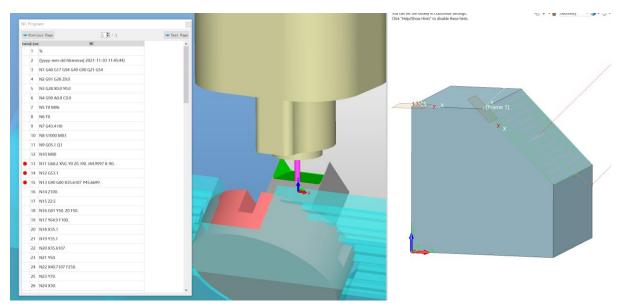
Intrinsic rotation: Revolve around the three axes of object coordinate. During the object revolving, its coordinate axes move around the object.

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

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



The following figure is Fanuc system index milling FMS:



5.7 ★FMS Support Auto On/Off RTCP

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

• Remove RTCP switch in FMS



ZWFMS - C:\Users\Administrator\AppData\Roaming\ZWSOFT\ZW3D\2630_x64\output\temp\FMSJob\jo	b.job -	- 🗆 🗙	<
File View Simulation Analysis Help			
🗲 🔿 🖂 🕨 🔰 ———— 🛯 🖳 🗶 🧊 🙆 🧶 🖫 🏶 🏶 🖱	📃 🗖 🖓 🏦	<u></u>	
Job Tree S ×			
Image: Setting Image: Step on clash Image: Step on clash <		×	
Job Tree Axes Properties 1 Pick Info Analysis Setting			
NC Program 🕫 ×			
2 revious Page 1 → Next Page -0.02			
tatu Line NC			
2 N1 G40 G17 G94 G49 G90 G21 ···			
3 N2 G91 G28 Z0.0 Background			
4 N3 G28 X0.0 Y0.0]Light Blue 🔽 Light	Grey	
NC Program Breakpoint 5X Control			
Error List 5 ×			
OK Cancel			

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

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

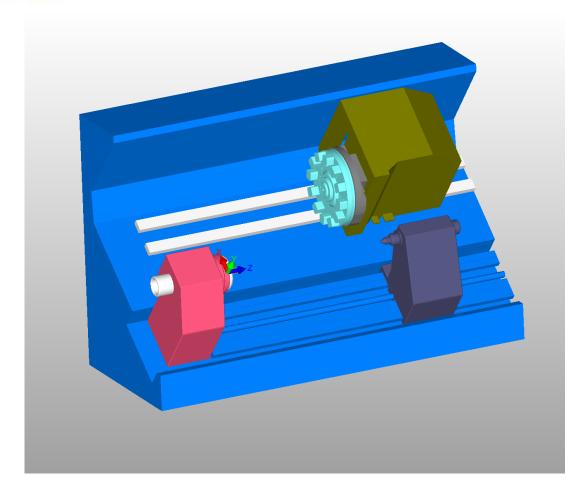
5.8 FMS Support Mill-Turn Simulation

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

• Add Mill-Turn machine tool case.

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

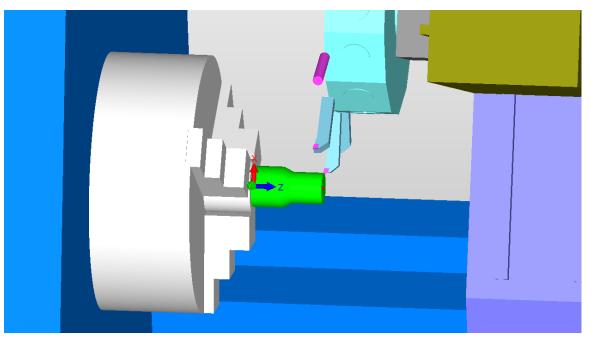




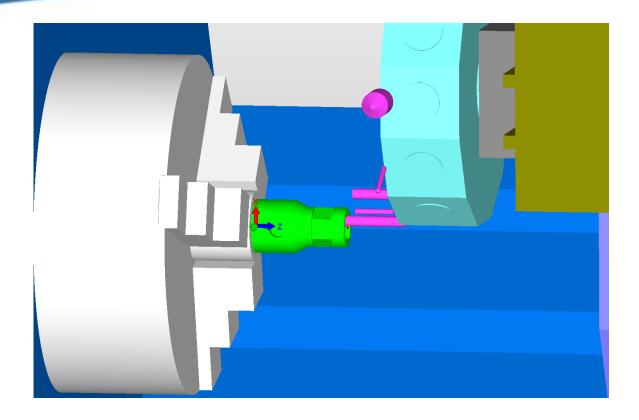
• Add turret setting (Only for Mill-turn machine tool, other type of machines do not need to set)

2	ZWFMS				×
	Turret	Tool Name		Tool ID	
1	Toolhead1		•		
2	Toolhead10	Drill4_R	•	10 -	
3	Toolhead11	Chamfer	•	11 •	
4	Toolhead2	OD2	-	2 -	
5	Toolhead3	OD3	•	3 •	
6	Toolhead4	D10R0_R	•	4 -	
7	Toolhead5	D10R0_F	•	5 •	
8	Toolhead6	DSR0_F_R	•	6 •	0.0
9	Toolhead7	DSRO_R_A	•	7 -	
10	Toolhead8	D8R0_F_A	•	8 -	
11	Toolhead9	Drill4_A	•	9 -	
	0K Ca	ncel			

• Support Mill-Turn machining FMS







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

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



💱 Output Setting	1			\Box	23
Select Machine					
Cr	eate		Edit		
Station Settin	g				
Setting					
Part Id	NC				
Programmer	Administrator				
Toolpath Space	MCS				
Relation Frame	Machine				
Tool Changes	Output				•
Coolant	From Tool				•
Comment					
Output File					
NC				-	
CL File	NC Code File	Operation List	Full Mach	ine Simul	ate
				ОК	

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



🦉 Output Setting	9			\Box	23
Select Machine					
Cr	eate		Edit		
Station Settin	g				
Setting					
Part Id	NC				
Programmer	Administrator				
Toolpath Space	Local				
Relation Frame	Machine				
Tool Changes	Output				*
Coolant	From Tool				-
Comment					
Output File					
NC				<u> </u>	
CL File	NC Code File	Operation List	Full Mac	hine Simu	ulate
				OK	

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

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



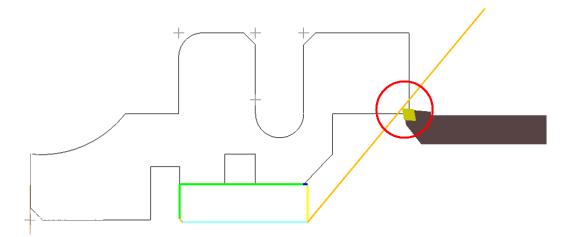
💱 Machine Manager					×
Definition			Library		
Machine Name	Machine 1		Machine		
Class	3-Axis M.C.	-			
🔲 Туре	Vertical	-			
Subtype	Rotating Head	-			
Post-Processor	ZWPost	*			
Post Configuration	ZW_Fanuc_3X				
XY Arcs	Yes	*	1		
YZ Arcs	No	Ψ.			
ZX Arcs	No	*			
Spacial Arc	No	~			
Helical Arc	No	*			
Check MULTAX	Yes	*			
MULTAX	No	-			
Accurate RAPIDs	No	Ψ.			
Scale	1				
#.xxxx	5				
Rewind	Yes	*			
Increment	1				
CUTCOM	None	-			
Offset Registers					
NC Extension	.nc			Delete	
Definition File	machine_all.mdf	-	Options		
Open I	Machine Definition File		Add T	o Library>	Apply Filter>
ОК	Apply	N	ew	Reset	Cancel

5.9 New Approach/Departure Motion Types in Turning Operation

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

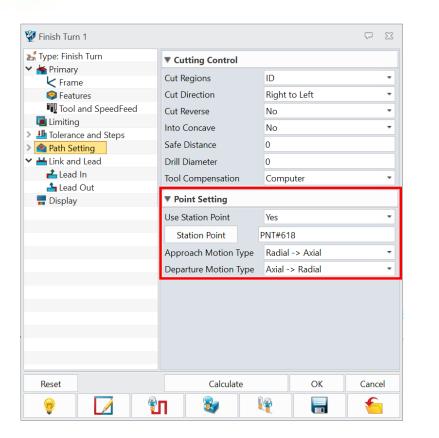
In the previous versions, tool could only move in the "Direct" way, which caused collision with workpiece.

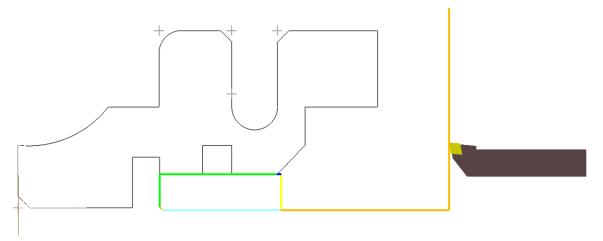




The two options "Radial->Axial" and "Axial -> Radial" in the new version can allow tool to enter or leave workpiece safety.

ZŴBD



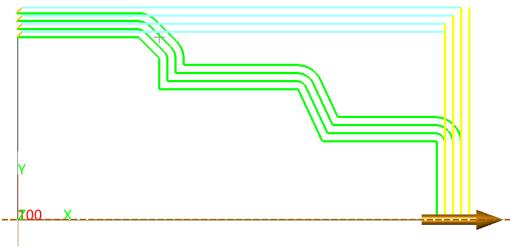


5.10New Multi-level Function in Finish Turing Operation

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

ZWJD

💱 Finish Turn 1			₽ 🛛		
🎢 Type: Finish Turn	+ IDICIAILCE AILU THICK				
✓ ▲ Primary ✓ Frame	Path Tolerance	0.01			
S Features	Axial Thick	0			
Tool and SpeedFeed	Radial Thick	0			
 Limiting Interance and Steps 	▼ Cutting Steps				
> 襘 Path Setting	Step Value	0.15			
 Link and Lead Lead In 	Number of Cuts	4			
📤 Lead Out					
🖶 Display					
Reset	Calculate	ОК	Cancel		
🦁 🗾 🖞	Π 💱 Ι		6		

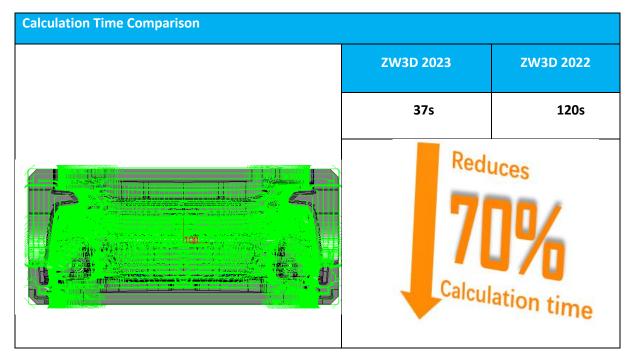




5.11 Kernel Algorithm Optimization

5.11.1 ★Speed Up QM Rough Operation Calculation

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



5.12 New Functionalities of Milling

5.12.1 Distinguish Hole Milling and Thread Milling

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

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

File	Setup Drill 2x Mill 3x Quick	5x Mill Turning	Tool Path Editor	Output Tools Vis	ualize Inquire	
U U		- E	😵 🞦	V_ +		þ
Mill2	Spiral Zigzag Box Contour VoluMi	2x Profile Ramp	Surface Nesting Engraving	Chamfer Corner Round	Helical Thread Opface	Inter Path Move
Tactics	2D Pocket	2	D Contour	Corner cut	2D Face	Directive



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

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

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

5.12.2 ★Support Taper Hole and Taper Boss Milling

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

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

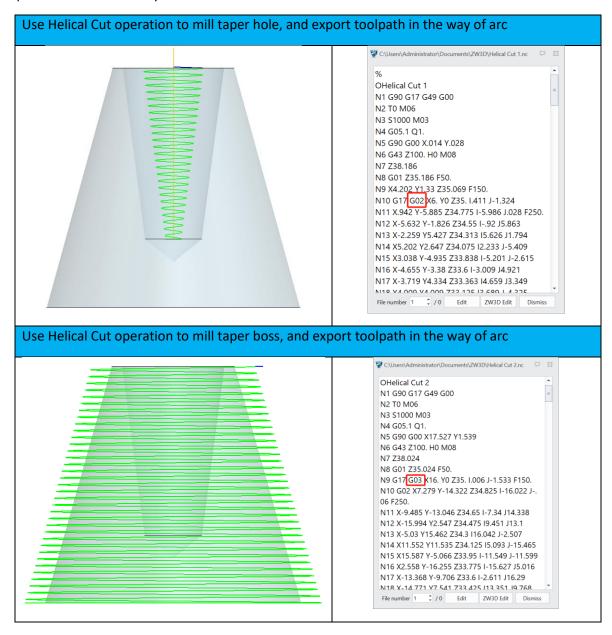


🖗 Hole Feature			
Name hole	1		
Type gene	eral		
Class part			
	Part002 (1)		
File			
4 040.00	ributes	î	
2. D10.00		- "@	
3. D10.00 Type Size			
	meter 10		
Dep	oth 10		
Таре	er Angle 5		
	erance 0.01		
Finis			
	nk Diameter nk Angle		
	ore Diameter		
	ore Depth		
Thre	d Diameter		
	d Depth		
	d Pitch 0		
	Hole Axis Start Point		
Create features for even			
	Apply Attributes		•
Add Holes	Rem	ove Holes	
	OK Cancel		
d "Taper A		ss Fosturo	
	Angle to Bo	JSSTEature	
•	angle to Bo	Jas reactive	
🖗 Boss Feature	angle to Bo		2
	cylboss 1		3
🖗 Boss Feature			
😨 Boss Feature Name	cylboss 1		
 Boss Feature Name Type 	cylboss 1 general		
Boss Feature Name Type Class	cylboss 1 general Part		
 Boss Feature Name Type Class Component File Bosses 	cylboss 1 general Part Part002		
Boss Feature Name Type Class Component File Bosses H0	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter	R.250227	
Boss Feature Name Type Class Component File Bosses	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height	□ Σ	
Boss Feature Name Type Class Component File Bosses H0 H1	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle	□ 5 8.250227 10 5.000000	
Boss Feature Name Type Class Component File Bosses H0 H1	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle Tolerance	 □ 5 8.250227 10 5.000000 0.01 	
Boss Feature Name Type Class Component File Bosses H0 H1	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle Tolerance Finish	₽ 5 8.250227 10 5.000000 0.01	
Boss Feature Name Type Class Component File Bosses H0 H1	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle Tolerance Finish Thrd Diameter	 □ ■ 8.250227 10 5.00000 0.01 0.01 0.01 0 	
Boss Feature Name Type Class Component File Bosses H0 H1	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle Tolerance Finish Thrd Diameter Thrd Height	 □ ■ 8.250227 10 5.00000 0.01 0.01 0 0 0 	
Boss Feature Name Type Class Component File Bosses H0 H1	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle Tolerance Finish Thrd Diameter	 □ ■ 8.250227 10 5.00000 0.01 0.01 0.01 0 	
Boss Feature Name Type Class Component File Bosses H0 H1	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle Tolerance Finish Thrd Diameter Thrd Height	 □ ■ 8.250227 10 5.00000 0.01 0.01 0 0 0 	
Boss Feature Name Type Class Component File Bosses H0 H1	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle Tolerance Finish Thrd Diameter Thrd Height Thrd Pitch	 □ 5 8.250227 10 5.000000 0.01 0.01	
Boss Feature Name Type Class Component File Bosses H0 H1	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle Tolerance Finish Thrd Diameter Thrd Diameter Thrd Height Thrd Pitch Boss Axis Apply Attribu	 □ 5 8.250227 10 5.000000 0.01 0.01	
Boss Feature Name Type Class Component File Bosses H0 H1 H2	cylboss 1 general Part Part002 Part002.Z3 Attributes Diameter Height Taper Angle Tolerance Finish Thrd Diameter Thrd Height Thrd Pitch Boss Axis Apply Attribu	■ 2 8.250227 10 5.00000 0.01 0.01 0 0 0 0 0 0 0 0 0 0 0 0 0	





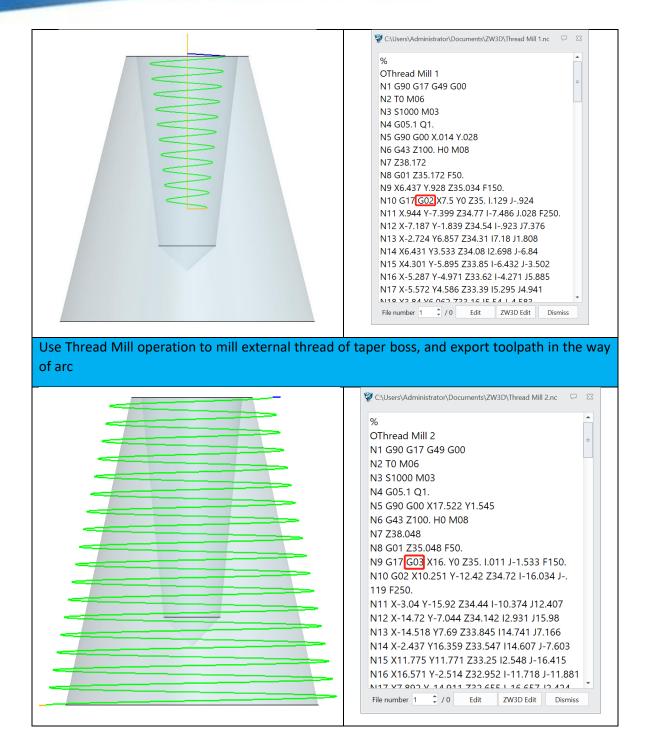
Use Helical Cut operation to generate the milling toolpath of taper hole and taper boss, respectively, and export NC code in the way of arc.



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

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





5.12.3 ★ 2D Pocket Operation Supports External Lead-in Open Pocket

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

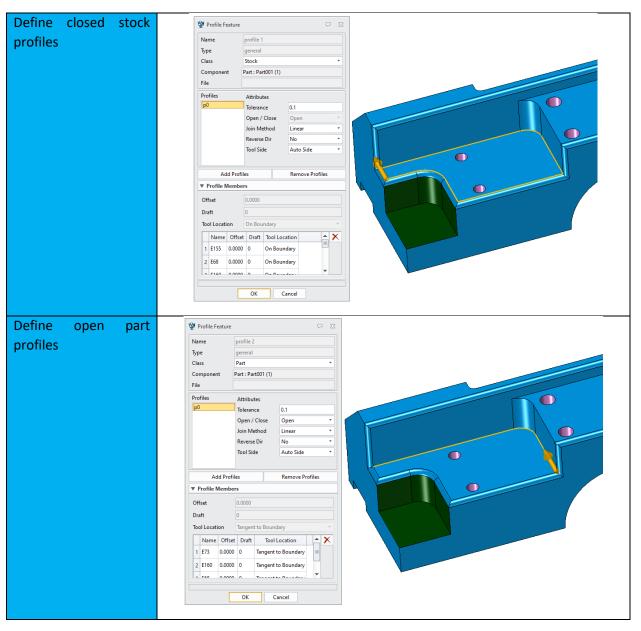




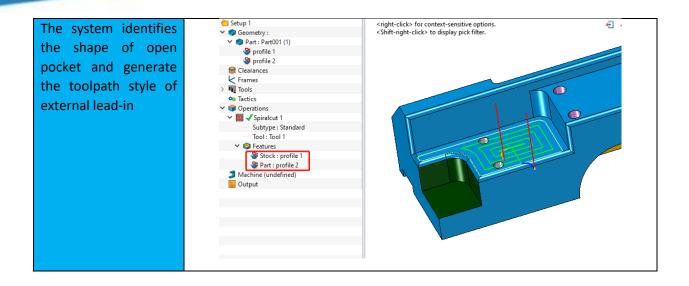
find out the part boundary (profile of stock type) that can be crossed and then start to lead in on the boundary, achieving the external lead-in of open pocket. When define stock profile, the selected profile must be closed; when define part profile, the selected profile can be open or closed. Processing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

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

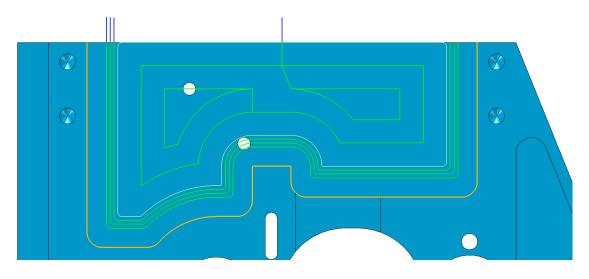
"Part" type profile——Tool cannot touch the profile







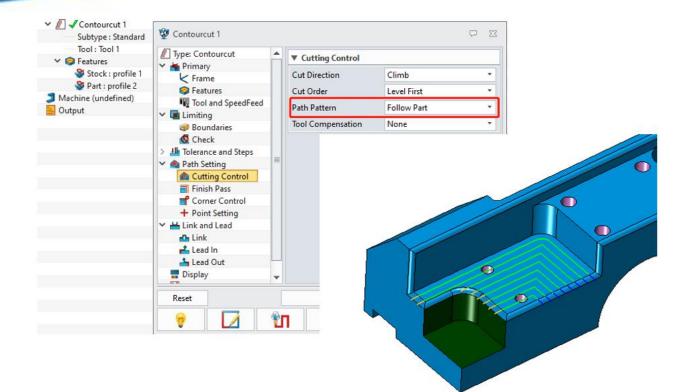
In the new version, when 2D pocket operation defines both "Stock" and "Part" profile features, the cleanup toolpath of operation can only be generated on the boundary of "Part" type. As the following figure shows, the Multiple levels cleanup can only be generated on the profile boundary of "Part" type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



5.12.4 **★**New Follow Part Path Pattern in 2X Contourcut operation

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





5.13 New Functionalities in Turning Model

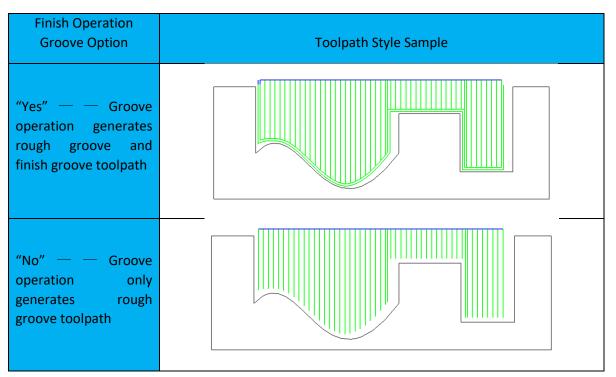
5.13.1 Finish Operation in Groove Operation Optimization

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

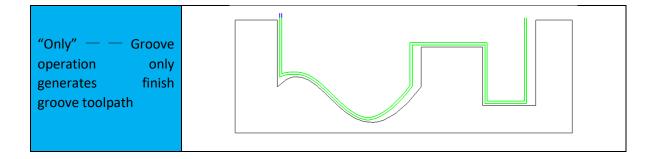
We added "Only" to the finish groove operation, which provides more processing strategy.

🚔 Type: Turn Groove	▼ Rough Cut	
 Frame 	Cut Regions	OD 🔹
Frame	Cut Direction	Right to Left 🔹
Tool and SpeedFeed	Cut Strategy	Zigzag *
 Limiting Interance and Steps 	Relief Amount	0.2
 Path Setting 	Stock Height	0
Rough Cut	Dwell Time (s)	0.
+ Point Setting	Rough Clearance	1
Link and Lead	Safe Distance	2
📥 Lead In 🚍 Display	Output Type	Custom 🔻
	▼ Finish Cut	
	Finish Groove	Yes 🔹
	Cut Direction	Yes No
	Finish Thick	Only
	Finish Number	3
	Finish Distance	2
	Retract Position	0
Reset	Calculate	OK Cancel

The available options of finish operation groove include the following:



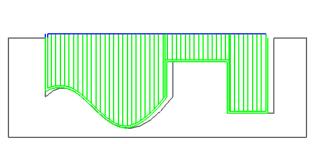




We added the parameters "Cut Direction" "Finish Number" and "Finish Distance" to the finish groove, which can be used to control the style of finish groove toolpath.

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

💯 Turn Groove 1		₽ X
Handreich Type: Turn Groove	▼ Rough Cut	^
Primary Frame	Cut Regions	OD -
S Features	Cut Direction	Right to Left 🔹
Tool and SpeedFeed	Cut Strategy	Zigzag 🔹
> 🏭 Tolerance and Steps	Relief Amount	0.2
> 🕋 Path Setting	Stock Height	0
 Link and Lead Lead In 	Dwell Time (s)	0. 📃
冒 Display	Rough Clearance	1
	Safe Distance	2
	Output Type	Custom 🔻
	▼ Finish Cut	
	Finish Groove	Yes 🔹
	Cut Direction	Left to Right 🔹 🗆
	Finish Thick	0
	Finish Number	2
	Finish Distance	1
	Retract Position	0
Reset	Calculate	OK Cancel
💡 🔽 👣	V	

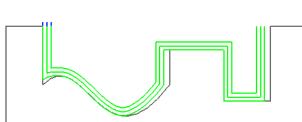


Finish Number——Use to set the finish groove number

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



📥 Type: Turn Groove	▼ Rough Cut			
 Mary Frame 	Cut Regions	OD	-	
Features	Cut Direction	Right to Left	-	
Tool and SpeedFeed	Cut Strategy	Zigzag	-	
> 🏭 Tolerance and Steps	Relief Amount	0.2		
> 🔷 Path Setting	Stock Height	0		
 Link and Lead Lead In 	Dwell Time (s)	0.		
Tisplay	Rough Clearance	1		
	Safe Distance	2		
	Output Type	Custom	-	
	▼ Finish Cut			
	Finish Groove	Only	•	
	Cut Direction	Left to Right	•	
	Finish Thick	0		
	Finish Number	3		
	Finish Distance	2		
	Retract Position	0	•	
Reset	Calculate	OK Ca	ncel	



5.14 Workflow and Usability Optimization

5.14.1 **★**Profile Feature Curve Attribute Optimization

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

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

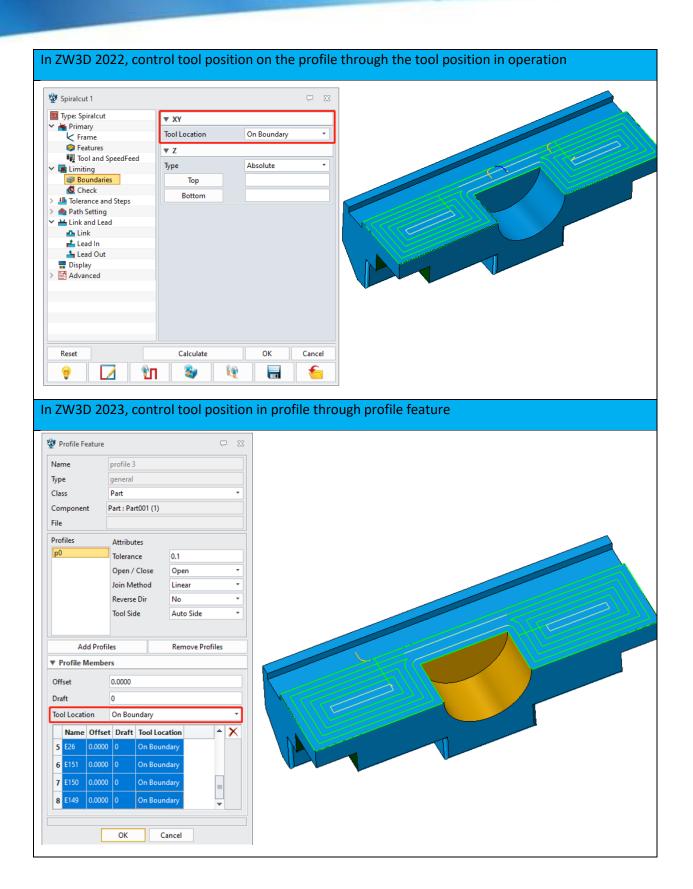
The tool position options include on boundary, tangent to boundary, past boundary.

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



Profile Featu	ire	₽ 33	24	P	rofile Fe	eature				Ģ	Σ
lame	profile 1			Nar	ne		profile 3				
ype	general			Тур	e		general				
lass	Part			Clas			Part				•
Component	Part001				nponen	t F	eatures				
ile	Part001.Z3PRT			File							
rofiles	Attributes				files		Attribut	es			
p0 Tolerance		0.1		p0			Toleran		0.1		
							Open /		Open		*
	Offset	0					Join Me		Linear		*
	Open / Close	Open 🔻					Reverse		No		•
	Join Method	Linear 🔻					Tool Sid	e	Auto Side		-
	Reverse Dir	No *									
	Tool Side	Auto Side 🔻			Ad	d Profil	es		Remove Pro	files	
Modify	Attributes	Apply Attributes		V P	rofile N	lember	s				
Add F	Profiles	Remove Profiles		Off	set		0.0000				
	ок	Cancel		Dra	ft		0				
				Тоо	l Locati	on	Tangent	to Bour	ndary		Ŧ
🥸 Cun	ve Attribute Modify	₩ 23			Name	Offset	Draft	Тоо	I Location		X
Tolera	nce 0.1			1	E15	0.0000	0	Tangent	t to Boundary		
				2	E13	0.0000	0	Tangent	t to Boundary		
Offset	-			3	E152	0.0000	0	-	t to Boundary		
Draft	0.0			\vdash	E62	0.0000			t to Boundary		
Tool Lo	ocation Tanger	nt to Boundary 🔻		4	202	0.0000	U	langen	t to boundary	•	

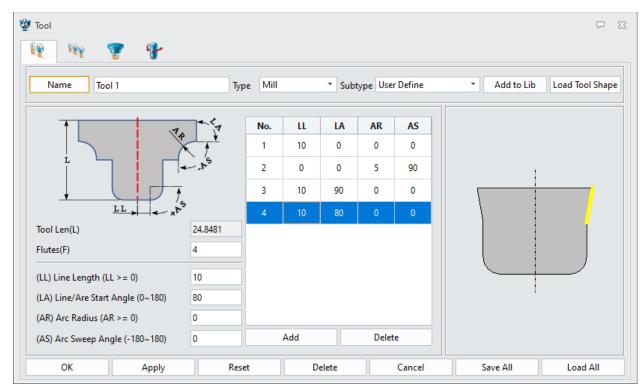






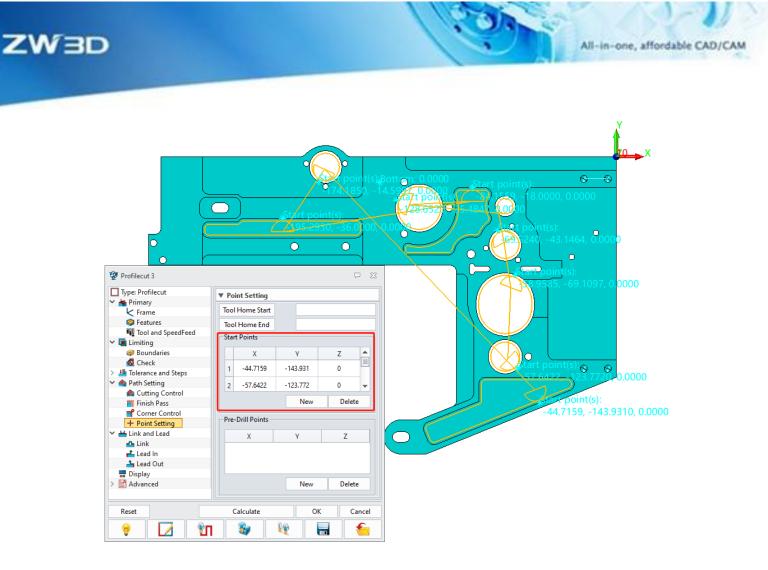
5.14.2 **★**Update User Customized Tool Functionality

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



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

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



5.14.4 CL New Export Variables in Files

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

C:\Users\ZW3D\Desktop\11111.cl	C 23	2 C:\Users\ZW3D\Desktop\11111.cl	C 23
PPRINT / Setup Class - ' PPRINT / Setup Type - ' PPRINT / Frame Name - ' PPRINT / Frame Name - ' PPRINT / Frame Approach - 7.5000000' PPRINT / Frame Approach - 7.5000000' PPRINT / Frame Approach - 7.500000' PPRINT / Voperation Toires register -' PPRINT / Operation Toires register -' PPRINT / Operation Toires register -' PPRINT / Operation Toires - 0.1000' OPER_ZMAX /11.00111 OPER_SUBTYPE / roffset2dcut HOLENUM /0 PPRINT / 'STEP 6.0000' PPRINT / 'STEP 6.0000' PPRINT / 'STEP 6.0000' PRINT / 'STEP 6.0000' PRINT / 'STEP 6.0000' PRINT / 'STEP 6.0000' PRINT / STEP 6.0000' PRINT / STEP 6.0000' PRINT / STEP 6.0000' PRINT / STEP 6.0000' COPER_ZMAX /11.0011	•	SS CAM-ID: ZW3D V15.1 PARTNO / 'NC' MACHIN/ZWPost,ZW,Fanuc_3X PPRINT / 'Programmed by ZW3D' PPRINT / 'Programmed by ZW3D' PPRINT / 'Total Time - 2 HOURST1 MINUTES:47 SECONDS' TMARK/1 SEQNO/0,INCR,1 FROM/0.00000, 0.0000, 500.0000 STOCK/-89.0000,-83.0000,-13.00000,56.00000,39.50000,12.00000 SS End startup sequence. SS Begin first tool sequence. PPRINT / 'Tool 0 is JPL 1: PPRINT / 'Holdre name is; PPRINT / 'Holdre name is; PPRINT / 'Holdre name is; PPRINT / 'Holdre name is; PPRINT / 'Tool 1: JPL 1: PPRINT / 'SHAPE PARAMETERS' CUTTER/S0.0000 PPRINT / TOOL Dia: 10, Tip Rad: 0, Len: 50.00000 PPRINT / 'DOL Dia: 10, Tip Rad: 0, Len: 50.00000 PPRINT / 'DOL Dia: 10, Tip Rad: 0, Len: 50.00000	• III
RAPID	A second s		



5.14.5 Support Open Operation List in WPS

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

5.14.6 STL Rest Stock Creation Optimization

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

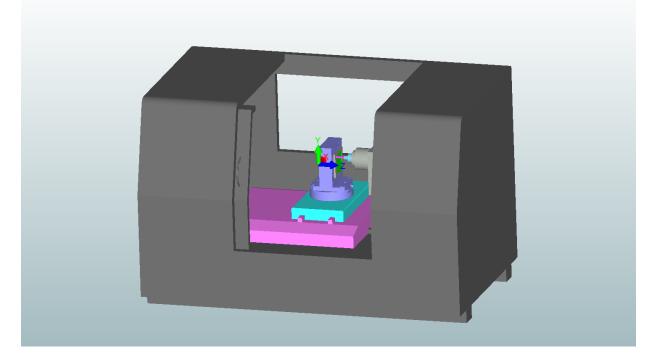
🔮 Solid Verify Session 🖓	23
Verify Tpath Show Tpath data	
Display operation list	
05. Zlevel 1, Tool 1	 Image: Simulation Options Imag
	-Save Solid STL
	Path C:\Users\ZW3D\Documents\ZW3D
Components	Name Rest_Stock_Part001_CAM .stl
Stock Part001_Stock.1	File Format O ASCII
Target Part Part001	STL File Unit 💿 mm 🔘 inch
Attachment	
+1 📕 🕅	
Motion Delay Update Interval	
☑ Optimized ☑ Clash stop	
Verify Tpath Verify File Data	
Skip suppressed operations	Use the rest as stock
Verify active operation only	Apply Cancel Close
Tool Move O Stock Move	
Options Close	

5.15 FMS

5.15.1 New Horizontal Machine

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





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

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

5.16Others

5.16.1New Milling and Turning CAM Plan Template

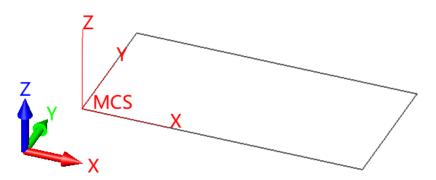
In ZW3D 2023, we added two CAM templates: Mill Template and Turn Template in the entry of CAM Plan.

ZW3D



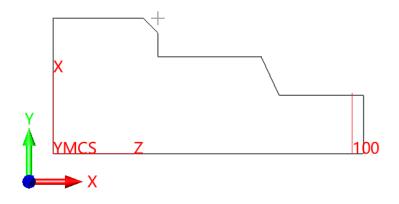
💱 Select a templa	te	Ģ	23
[Default]			
MillTemplate			
TurnTemplate			
ОК	Canc	el	

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



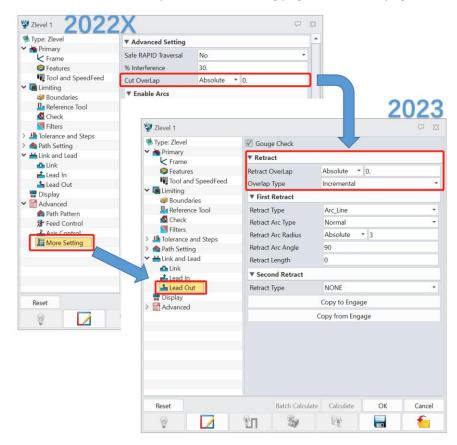
MCS orientation in Turn Template are not identical with the word coordinate system. Z axis of MCS is the same with X-axis orientation of the world coordinate system, and X-axis with Y-axis of the world coordinate system, which are convenient to user for turning programming.





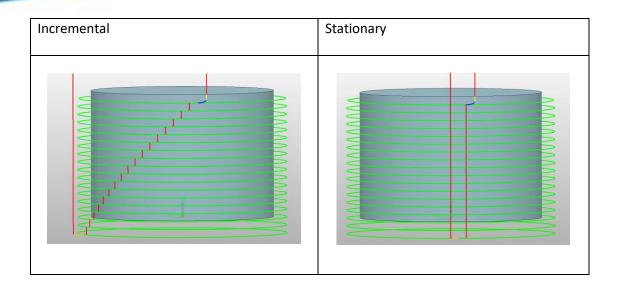
5.16.2New Overlap Style in ZLevel Operation

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



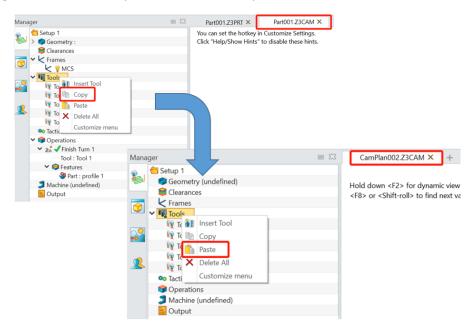
• Newly add stationary style in Overlap:





5.16.3New Tool Copy/Paste

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



5.16.4Enable Auto Write ORIGIN in Output by Default

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



ZW3D

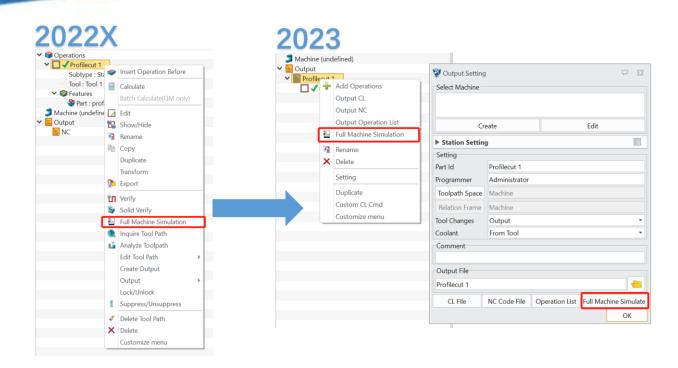
configuration is enabled by default and the option "Write ORIGIN in output" in the new frame is checked by default.

🦉 Frame 🖓 🖾	🦉 Configuration		\Box	23
Name MCS	General Part	CAM Settings		
Clear Z 100	2D Color	Move to/from Setup clear Z □ Auto add all features ✓ Write speeds as integers □ Auto add new features Enable 3 Axis Direct Nurbs		
Head none	Background Display Files	Clear Z 100 Image: Clear Z CAM update mode 1 Image: Clear Z NC extension name .nc Image: Clear Z		
	CAM	Output filename conflicts None Image: Conflict conflic	<u></u>	
Turn Mode Work Plane Z-X Plane Fixture Offset Offset Register Auto Write ORIGIN in Output Define Frame Datum Create Datum Select Datum Frame Attribute	User PDM ECAD Routing	CAM operation library CAM machine library CAM output folder C\Users\Administrator\Documents\ZW3D		
Color Reset OK Cancel	Reset Default	OK Cancel	Apply	- y

5.16.5Change FMS Start Position

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





5.16.6Remove Some Functions from Machine Manager and Output Program Setting

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

ZW3D

🖗 Machine Manager							⊽ ∑
Definition				Library			
Machine Name				Machine			<u></u>
Class	3-Axis M	1.C.	-				
🔲 Туре	Vertical		-				
Subtype	Rotating	Head	•				
Post-Processor	ZWPost		•				
Post Configuration	Anilam_0	Crusader					
XY Arcs	Yes		-				
YZ Arcs	No		-				
ZX Arcs	No		-				
Spacial Arc	No		*				
Helical Arc	No		-				
Check MULTAX	Yes		*				
MULTAX	No		•				
Accurate RAPIDs	No		-				
Scale	1						
#.xxxx	5						
Rewind	Yes		-				
Increment	1						
CUTCOM	None		-				
Offset Registers					De	elete	
NC Extension	.nc			Options	ol Changer	Rota	ry Axes and Offsets
Definition File	machine	_all.mdf		-	rameters	NOT	Limits
Open Machine Defini	tion File	Legacy Definit	tion Files 🗙		To Library>		Apply Filter>
ОК		Apply	Ne	ew.	Reset		Cancel

Part Id	NC		
Programmer	Administrator		
Coolant	From Tool		Ŧ
Toolpath Space	Machine		
Relation Frame	Machine		
Tool Changes	Output		
Tool Num	Tool Id	X	•
Speeds/Feeds	Output		•
Comment			
Folder Name		4	-
Display Output			
		OK	