ZW3D WHAT'S NEW

V 2025 SP

ZWSOFT CO., LTD.(Guangzhou)

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ZW3D™ V2025 SP What's New

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Highlights of ZW3D 2025

ZW3D 2025
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To speed up product design and increase design efficiency, we have made great improvements in efficiency in ZW3D 2025. This version will help you complete all aspects of product design, simulation and verification, and production and manufacturing more efficiently, which can save time and resources.

- ✓ **Faster and more efficient design experience.** In ZW3D 2025, you can accelerate product design by the aid of feature efficiency and process optimization in engineering drawing, part and assembly design and CAM machining design. For instance, the considerable efficiency improvement in engineering drawing projection, the smooth design supporting 10,000-level assemblies, and the fast CAM machining toolpath simulation.
- ✓ **Cover more design processes.** Kinematics and low frequency electromagnetic simulation are added to ZW3D 2025. The kinematical simulation can help you evaluate the accuracy and rationality of product motion mechanism in the early stage of product design, and the low frequency electromagnetic simulation can help you achieve simulation and optimization analysis of electromagnetic field distribution, performance parameters of multi-type motors, transformers, actuators, sensors, wireless charging and other mechanical and electrical equipment, which decreases design costs and avoids repetition of late design changes. Besides, the enhanced functionalities of harness design, including harness branching and merging and nailboard, can help you carry out the whole process from harness design to manufacture.

Key Improvements

Basic:	★New File Switch Permission
	★New Reload File
	★Material Library
	★New Series Value Property
	★Commands Hotkey Globally Related/Individual Module Settings
	★Model Match Function
Translator:	★Input Format Upgrade
	★Batch Export Function Optimization
Sketch & Wireframe:	★Sketch Over-defined Process Improvement
Shape Design:	★New "To Next" to Extrude Function
Assembly Design:	★Large Assembly Efficiency Improvement
	★One Click Regeneration & Auto Regeneration
	★Link Manager
	★Assembly Constraints
	★Assembly Animation Reconstruction
	★Assembly Component Quick Insertion

Drawing Sheet Design:	★New Discrete Projection
	★Dimension Attribute Enhancement
	★Easy-to-use Multiple Arrow (Copy and Attached Object Correctness)
	★BOM & Balloon Improvement
	★Feature Control
	★Drawing Sheet Block
Structure & Weldment Design:	★Unify Structural Member Direction and Auto-Creating Joint Optimization
	★New Batch Editing Structure Function
	★New Batch Create Joint Function
	★Merge Rule Improvement
	★Support Template Import & Export and Online Editing
	★Structure Wizard Function
	★UI/UE of Structure Member Module Entire Upgrade
Mold Design:	★New Projection Area Function
Piping & Tubing Design:	★Insert Routing Part Optimization
	★Spool Drawing Optimization
	★ISO Setting Optimization

Harness Design:	★Harness Creation Enhancement
	★Adapt Harness for Dimension Measurement
	★New Global Wiring Table
	★New Harness Nailboard
CAM:	★Overall Innovation of Solidverify
API:	★API System Upgrade and Interface Expansion
Simulation:	★Kinematics Simulation
	★Low Frequency Electromagnetic Simulation

1 Basic

1.1 File Management

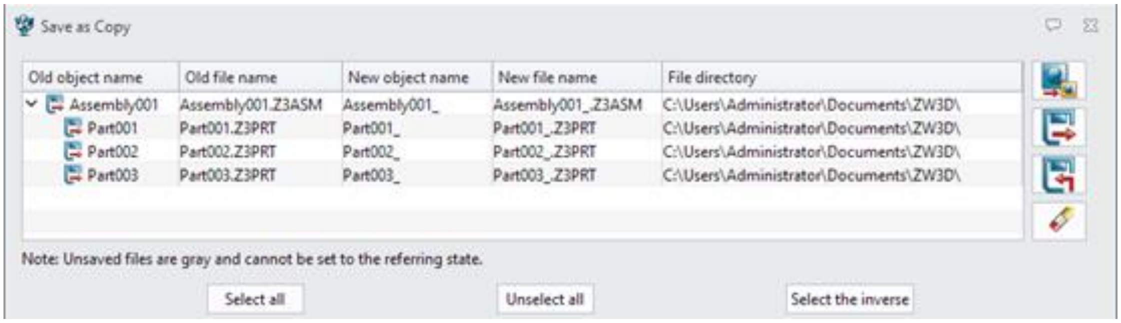
The optimized "**Pack**" content of file management, the new "**File Switch Permission**" and "**Reload**" content can not only strengthen the packaging ability, but also better realize the free switching permissions and reloading of the design model in the collaborative design process. Please check the following chapters and the help documentation for other improvements and detailed descriptions.

1.1.1 New Custom Save Path for Different Files in Save as Copy

Due to the specific file management requirements, some users need to define different save paths for different files under the assembly. Therefore, the function of custom save path for different files has been added to the replica, which supports custom save path for different files assembled to meet the above requirements.

【What can be done】:

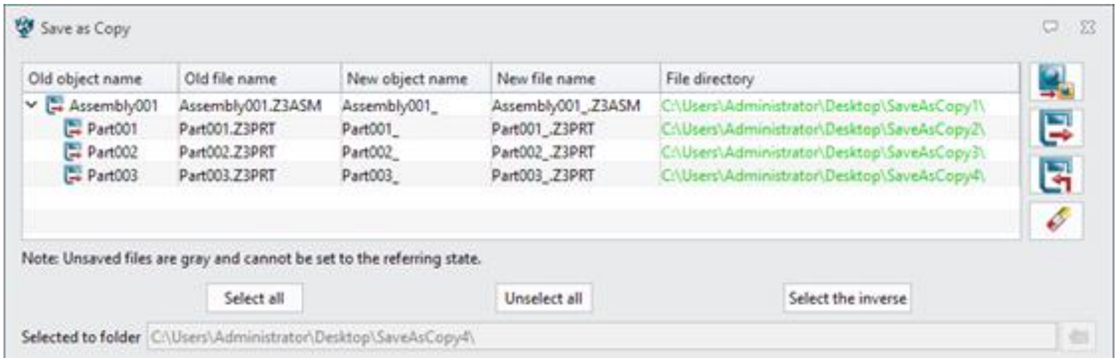
- ✓ Separately define save as copy path for different files under assembly.
- ✓ Highlight the modified save as copy path.
- ✓ Maintain the association between a file saved in a customized path and other files.



Old object name	Old file name	New object name	New file name	File directory
Assembly001	Assembly001.Z3ASM	Assembly001_	Assembly001_.Z3ASM	C:\Users\Administrator\Documents\ZW3D\
Part001	Part001.Z3PRT	Part001_	Part001_.Z3PRT	C:\Users\Administrator\Documents\ZW3D\
Part002	Part002.Z3PRT	Part002_	Part002_.Z3PRT	C:\Users\Administrator\Documents\ZW3D\
Part003	Part003.Z3PRT	Part003_	Part003_.Z3PRT	C:\Users\Administrator\Documents\ZW3D\

Note: Unsaved files are gray and cannot be set to the referring state.

ZW3D 2024



Old object name	Old file name	New object name	New file name	File directory
Assembly001	Assembly001.Z3ASM	Assembly001_	Assembly001_.Z3ASM	C:\Users\Administrator\Desktop\SaveAsCopy1\
Part001	Part001.Z3PRT	Part001_	Part001_.Z3PRT	C:\Users\Administrator\Desktop\SaveAsCopy2\
Part002	Part002.Z3PRT	Part002_	Part002_.Z3PRT	C:\Users\Administrator\Desktop\SaveAsCopy3\
Part003	Part003.Z3PRT	Part003_	Part003_.Z3PRT	C:\Users\Administrator\Desktop\SaveAsCopy4\

Note: Unsaved files are gray and cannot be set to the referring state.

Selected to folder: C:\Users\Administrator\Desktop\SaveAsCopy4\

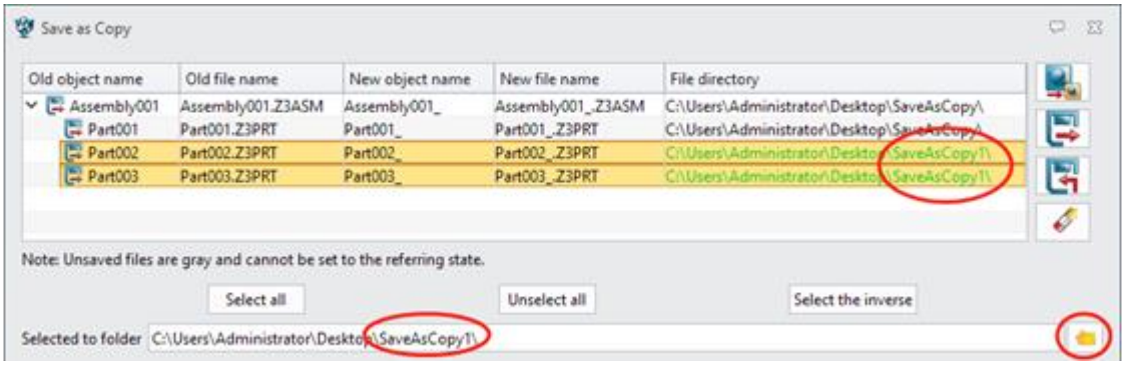
ZW3D 2025

【Note】:

- If you fill in a path that does not exist, a new folder will be created after you click save or save and open.
- If the folder cannot be created because of the invalid path, the file in the invalid path will not be saved as a copy. In this case, when open the assembly file of which the copy is successfully saved, it will reference the file in the original path.

【Example】:

As shown in the following figure, the assembly file contains 4 parts. After selecting the target part, click the folder icon in "**Save as Copy**" to select the file saving path, or directly modify the path text and save.



Old object name	Old file name	New object name	New file name	File directory
Assembly001	Assembly001.Z3ASM	Assembly001_	Assembly001_.Z3ASM	C:\Users\Administrator\Desktop\SaveAsCopy\
Part001	Part001.Z3PRT	Part001_	Part001_.Z3PRT	C:\Users\Administrator\Desktop\SaveAsCopy\
Part002	Part002.Z3PRT	Part002_	Part002_.Z3PRT	C:\Users\Administrator\Desktop\SaveAsCopy1\
Part003	Part003.Z3PRT	Part003_	Part003_.Z3PRT	C:\Users\Administrator\Desktop\SaveAsCopy1\

Note: Unsaved files are gray and cannot be set to the referring state.

Selected to folder: C:\Users\Administrator\Desktop\SaveAsCopy1\

【Where is it】:

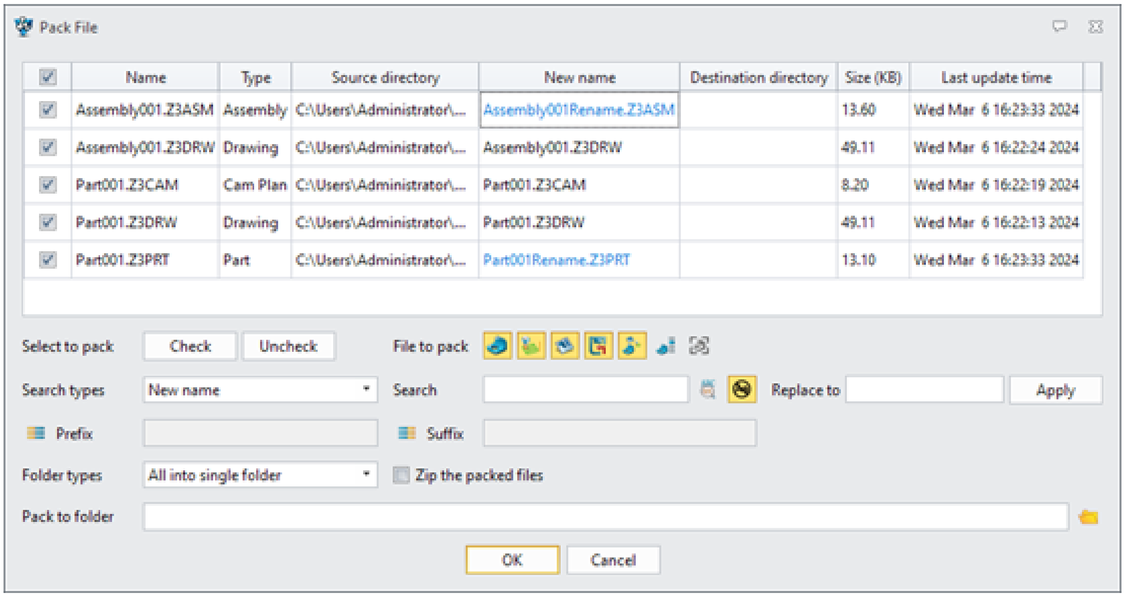
File >> Save >> Save as Copy

1.1.2 New Drawing Sheet/CAM Plan Renamed with Original File in Pack File

If users need to rename a file when using the File Pack function, the engineering drawing and CAM plan are required to be consistent with the corresponding part or assembly in most cases. Therefore, we allow the engineering drawing/CAM plan to be renamed with the original file in Pack File. After users rename parts or assemblies, the corresponding engineering drawing or CAM plan will be automatically renamed. Users are free from modifying names again, which significantly improves the efficiency of pack renaming.

【What can be done】:

- ✓ In the Pack File function, check "Rename Sheet/CAM as Part/Assembly file" to rename the parts or assembly file, and the corresponding engineering drawing and CAM name will be automatically renamed.



Pack File

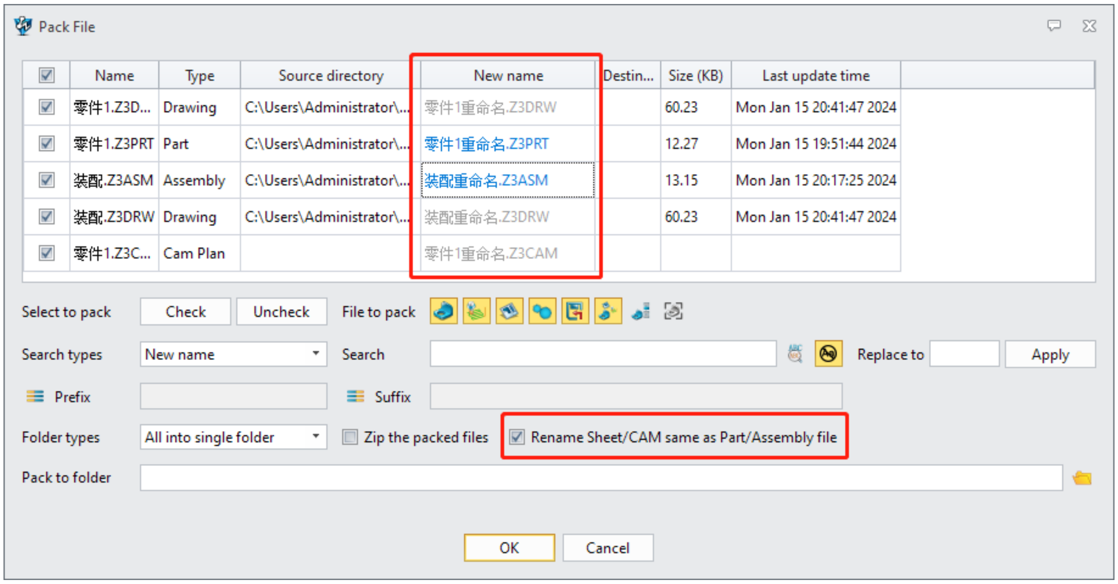
☑	Name	Type	Source directory	New name	Destination directory	Size (KB)	Last update time
☑	Assembly001.Z3ASM	Assembly	C:\Users\Administrator\...	Assembly001Rename.Z3ASM		13.60	Wed Mar 6 16:23:33 2024
☑	Assembly001.Z3DRW	Drawing	C:\Users\Administrator\...	Assembly001.Z3DRW		49.11	Wed Mar 6 16:22:24 2024
☑	Part001.Z3CAM	Cam Plan	C:\Users\Administrator\...	Part001.Z3CAM		8.20	Wed Mar 6 16:22:19 2024
☑	Part001.Z3DRW	Drawing	C:\Users\Administrator\...	Part001.Z3DRW		49.11	Wed Mar 6 16:22:13 2024
☑	Part001.Z3PRT	Part	C:\Users\Administrator\...	Part001Rename.Z3PRT		13.10	Wed Mar 6 16:23:33 2024

Search types: New name

Folder types: All into single folder

☐ Zip the packed files

ZW3D 2024



Pack File

☑	Name	Type	Source directory	New name	Destin...	Size (KB)	Last update time
☑	零件1.Z3D...	Drawing	C:\Users\Administrator\...	零件1重命名.Z3DRW		60.23	Mon Jan 15 20:41:47 2024
☑	零件1.Z3PRT	Part	C:\Users\Administrator\...	零件1重命名.Z3PRT		12.27	Mon Jan 15 19:51:44 2024
☑	装配.Z3ASM	Assembly	C:\Users\Administrator\...	装配重命名.Z3ASM		13.15	Mon Jan 15 20:17:25 2024
☑	装配.Z3DRW	Drawing	C:\Users\Administrator\...	装配重命名.Z3DRW		60.23	Mon Jan 15 20:41:47 2024
☑	零件1.Z3C...	Cam Plan		零件1重命名.Z3CAM			

Select to pack: Check | Uncheck | File to pack

Search types: New name | Search | Replace to | Apply

Prefix | Suffix

Folder types: All into single folder | ☐ Zip the packed files | ☑ Rename Sheet/CAM same as Part/Assembly file

Pack to folder

OK | Cancel

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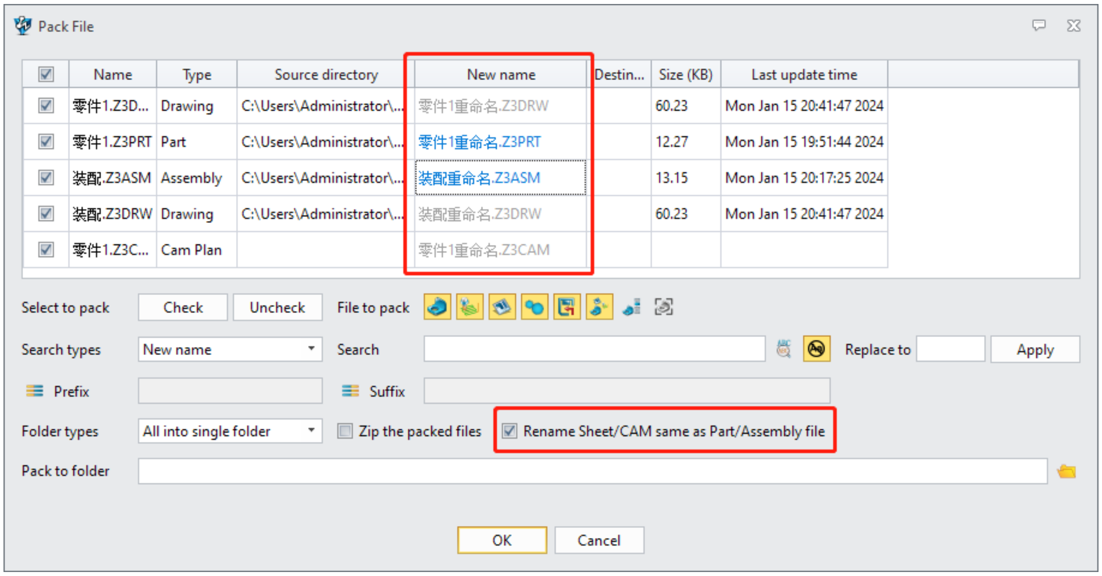
【Example】:

As shown in the figure below, "Rename Sheet/CAM as Part/Assembly file" is checked by default. When renaming "Assembly.Z3ASM" as "Assembly rename.Z3ASM", its corresponding "Engineering Drawing Assembly.Z3DRW" is automatically renamed as "Assembly rename.Z3DRW" with the original file. As "Part 1.Z3PRT" is renamed as "Part 1 rename.Z3PRT", its corresponding engineering drawing and CAM plan are also automatically renamed.



Pack File

☑	Name	Type	Source directory	New name	Destin...	Size (KB)	Last update time
☑	零件1.Z3D...	Drawing	C:\Users\Administrator\...	零件1重命名.Z3DRW		60.23	Mon Jan 15 20:41:47 2024
☑	零件1.Z3PRT	Part	C:\Users\Administrator\...	零件1重命名.Z3PRT		12.27	Mon Jan 15 19:51:44 2024
☑	装配.Z3ASM	Assembly	C:\Users\Administrator\...	装配重命名.Z3ASM		13.15	Mon Jan 15 20:17:25 2024
☑	装配.Z3DRW	Drawing	C:\Users\Administrator\...	装配重命名.Z3DRW		60.23	Mon Jan 15 20:41:47 2024
☑	零件1.Z3C...	Cam Plan		零件1重命名.Z3CAM			

Select to pack: Check | Uncheck | File to pack

Search types: New name | Search | Replace to | Apply

Prefix | Suffix

Folder types: All into single folder | ☐ Zip the packed files | ☑ Rename Sheet/CAM same as Part/Assembly file

Pack to folder

OK | Cancel

【Where is it】:

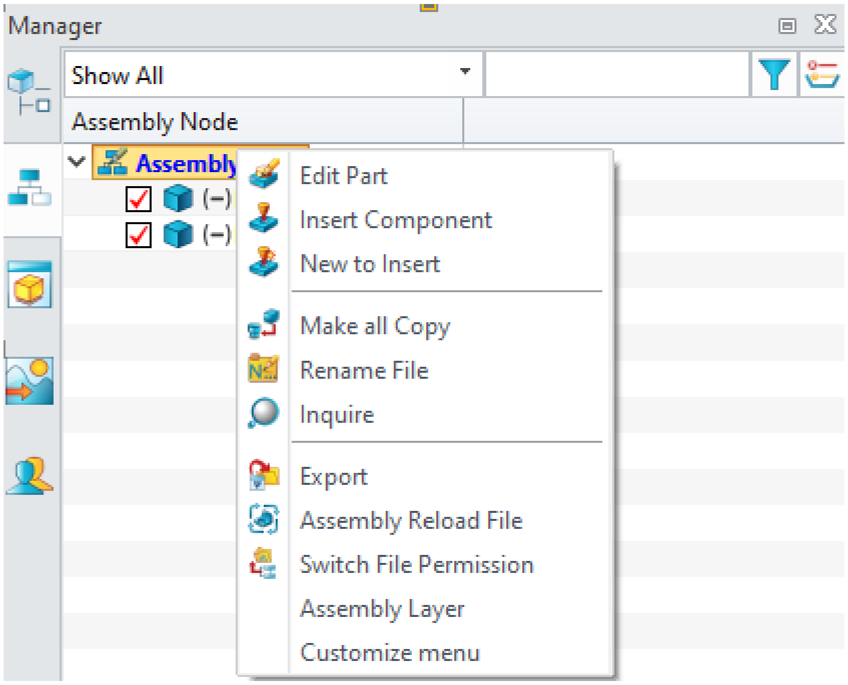
File >> Pack

1.1.3 ★New File Switch Permission

In the process of **collaborative design**, the new "**Switch File Permission**" is used to change the permission status of the current file, such as shifting the file permission from write to read-only or vice versa. Without closing the model, you can change the file permission, achieving the purpose of quickly changing the write permission.

【What can be done】:

- ✓ File permission changed from write to read-only.
- ✓ File permission changed from read-only to write.


Manager
Show All
Assembly Node
Assembly
Edit Part
Insert Component
New to Insert
Make all Copy
Rename File
Inquire
Export
Assembly Reload File
Switch File Permission
Assembly Layer
Customize menu

【Notes】:

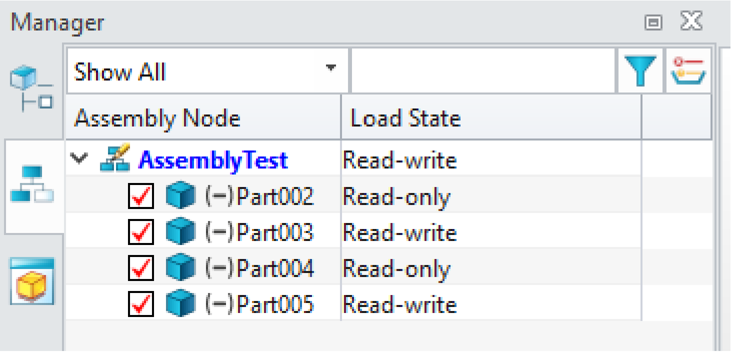
- You can only switch between write and read-only. The file cannot be switched to write if the system restricts it to read-only.
- You can use the "Load State" of the "Attributes Manager" to view the permissions of each

component.

- Virtual components are determined by the permissions of the assembly they belong to.

【Example】：

1. As multiple accounts open the same assembly model of the shared network at the same time, the account with the “write” permission right-clicks the target component and runs the “**Switch File Permission**” command to change the target component to “read-only”. The account with the “read-only” permission right-clicks the component and runs the “**Switch File Permission**” command to change the target component as “write”. You can exchange permissions without closing or re-opening the model, so that other accounts can continue to edit and modify the model.



Assembly Node	Load State
AssemblyTest	Read-write
☑ (−)Part002	Read-only
☑ (−)Part003	Read-write
☑ (−)Part004	Read-only
☑ (−)Part005	Read-write

【Where is it】：

Part/Assembly Manager >> File >> Switch File Permission

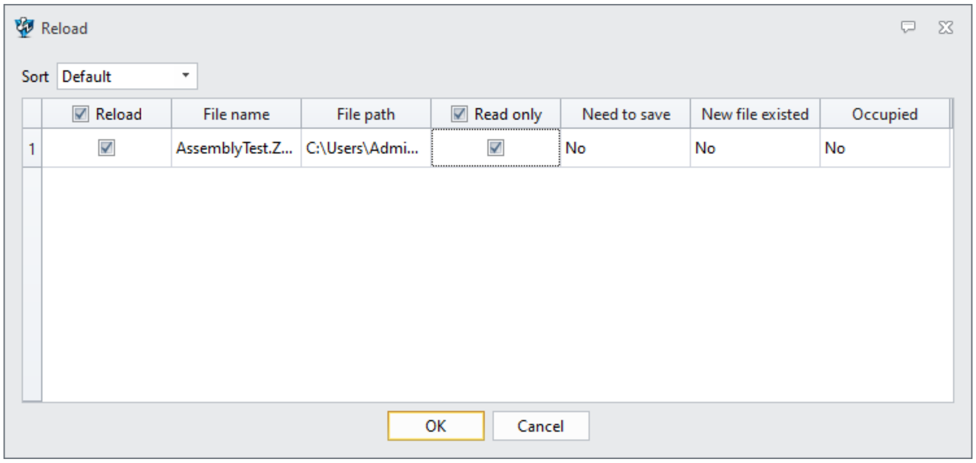
1.1.4 ★New Reload File

In the process of **collaborative design**, the new "Reload File" is used to reload model information, such as switching file permissions. Without closing the model, you can reload the latest state of the file to re-read the model.

【What can be done】：

- ✓ Switch file permissions and check whether the file is occupied.
- ✓ Check whether the file has a newer version.

- ✓ Determine whether the model is lost due to reloading and needed to be saved.
- ✓ Query the model location.



Sort: Default

	☑ Reload	File name	File path	☑ Read only	Need to save	New file existed	Occupied
1	☑	AssemblyTest.Z...	C:\Users\Admi...	☑	No	No	No

【Notes】：

- Reloading can only indicate whether the model needs to be saved, while it cannot save it. To save the model, you should cancel reloading and save it separately.
- The file cannot be switched to write if the system restricts it to read-only.
- Virtual components are determined by the permissions of the assembly they belong to.

【Example】：

When multiple accounts design the same assembly model at the same time, the account with the "write" permission modifies and saves the target component, and the account with the "read only" permission reloads the component using the "**Reload File**" command. Without closing or reopening the model, you can get the latest model, achieving the purpose of quickly finding and updating the model.

【Where is it】：

Part/Assembly >> File >> Reload File

1.1.5 Resource Configuration Optimization

ZW3D 2025 has upgraded the software resource configuration. The software has further improved its capabilities in configuration file support, hotkey configuration, interface configuration, and secondary development support.

【What can be done】:

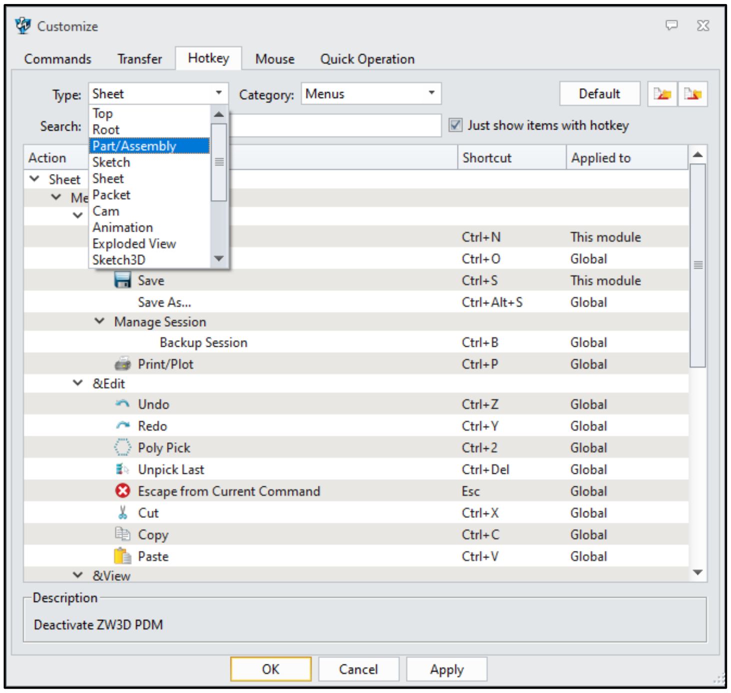
- ✓ The version compatibility of configuration files has been improved. User directory files and role files of ZW3D 2025 can be compatible with ZW3D 2025 and later versions.
- ✓ The hotkeys support the configuration of multiple environments. The same command can be configured with different keys in different environments.
- ✓ The software interface can be updated immediately without restarting the software after switching the software authorization.
- ✓ The resource configuration capability of secondary development users has been improved. For example, users can save from cross-version adaptation, modify the current layout and command distribution in controls, add new commands, and register a local configuration file path.

【Note】:

- ZW3D 2025 is not perfectly compatible with older configuration files.

【Example】:

1. Multi-environment configuration of the hotkeys. When using “Global”, you can uniformly set the command’s hotkeys in all environments. When “This module” is used, you can set different hotkeys for the command in different environments.


Customize
Commands
Transfer
Hotkey
Mouse
Quick Operation
Type:
Sheet
Top
Root
Part/Assembly
Sketch
Sheet
Packet
Cam
Animation
Exploded View
Sketch3D
Category:
Menus
Default
Search:
☑ Just show items with hotkey
OK
Cancel
Apply

Action	Shortcut	Applied to
Sheet		
Me		
	Ctrl+N	This module
	Ctrl+O	Global
Save	Ctrl+S	This module
Save As...	Ctrl+Alt+S	Global
Manage Session		
Backup Session	Ctrl+B	Global
Print/Plot	Ctrl+P	Global
&Edit		
Undo	Ctrl+Z	Global
Redo	Ctrl+Y	Global
Poly Pick	Ctrl+2	Global
Unpick Last	Ctrl+Del	Global
Escape from Current Command	Esc	Global
Cut	Ctrl+X	Global
Copy	Ctrl+C	Global
Paste	Ctrl+V	Global
&View		

Description

Deactivate ZW3D PDM

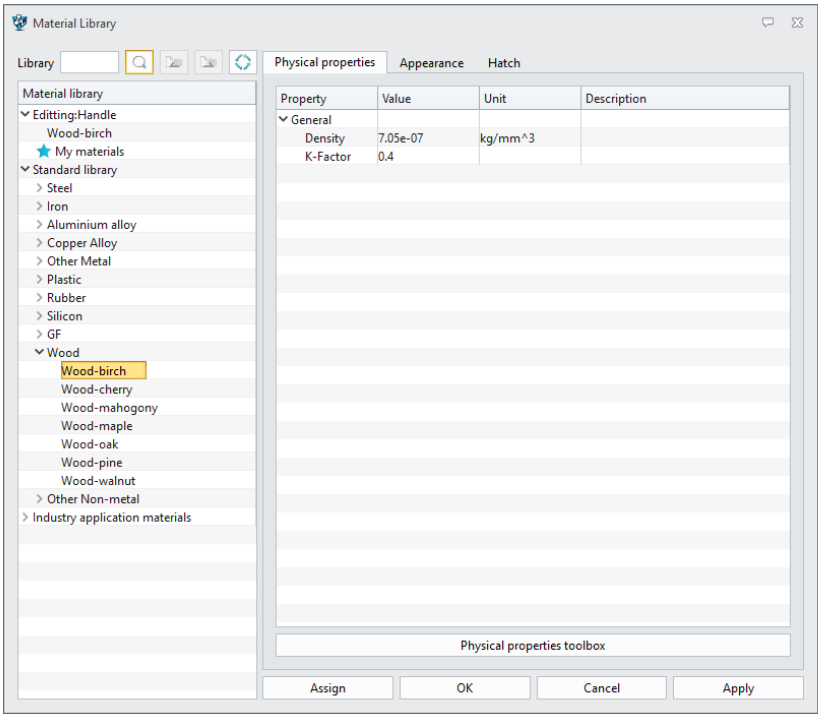
【Where is it】：

All Environments >> Ribbon Bar >> Customize >> Hotkey

1.2 ★Material Library

Material Library has been added to ZW3D 2025, allowing you to manage and invoke uniformly the material library panel across all platforms. Besides, the materials are classified according to the attribute categories, and the rapid reuse of materials is supported. The unified entrance of material library helps you to define material library. In addition, you can assign material appearance and section lines. For multi-entity parts or assembly files and drawing sheets, you can better distinguish which entity or part has the same material through material appearance.

Functions such as face attributes, texture map, metal finishing, and light effect are merged into the Appearance command, which decreases the number of commands. The preset Appearance Library provides non-professional rendering users with convenient reference parameters of product appearance, reducing the learning costs.


Material Library
Library
Physical properties
Appearance
Hatch
Material library
Editting:Handle
Wood-birch
My materials
Standard library
Steel
Iron
Aluminium alloy
Copper Alloy
Other Metal
Plastic
Rubber
Silicon
GF
Wood
Wood-birch
Wood-cherry
Wood-mahogony
Wood-maple
Wood-oak
Wood-pine
Wood-walnut
Other Non-metal
Industry application materials
Physical properties toolbox
Assign
OK
Cancel
Apply

Property	Value	Unit	Description
General			
Density	7.05e-07	kg/mm^3	
K-Factor	0.4		

【What can be done】：

- ✓ Specify part/entity material.
- ✓ Edit material parameters and add different material parameters.
- ✓ Self-define materials and assign material appearance and section lines.
- ✓ Import/Export material library folder and TXT file is supported in import.
- ✓ Quickly assign different appearances in the library to parts or entities.
- ✓ Appearance format painter can help you quickly reuse appearance.
- ✓ Clearing files or specific shape appearances are supported.

【Note】：

- At present, the component texture mapping is not supported in the assembly environment.

【Example】：

Please check the help documentation.

【Where is it】:

Tools >> Material

Visualize >> Appearance

File Browser >> Appearance Library

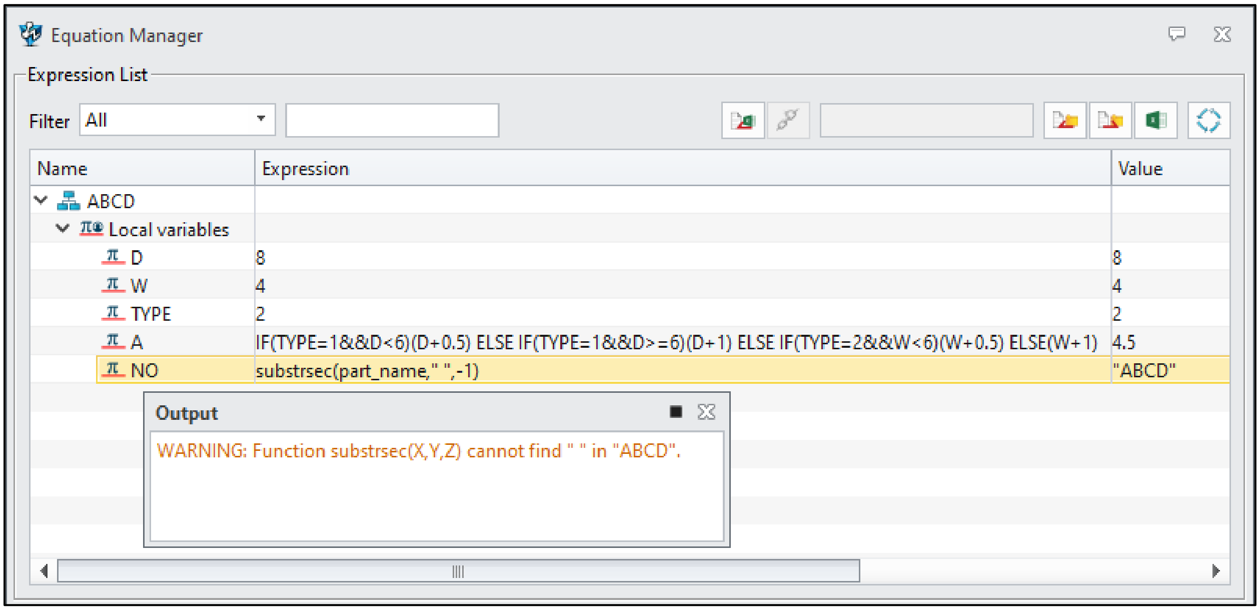
1.3 Tools

1.3.1 Equation Manager Optimization

ZW3D 2025 has optimized the equation manager mechanism, which increases the accuracy of the function parses, including the substrsec function and the if function, and enhances the refreshing mechanism.

【What can be done】:

- ✓ When the substrsec function fails to distinguish between front and back, it will extract all by default.
- ✓ The optimized parsing logic of the if function supports variable names including the "if" character.



Name	Expression	Value
ABCD		
Local variables		
D	8	8
W	4	4
TYPE	2	2
A	IF(TYPE=1&&D<6)(D+0.5) ELSE IF(TYPE=1&&D>=6)(D+1) ELSE IF(TYPE=2&&W<6)(W+0.5) ELSE(W+1)	4.5
NO	substrsec(part_name," ",-1)	"ABCD"

Output

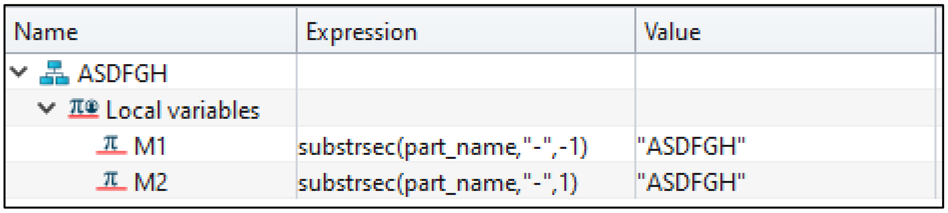
```
WARNING: Function substrsec(X,Y,Z) cannot find " " in "ABCD".
```

【Note】:

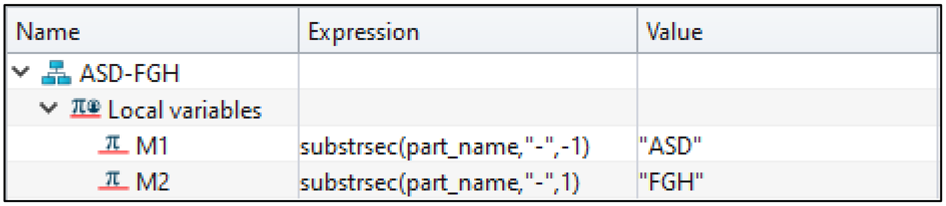
- The if function format should be correct.

【Example】：

1. A part name is composed of "drawing name", “-”, and "drawing number". When using the "substrsec" function to extract the front and back parts of "-" in the part name and place them in the "sheet_name" and "sheet_code" properties, if the actual part name created based on the template does not include "-" for the time being, the older version of ZW3D cannot extract the function and the properties will stay empty. If the new function logic cannot obtain the Y boundary, it will extract all by default. If you modify the part name later, a modified name with “-” can be used normally.

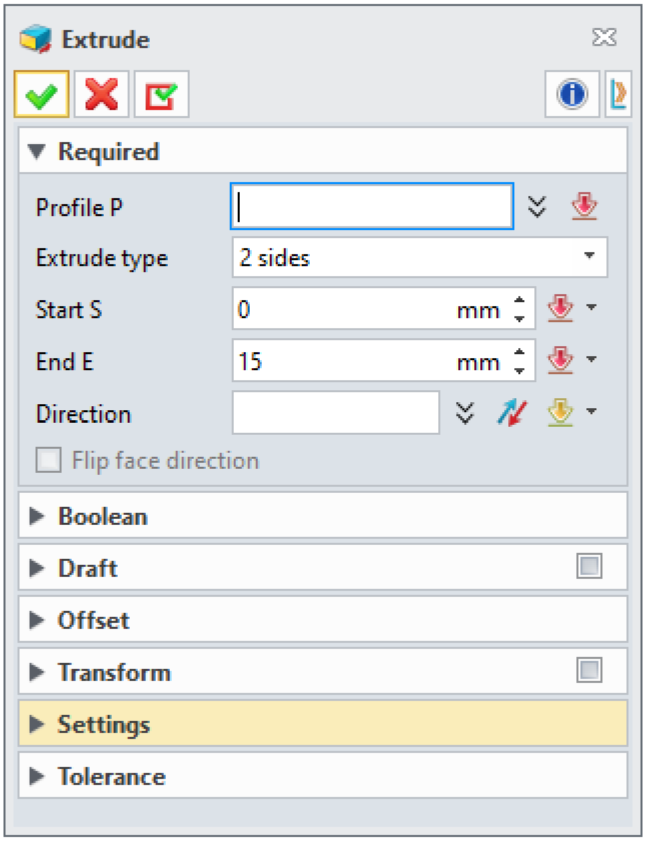


Name	Expression	Value
ASDFGH		
Local variables		
M1	substrsec(part_name,"-",-1)	"ASDFGH"
M2	substrsec(part_name,"-",1)	"ASDFGH"



Name	Expression	Value
ASD-FGH		
Local variables		
M1	substrsec(part_name,"-",-1)	"ASD"
M2	substrsec(part_name,"-",1)	"FGH"

2. Variable names include IF, if, such as DIF, ifc, etc.



Extrude
Required
Boolean
Draft
Offset
Transform
Settings
Tolerance

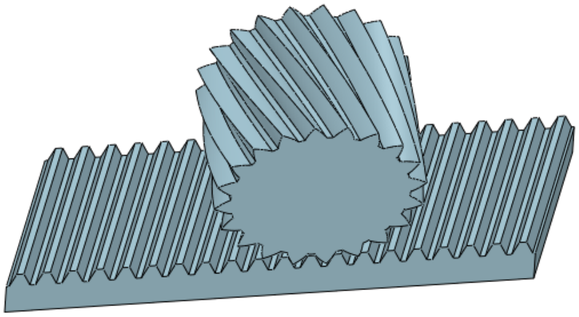
Profile P
Extrude type: 2 sides
Start S: 0 mm
End E: 15 mm
Direction
☐ Flip face direction

【Where is it】:

Part/Assembly >> Tools >> Equation Manager

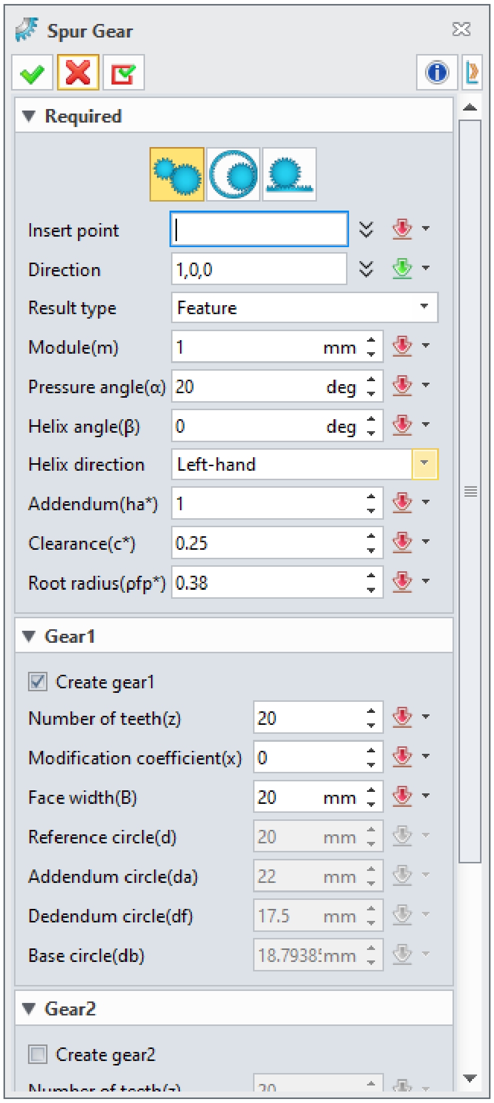
1.3.2 Spur Gear Optimization

The mechanical design process may include helical racks. The optimized "**Spur Gear**" can produce more accurate racks, such as helical racks. The teeth number of a helical rack must be complete for reasonable manufacturing purposes.



【Where is it】:

✓ Generate racks with full teeth only.



Spur Gear

▼ Required

Insert point
Direction 1,0,0
Result type Feature
Module(m) 1 mm
Pressure angle(α) 20 deg
Helix angle(β) 0 deg
Helix direction Left-hand
Addendum(ha*) 1
Clearance(c*) 0.25
Root radius(pfp*) 0.38

▼ Gear1

☑ Create gear1
Number of teeth(z) 20
Modification coefficient(x) 0
Face width(B) 20 mm
Reference circle(d) 20 mm
Addendum circle(da) 22 mm
Dedendum circle(df) 17.5 mm
Base circle(db) 18.7938 mm

▼ Gear2

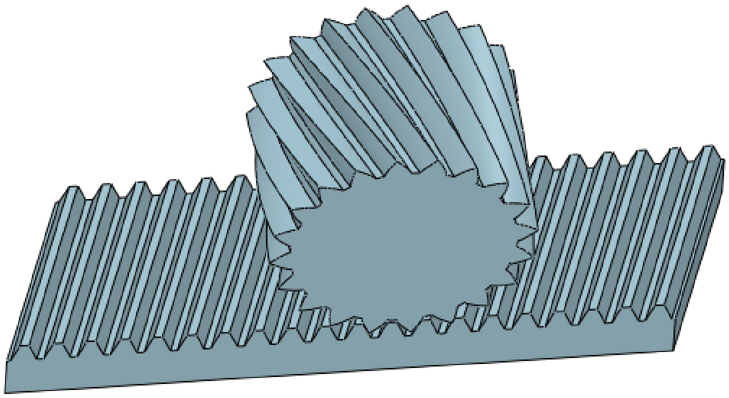
☐ Create gear2

【Note】:

- There is a small amount of interference between the parametric generating teeth.

【Example】:

1. Generate a rack with a spiral angle of 20 degrees by using "**Pinion and rack mechanism**" of the "**Spur Gear**" command. The teeth shape of the rack always remains complete so that it can be manufactured and used.



【Where is it】:

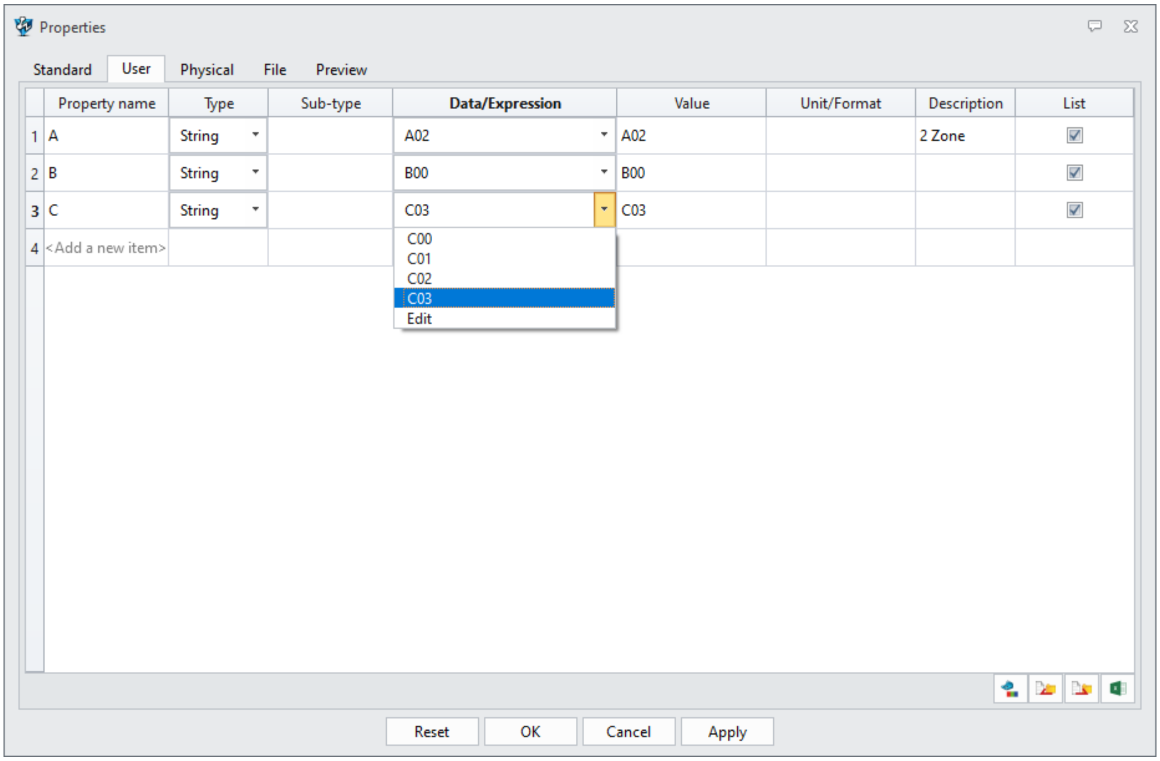
Part/Assembly >> Tools >> Spur Gear

1.3.3 ★New Series Value Property

In the part and assembly design process, the new "**Set Series Value** " will set multiple target values, such as an attribute with only an optimal drop-down option value. By presetting the list series values, the "Data/Expression" corresponding to the attribute is preferentially selected.

【What can be done】:

- ✓ The string type supports list series values.
- ✓ The integer type supports list series values.
- ✓ The number type supports list series values.
- ✓ Add descriptions in Attribute/Series Value.



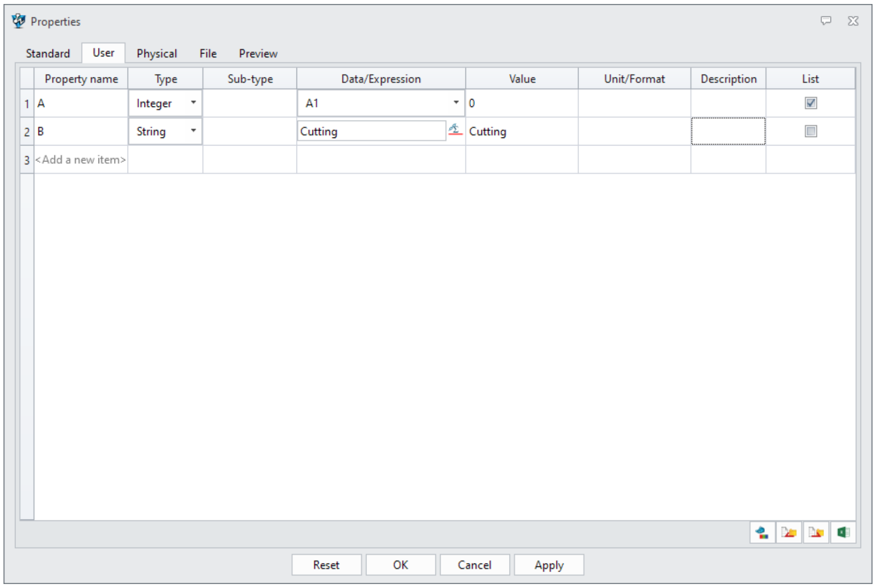
	Property name	Type	Sub-type	Data/Expression	Value	Unit/Format	Description	List
1	A	String		A02	A02		2 Zone	☑
2	B	String		B00	B00			☑
3	C	String		C03	C03			☑
4	<Add a new item>							

【Notes】:

- The exported or imported tables with series values are not supported at the time and can only be edited in the table.
- Currently, the association selection between attribute values is not supported.

【Example】:

1. Attribute A represents the manufacturing process and there are only two manufacturing processes. You can add two processes through the "**List**" option of the "**Property**" command for easy selection and modification. At the same time, it can be used to predefine the attributes of parts and assembly templates.
2. Attribute B represents manufacturing technology, which requires noting the boundary conditions of manufacturing technology. You can add detailed descriptions in the “**Description**” option of the “**Property**” command.



	Property name	Type	Sub-type	Data/Expression	Value	Unit/Format	Description	List
1	A	Integer		A1	0			☑
2	B	String		Cutting	Cutting			☐
3	<Add a new item>							

【Where is it】:

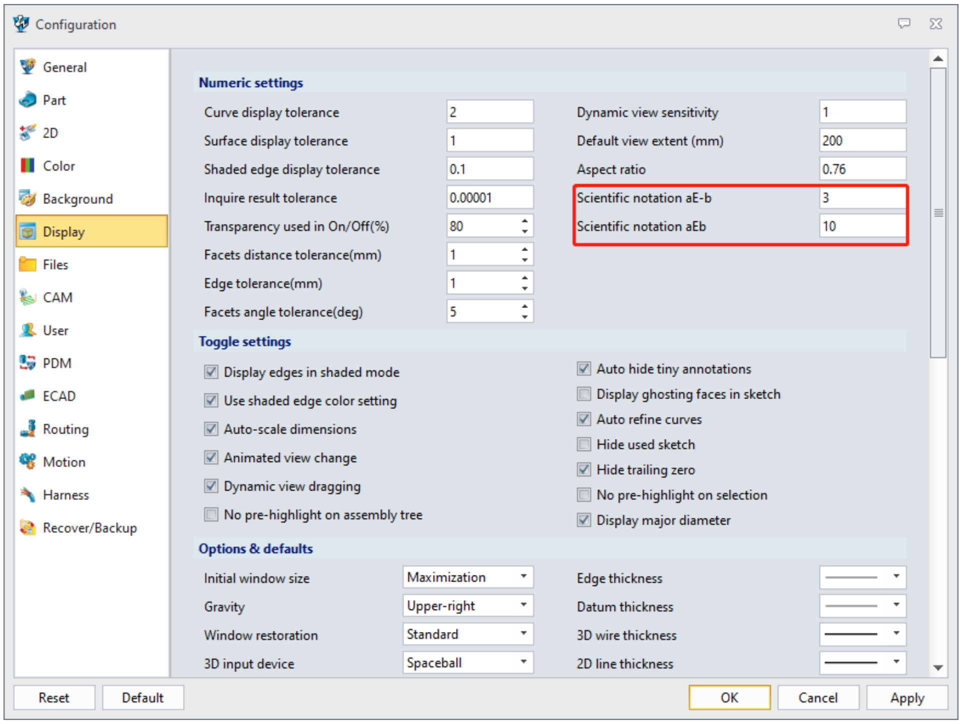
Part/Assembly >> Tools >> Properties

1.3.4 New Scientific Notation Control

In ZW3D, the physical property values are usually automatically converted to scientific notation, which cannot be quickly identified compared to conventional values. Set the conversion boundary of the maximum and minimum scientific notation by using the "**Display**" option of the "**Configuration**" function in the upper right corner of ZW3D. By setting the boundary of b, the value's control ability of the display style has been improved, achieving the purpose of controlling the data display style of attribute values.

【What can be done】:

- ✓ When aE-b is set as 3, values less than 0.01 (3 decimal places) are converted to scientific notation.
- ✓ When aEb is set as 6, values greater than or equal to 1,000,000 (6-digit integers) are converted to scientific notation.



Numeric settings

Curve display tolerance	2	Dynamic view sensitivity	1
Surface display tolerance	1	Default view extent (mm)	200
Shaded edge display tolerance	0.1	Aspect ratio	0.76
Inquire result tolerance	0.00001	Scientific notation aE-b	3
Transparency used in On/Off(%)	80	Scientific notation aEb	10
Facets distance tolerance(mm)	1		
Edge tolerance(mm)	1		
Facets angle tolerance(deg)	5		

Toggle settings

☑ Display edges in shaded mode
☑ Use shaded edge color setting
☑ Auto-scale dimensions
☑ Animated view change
☑ Dynamic view dragging
☐ No pre-highlight on assembly tree
☑ Auto hide tiny annotations
☐ Display ghosting faces in sketch
☑ Auto refine curves
☐ Hide used sketch
☑ Hide trailing zero
☐ No pre-highlight on selection
☑ Display major diameter

Options & defaults

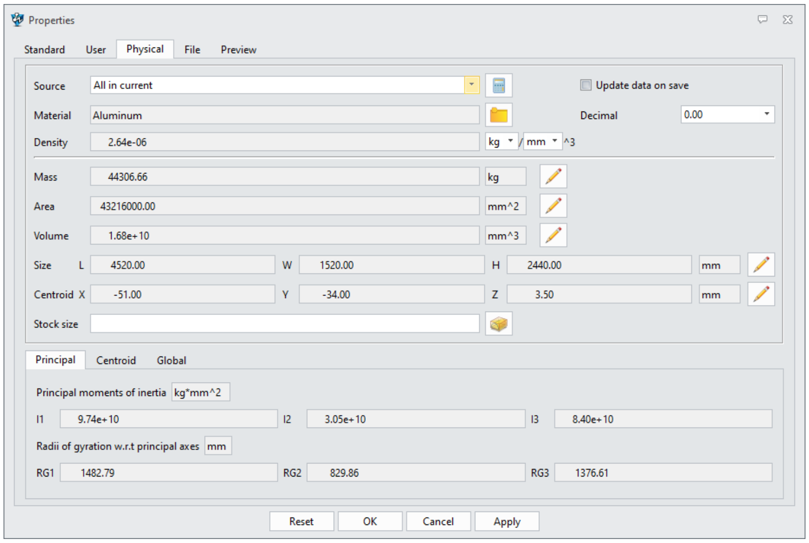
Initial window size	Maximization	Edge thickness	
Gravity	Upper-right	Datum thickness	
Window restoration	Standard	3D wire thickness	
3D input device	Spaceball	2D line thickness	

【Notes】：

- The minimum value of aE-b is 1 and the maximum value 7. When setting aE-b to a larger size, pay attention to setting reasonable and valid decimal places.
- The minimum value of aEb is 1 and the maximum value is 14.

【Example】：

1. If you want scientific notation to appear in material properties, you can use the "Display" option in "Configuration" to set aE-b to a maximum of 7 and aEb to a maximum of 14. Review the physical property value and set the decimal to 7 digits.


Properties
Standard
User
Physical
File
Preview
Principal
Centroid
Global
Reset
OK
Cancel
Apply

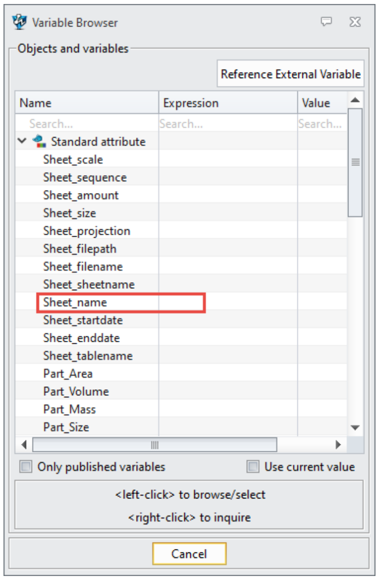
Source: All in current
☐ Update data on save
Material: Aluminum
Decimal: 0.00
Density: 2.64e-06 kg / mm ^3
Mass: 44306.66 kg
Area: 43216000.00 mm^2
Volume: 1.68e+10 mm^3
Size L: 4520.00 W: 1520.00 H: 2440.00 mm
Centroid X: -51.00 Y: -34.00 Z: 3.50 mm
Stock size:
Principal moments of inertia: kg*mm^2
I1: 9.74e+10 I2: 3.05e+10 I3: 8.40e+10
Radii of gyration w.r.t principal axes: mm
RG1: 1482.79 RG2: 829.86 RG3: 1376.61

【Where is it】:

Part/Assembly >> Configuration >> Display

1.3.5 New Sheet Name Variable Property

In ZW3D 2025, we add new Sheet_name attribute to solve the problem of obtaining sheet name in the drawing sheet, achieving the goal of self-refreshing the variables.


Variable Browser
Objects and variables
Reference External Variable
Name
Expression
Value
Search...
Search...
Search...
☐ Only published variables
☐ Use current value
Cancel

Standard attribute
Sheet_scale
Sheet_sequence
Sheet_amount
Sheet_size
Sheet_projection
Sheet_filepath
Sheet_filename
Sheet_sheetname
Sheet_name
Sheet_startdate
Sheet_enddate
Sheet_tablename
Part_Area
Part_Volume
Part_Mass
Part_Size

<left-click> to browse/select
<right-click> to inquire

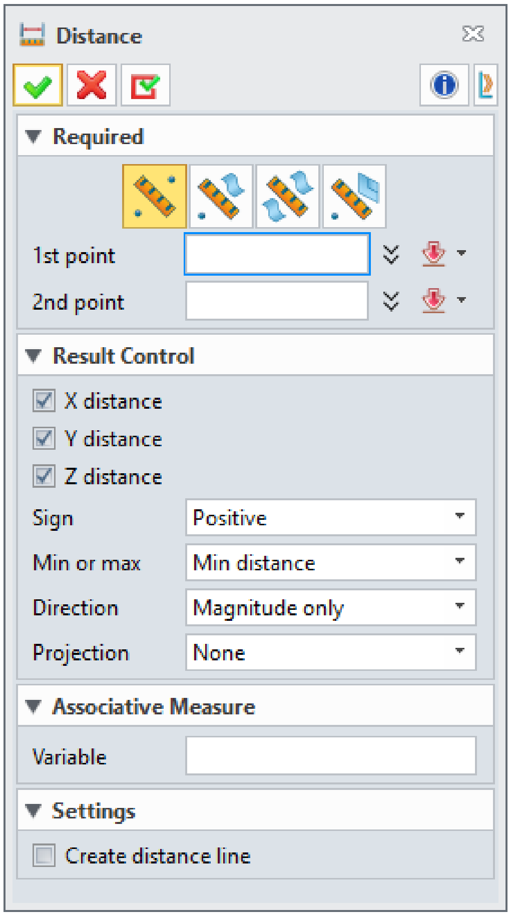
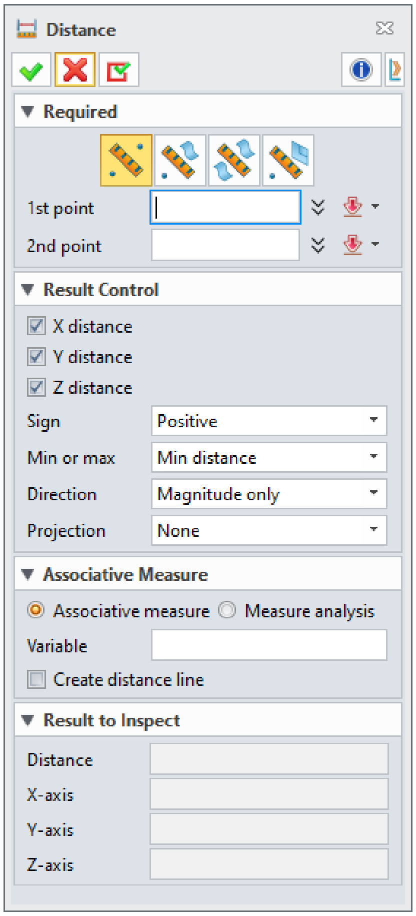
【Where is it】:

Drawing Sheet >> Tools >> Variable Browser

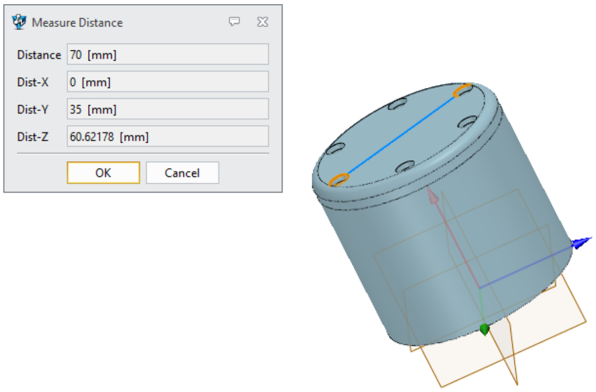
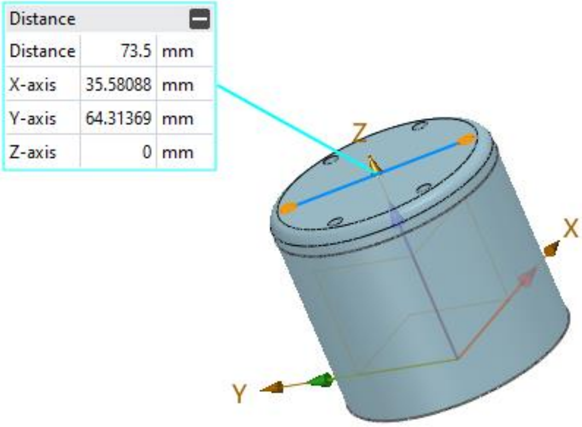
1.4 Inquire

1.4.1 Measure

In ZW3D 2025, the measurement tools get fully upgraded, and measuring analysis is supported, by which users can choose whether to enable the real-time update of measuring results in the Configuration option. When the function is on, users can check the object's change in real time through measure analysis and set the threshold value condition. When the measured result meets the threshold condition, for example, when the distance between two surfaces exceeds the set threshold, the system will warn users to modify the model design.

ZW3D 2024	ZW3D 2025
 **Distance** **Required** 1st point 2nd point **Result Control** ☑ X distance ☑ Y distance ☑ Z distance Sign: Positive Min or max: Min distance Direction: Magnitude only Projection: None **Associative Measure** Variable **Settings** ☐ Create distance line	 **Distance** **Required** 1st point 2nd point **Result Control** ☑ X distance ☑ Y distance ☑ Z distance Sign: Positive Min or max: Min distance Direction: Magnitude only Projection: None **Associative Measure** ● Associative measure ○ Measure analysis Variable ☐ Create distance line **Result to Inspect** Distance X-axis Y-axis Z-axis

In addition, the display method is upgraded to better check the measuring result.

ZW3D 2024	ZW3D 2025
 Measure Distance Distance 70 [mm] Dist-X 0 [mm] Dist-Y 35 [mm] Dist-Z 60.62178 [mm] 	 Distance Distance 73.5 mm X-axis 35.58088 mm Y-axis 64.31369 mm Z-axis 0 mm

【What can be done】：

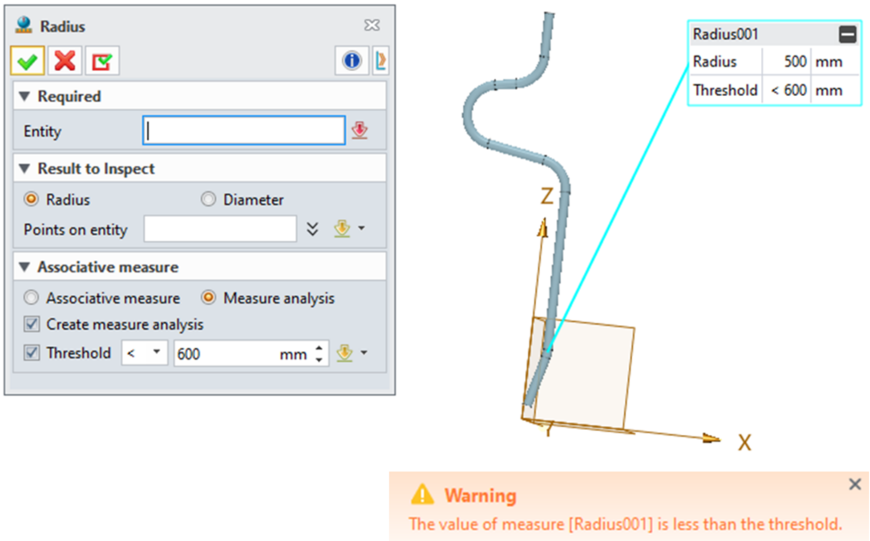
- ✓ Inquire about object data in real time when designing model.
- ✓ Set threshold conditions in model design. Tips will pop up when the model satisfies the threshold conditions to avoid parameter modification resulting in the model size out of limit.

【Note】：

- When the real-time update of measuring results is enabled, the software will automatically update the measuring results after modifying the model, and the overall time of the model update will become slightly longer.

【Example】：

1. In piping design, the minimum bending radius of a curve is taken as the measured value and the threshold is set. The system will prompt you if the minimum bending radius exceeds the threshold.


Radius
Required
Entity
Result to Inspect
Points on entity
Associative measure
Z
X

○ Radius ● Radius ○ Diameter

○ Associative measure ● Measure analysis

☑ Create measure analysis

☑ Threshold < 600 mm

Radius001		
Radius	500	mm
Threshold	< 600	mm

Warning

The value of measure [Radius001] is less than the threshold.

【Where is it】:

Inquire>> Measure

1.4.2 New Solid Compare（ZW3D 2025 SP）

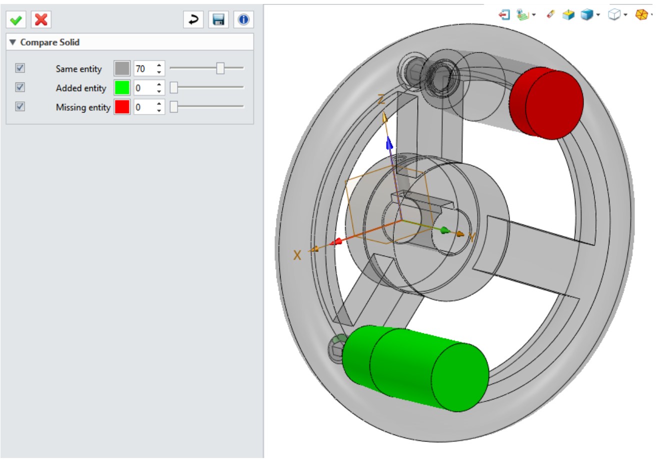
ZW3D 2025 SP includes a Solid Compare Function. It supports comparing the solid differences between two parts and displays the differences using different colors.

【What can be done】:

- ✓ Supports comparing solid differences between parts.

【Example】:

Select two similar parts for Solid Compare; differences between the parts can be observed in the drawing area using different colors.



Compare Solid

☑ Same entity ▨ 70

☑ Added entity 🟩 0

☑ Missing entity 🟥 0

【Where is it】:

Inquire>> Compare Parts

1.5 Customize Configuration Optimization

The “**Customize configuration”** realizes the flexible setting of command hotkeys, including the quick calling of frequent actions like OK, Cancel, and Apply, and the efficient reuse of model face colors.

1.5.1 ★Commands Hotkey Globally Related/Individual Module Settings

When users set command hotkeys, the new “Global” function can automatically unify the same command hotkey in different models, while the new “This module” function can make the hotkey setting limited to the current model. Therefore, users can flexibly set command hotkey. After laying out the Command List of hotkey settings again, the clearer distinction of commands in different modules enables users to find the corresponding commands faster.

【What can be done】:

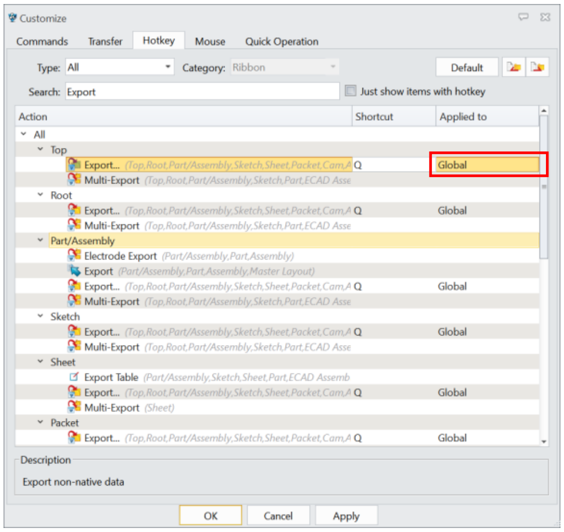
- ✓ Mostly, the same command in different modules supports one-click setting, which means the same hotkey in different file environments can activate the same command.
- ✓ Commands can be set to different hotkeys in distinct modules, which means the same command can be set to different hotkeys, and the same hotkey can activate different commands.

【Note】:

- The hotkey settings do not support the command actions of OK, Cancel, and Apply.

【Example】:

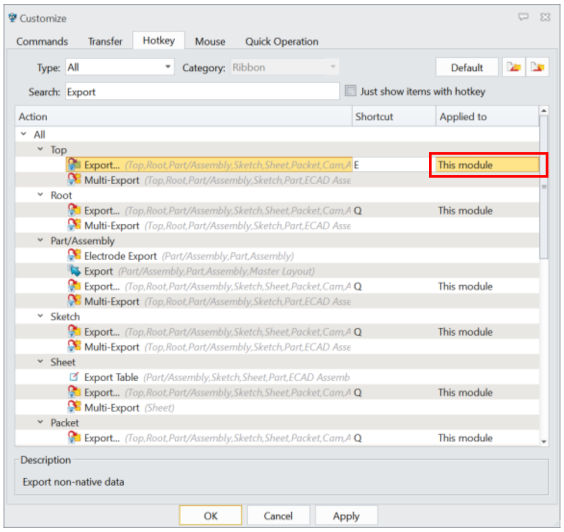
1. Set “All” as “Type” in the Hotkey configuration panel in Customize and enter “Export” in “Search”.
2. If you set any the “Export” command hotkey as “Q” and select “Global” in “Applied to”, then the hotkey of “Export” command in other modules will be set as “Q” automatically.
3. If you select “This module” in “Applied to” for any “Export” and set its hotkey as “E”, then the settings will not affect other “Export” commands’ hotkey.



Type: All
Category: Ribbon
Search: Export
☐ Just show items with hotkey

Action	Shortcut	Applied to
All		
Top		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	Global
Multi-Export *(Top,Root,Part/Assembly,Sketch,Part,ECAD Asse*		
Root		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	Global
Multi-Export *(Top,Root,Part/Assembly,Sketch,Part,ECAD Asse*		
Part/Assembly		
Electrode Export *(Part/Assembly,Part,Assembly)*		
Export *(Part/Assembly,Part,Assembly,Master Layout)*		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	Global
Multi-Export *(Top,Root,Part/Assembly,Sketch,Part,ECAD Asse*		
Sketch		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	Global
Multi-Export *(Top,Root,Part/Assembly,Sketch,Part,ECAD Asse*		
Sheet		
Export Table *(Part/Assembly,Sketch,Sheet,Part,ECAD Assemb*		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	Global
Multi-Export *(Sheet)*		
Packet		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	Global

Description
Export non-native data



Type: All
Category: Ribbon
Search: Export
☐ Just show items with hotkey

Action	Shortcut	Applied to
All		
Top		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	E	This module
Multi-Export *(Top,Root,Part/Assembly,Sketch,Part,ECAD Asse*		
Root		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	This module
Multi-Export *(Top,Root,Part/Assembly,Sketch,Part,ECAD Asse*		
Part/Assembly		
Electrode Export *(Part/Assembly,Part,Assembly)*		
Export *(Part/Assembly,Part,Assembly,Master Layout)*		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	This module
Multi-Export *(Top,Root,Part/Assembly,Sketch,Part,ECAD Asse*		
Sketch		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	This module
Multi-Export *(Top,Root,Part/Assembly,Sketch,Part,ECAD Asse*		
Sheet		
Export Table *(Part/Assembly,Sketch,Sheet,Part,ECAD Assemb*		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	This module
Multi-Export *(Sheet)*		
Packet		
Export... *(Top,Root,Part/Assembly,Sketch,Sheet,Packet,Cam,A*	Q	This module

Description
Export non-native data

【Where is it】:

All Environments >> Customize >> Hotkey

1.5.2 Quick Operation Settings of OK/Cancel/Apply

When users are performing command operations to geometric objects in the drawing areas, they can directly set the command actions such as OK, Cancel, and Apply in “Mouse Gesture” or “Shortcut Bar” without returning to the command panel and clicking relevant buttons, which reduces unnecessary operation and improves the command efficiency.

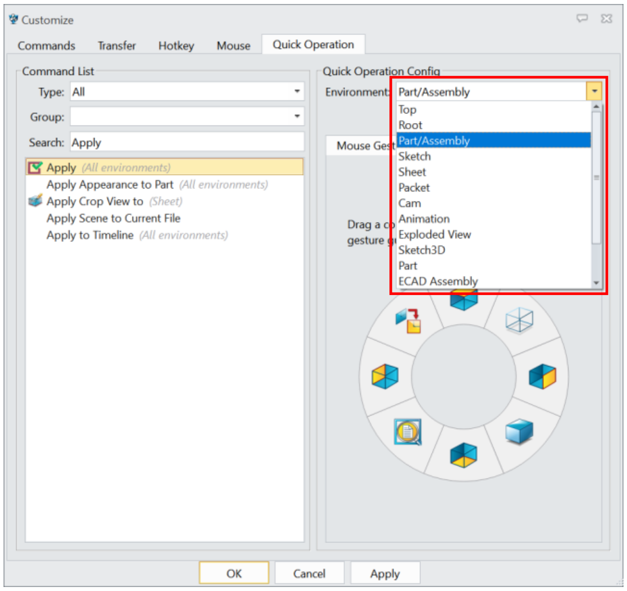
【What can be done】:

✓ Support configuring the actions of OK, Cancel, and Apply to the Mouse Gesture.

✓ Support configuring the actions of OK, Cancel, and Apply to the Shortcut Bar.

【Note】:

- Command’s Mouse Gesture and Shortcut Bar need to distinguish the modules.


Customize
Commands
Transfer
Hotkey
Mouse
Quick Operation
Command List
Quick Operation Config
Top
Root
Part/Assembly
Sketch
Sheet
Packet
Cam
Animation
Exploded View
Sketch3D
Part
ECAD Assembly
OK
Cancel
Apply

Type: All

Group:

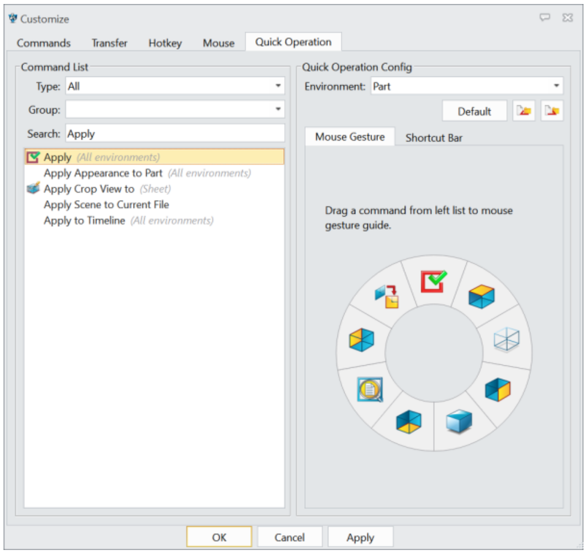
Search: Apply

Apply (All environments)
Apply Appearance to Part (All environments)
Apply Crop View to (Sheet)
Apply Scene to Current File
Apply to Timeline (All environments)

Environment: Part/Assembly

【Example】:

1. Select “All” as “Type” in the Quick Operation panel in the Customize command, enter “Apply” in “Search”, and then select “Part” in “Environment”.
2. Drag “Apply” in the Command List to “Mouse Gesture” and “Shortcut Bar” respectively.


Customize
Commands
Transfer
Hotkey
Mouse
Quick Operation
Command List
Quick Operation Config
Default
Mouse Gesture
Shortcut Bar
OK
Cancel
Apply

Type: All

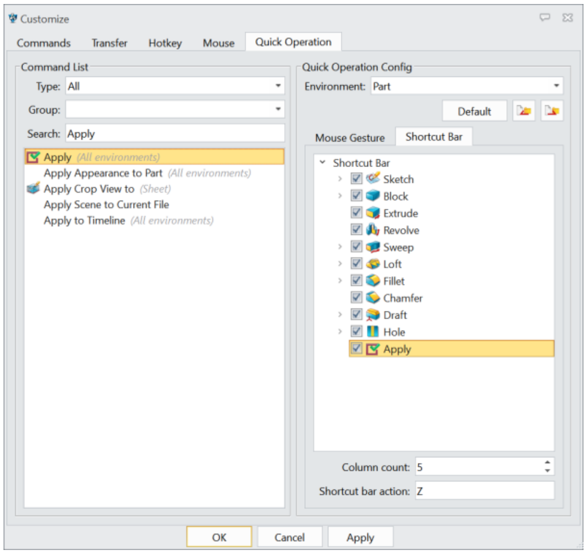
Group:

Search: Apply

Apply (All environments)
Apply Appearance to Part (All environments)
Apply Crop View to (Sheet)
Apply Scene to Current File
Apply to Timeline (All environments)

Environment: Part

Drag a command from left list to mouse gesture guide.


Customize
Commands
Transfer
Hotkey
Mouse
Quick Operation
Command List
Quick Operation Config
Default
Mouse Gesture
Shortcut Bar
OK
Cancel
Apply

Type: All

Group:

Search: Apply

Apply (All environments)
Apply Appearance to Part (All environments)
Apply Crop View to (Sheet)
Apply Scene to Current File
Apply to Timeline (All environments)

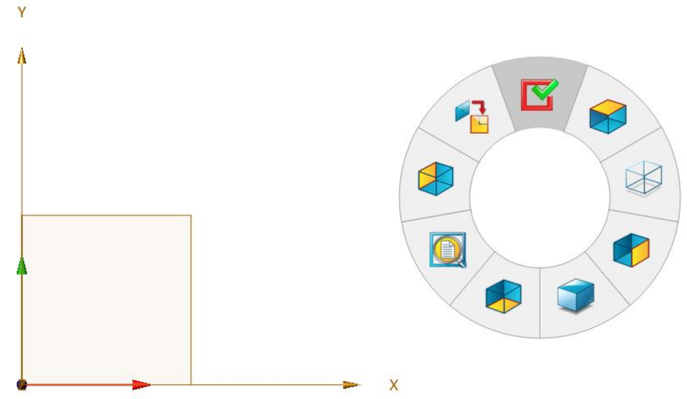
Environment: Part

- Shortcut Bar
  - ☑ Sketch
  - ☑ Block
  - ☑ Extrude
  - ☑ Revolve
  - ☑ Sweep
  - ☑ Loft
  - ☑ Fillet
  - ☑ Chamfer
  - ☑ Draft
  - ☑ Hole
  - ☑ Apply

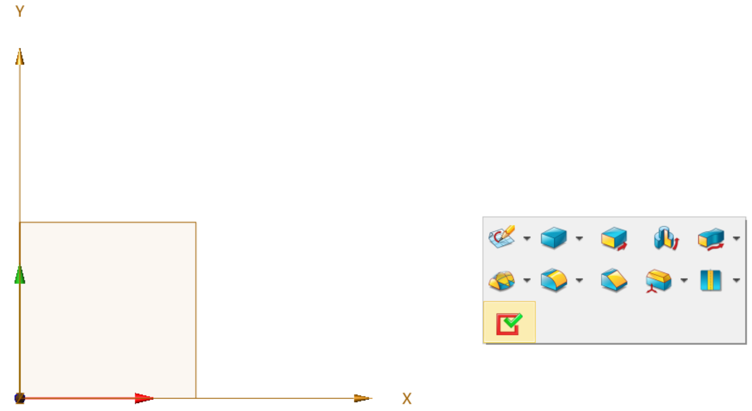
Column count: 5

Shortcut bar action: Z

3. Open any part file, press Shift and mouse's right-click to slide in the drawing area, and then open the Mouse Gesture panel.



4. Open any part file, press Z button in the drawing area, and then open the Shortcut Bar.



【Where is it】:

All Environment >> Customize >> Quick Operation

1.5.3 New Used Color and Custom Color in Face Color

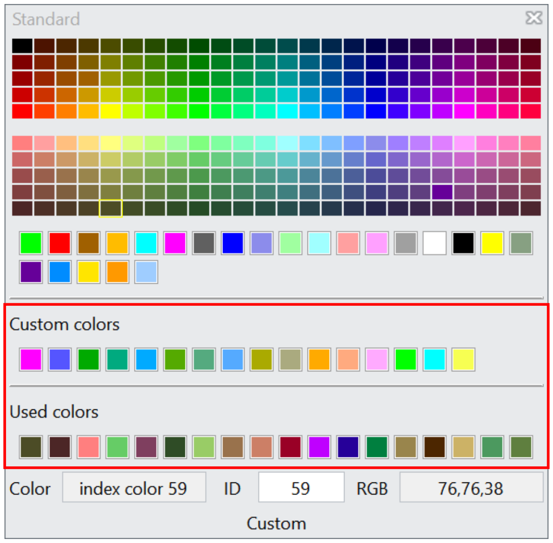
When changing model face colors, users can not only check the used colors in the Face Color panel for reuse but also directly use the set custom colors without re-entering the Custom Color panel, which is beneficial to the application of enterprise model color standards by reducing the operation steps and increasing color efficiency.

【What can be done】:

- ✓ Custom colors and used colors can be directly applied to the Face Color panel.

【Notes】:

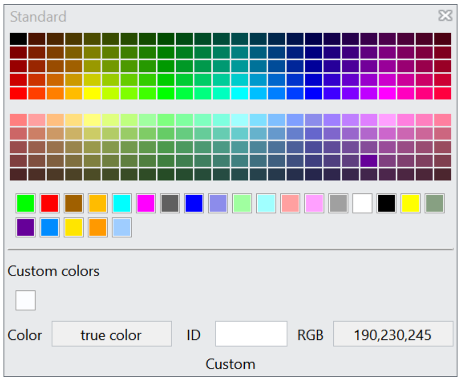
- Custom colors can display 16 colors at most.
- Used colors can display 18 colors at most.


Standard
Custom colors
Used colors
Custom

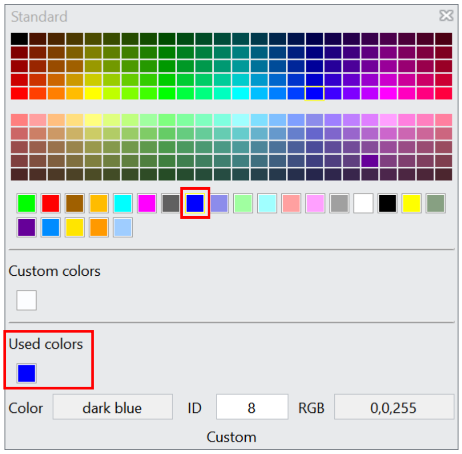
Color: index color 59 ID: 59 RGB: 76,76,38

【Example】:

1. Open the Face Color panel, select any color, and then "Used colors" pop up.


Standard
Custom colors
Custom

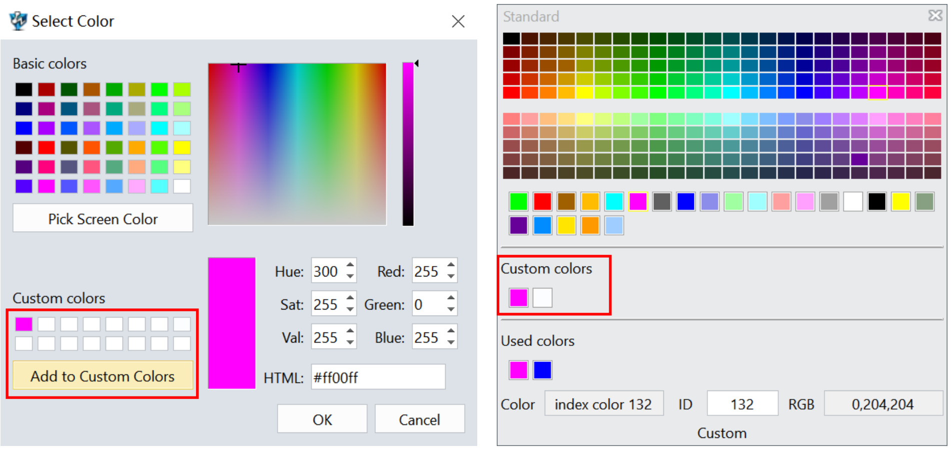
Color: true color ID: RGB: 190,230,245


Standard
Custom colors
Used colors
Custom

Color: dark blue ID: 8 RGB: 0,0,255

2. Click “Custom” to enter the Custom Colors panel, and then click “Add to Custom Colors” to add self-defined colors.

3. Click “OK” and the color is added to Custom colors simultaneously.


Select Color
Basic colors
Pick Screen Color
Custom colors
Add to Custom Colors
OK
Cancel
Standard
Custom colors
Used colors
Custom

Hue: 300
Sat: 255
Val: 255
Red: 255
Green: 0
Blue: 255
HTML: #ff00ff

Color index color 132 ID 132 RGB 0,204,204

【Where is it】：

All Environment >> DA ToolBars >>Face Color

1.5.4 Mouse Behavior Settings Expansion （ZW3D 2025 SP）

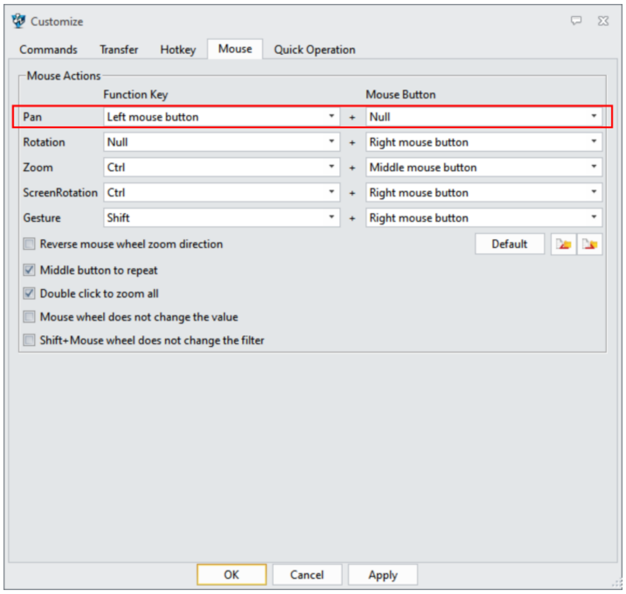
The function key of mouse behavior adds the configuration item of the left mouse button, which provides the setting methods of "left mouse button + mouse middle button" and "left mouse button + right mouse button" for translation, rotation, zoom, screen rotation shaft and gestures and other operations, so that users can do more operations with one hand.

【What can be done】:

- ✓ Mouse behavior operation supports the setting of "left mouse button + middle mouse button"
- ✓ Mouse behavior operation supports the setting of "left mouse button + right mouse button"

【Note】:

- Because long press and dragging the left mouse button has been used for box selection, the mouse behavior is set to "left mouse button + none" will not take effect.

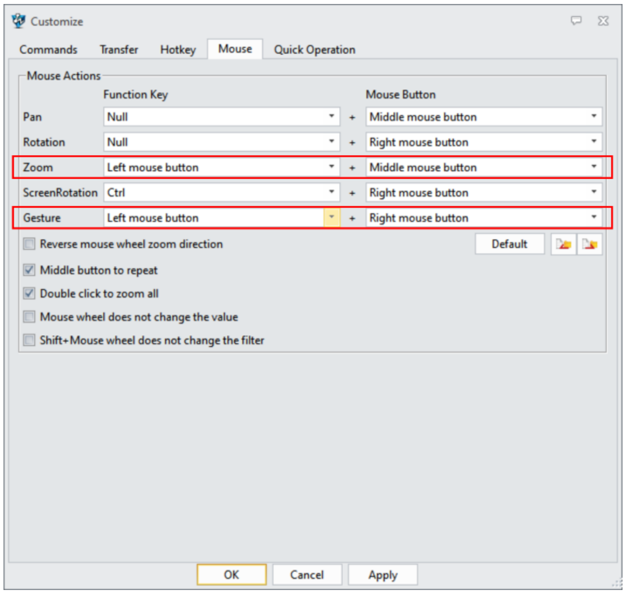

Customize
Commands
Transfer
Hotkey
Mouse
Quick Operation
Mouse Actions
Default
OK
Cancel
Apply

	Function Key		Mouse Button
Pan	Left mouse button	+	Null
Rotation	Null	+	Right mouse button
Zoom	Ctrl	+	Middle mouse button
ScreenRotation	Ctrl	+	Right mouse button
Gesture	Shift	+	Right mouse button

☐ Reverse mouse wheel zoom direction
☑ Middle button to repeat
☑ Double click to zoom all
☐ Mouse wheel does not change the value
☐ Shift+Mouse wheel does not change the filter

【Example】:

1. Open the custom panel, set "Zoom" to "left mouse button + middle mouse button", set "left mouse button + right mouse button", and click OK.


Customize
Commands
Transfer
Hotkey
Mouse
Quick Operation
Mouse Actions
Default
OK
Cancel
Apply

	Function Key		Mouse Button
Pan	Null	+	Middle mouse button
Rotation	Null	+	Right mouse button
Zoom	Left mouse button	+	Middle mouse button
ScreenRotation	Ctrl	+	Right mouse button
Gesture	Left mouse button	+	Right mouse button

☐ Reverse mouse wheel zoom direction
☑ Middle button to repeat
☑ Double click to zoom all
☐ Mouse wheel does not change the value
☐ Shift+Mouse wheel does not change the filter

2. Press the "left mouse button + middle mouse button" and slide up and down to zoom the view in the drawing area.

3. Press the "left mouse button + right mouse button" and slide to enable the mouse gesture in the drawing area.

【Where is it】

All Environment >> Customize >> Mouse

1.6 User Interface Optimization

The **"UI experience optimization"** realizes the flexible drag of the file windows order, the consistent location of common controls when switching environments, the simplified right-click panel, and the quick batch operation of the assembly config table, etc.

1.6.1 Drag File Tabs Side to Side for Sorting

When switching file windows, users can drag the commonly used window to the left and the related window to the side, which helps to quickly switch or find the corresponding window for better managing multiple windows.

【What can be done】:

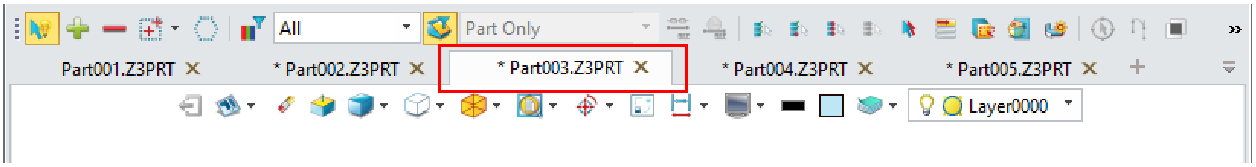
- ✓ Long pressing a file window tag with the mouse left-click button and moving side to side can switch the window order.

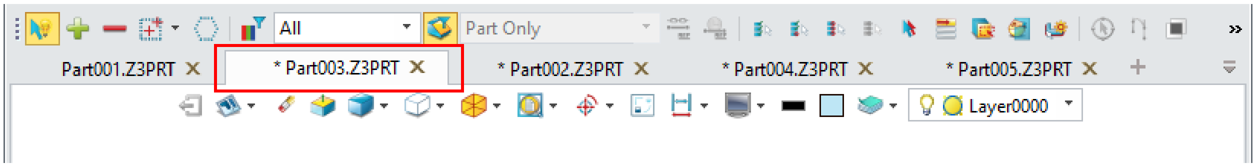
【Note】:

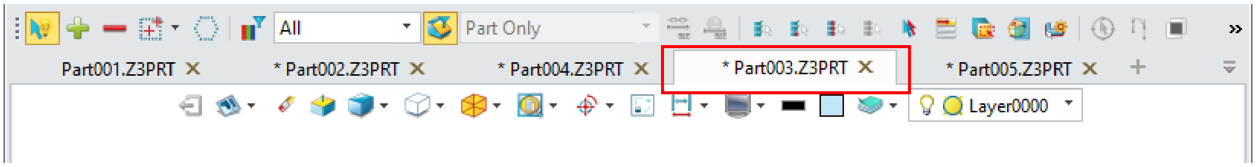
- When dragging a file window, the dragged window will be switched as the current window automatically.

【Example】:

1. Open several files at the same time.
2. Hold down the left-click button on any window tag and move side to side to change the tab location.


All
Part Only
Part001.Z3PRT
* Part002.Z3PRT
* Part003.Z3PRT
* Part004.Z3PRT
* Part005.Z3PRT
Layer0000


All
Part Only
Part001.Z3PRT
* Part003.Z3PRT
* Part002.Z3PRT
* Part004.Z3PRT
* Part005.Z3PRT
Layer0000


All
Part Only
Part001.Z3PRT
* Part002.Z3PRT
* Part004.Z3PRT
* Part003.Z3PRT
* Part005.Z3PRT
Layer0000

【Where is it】:

All Environment >> Document Title

1.6.2 Common Controls Stay Consistent when Switching Environments

When users switch files or jump to different module environments, the location and status of common controls, including the History Manager, File Browser, and Output Panel, will stay consistent, which can make users feel more comfortable during operations and help them jump to the position or state of common controls.

【What can be done】:

✓ When switching file windows, the location and status of common controls, such as the History Manager, File Browser, and Output Panel, can remain consistent.

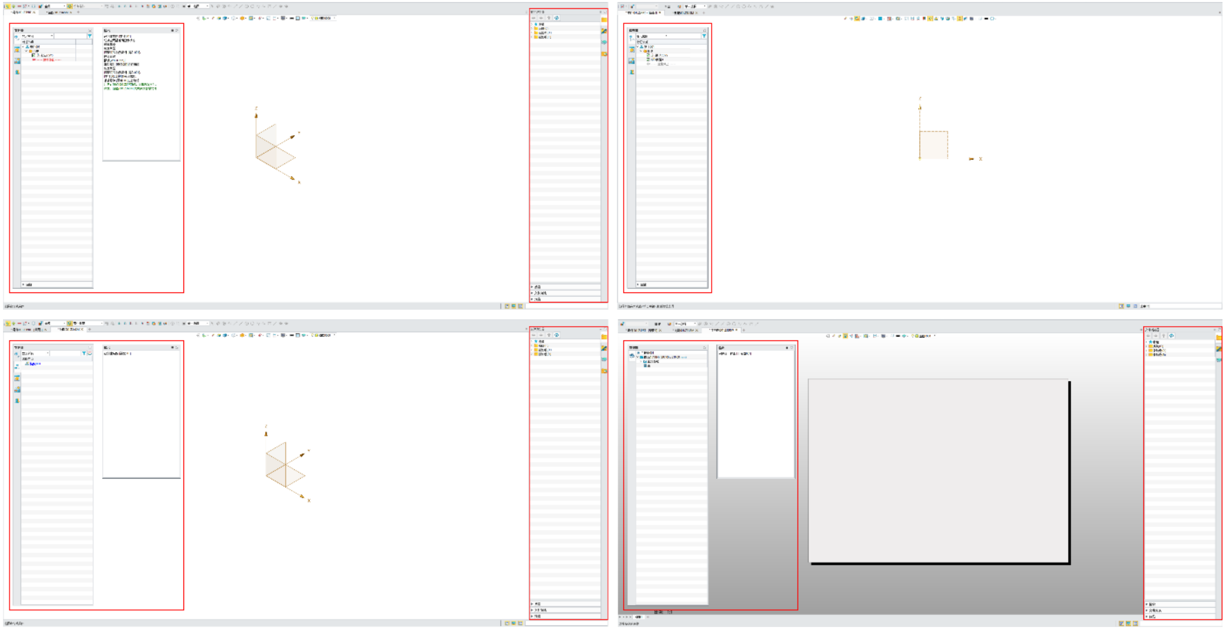
【Note】:

- As the sketch environment has no history manager, the file browser and output panel are in the hidden state by default when switching.

【Example】:

1. Open several files at the same time.
2. After the location and status of common controls, including the History Manager, File Browser,

and Output Panel, are changed at will, switch the file or jump to a different environment.



【Where is it】：

All Environment >>History Manager, File Browser and Output Panel

1.6.3 Fold Attributes and Layer Functions in Right-Click Menu

When opening the right-click menu, users can categorize and fold the related commands of Attributes and Layer. It not only can reduce the height of the menu and the number of displayed functions that affect the user experience, but also enables users to quickly search the related functions of the Attributes and Layer.

【What can be done】：

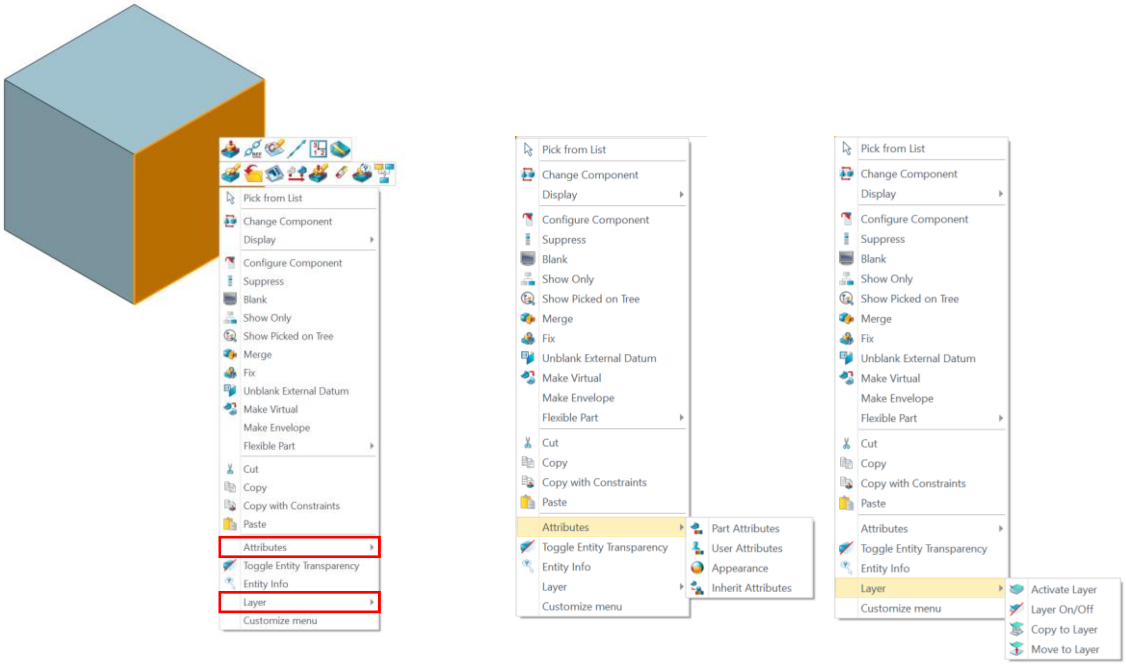
- ✓ The right-click menu for objects in the History Manager or the drawing area folds the secondary-level menus of the functions related to Attributes and Layer by default.
- ✓ Only when the mouse hovering over the Attributes and Layer menus can the secondary-level menus unfold.

【Note】：

- The second-level menus will be folded if the mouse moves out.

【Example】:

1. Pick any face of a component in the drawing area of an assembly file and open the right-click menu.
2. Use the mouse to hover over the Attributes and Layer menus respectively.


Pick from List
Change Component
Display
Configure Component
Suppress
Blank
Show Only
Show Picked on Tree
Merge
Fix
Unblank External Datum
Make Virtual
Make Envelope
Flexible Part
Cut
Copy
Copy with Constraints
Paste
Attributes
Toggle Entity Transparency
Entity Info
Layer
Customize menu
Part Attributes
User Attributes
Appearance
Inherit Attributes
Activate Layer
Layer On/Off
Copy to Layer
Move to Layer

【Where is it】:

All Environment >> Right-click Menu

1.6.4 Operate Assembly Components in Batches by Config Table

When users suppress the assembly components by the config table, the batch operation can dramatically reduce the burden of configuration management in multiple components and big assembly, improving efficiency.

【What can be done】:

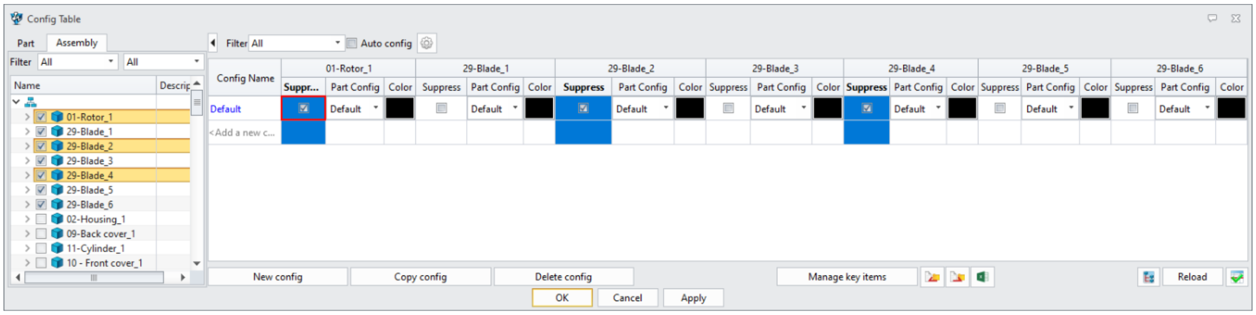
- ✓ In the Config Table, after picking multiple components in the Component Config Sheet in Assembly, you can check or uncheck the Suppress option in batches.

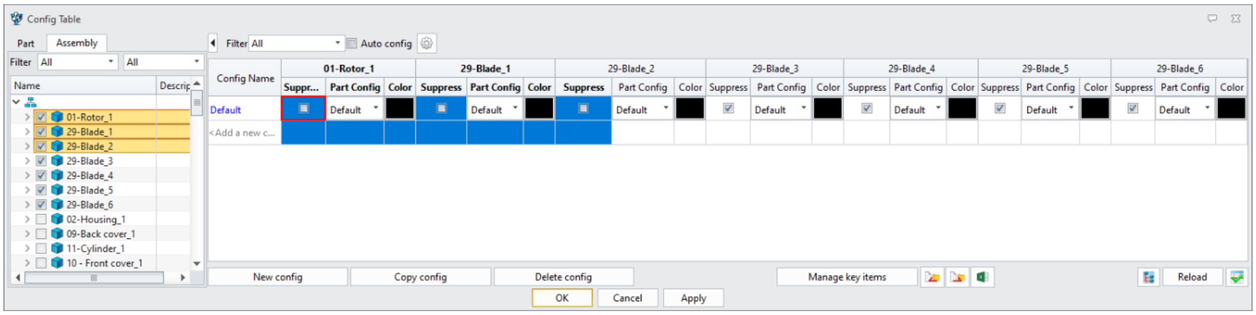
【Note】:

- After picking multiple components, you should check or uncheck the Suppress option of any selected component to perform batch operations.

【Example】:

1. Open any assembly file, run the Config Table command, and select the assembly bar.
2. Pick any multiple components, check or uncheck any picked Suppress option in the config table.


Config Table
Part
Assembly
Filter All
All
Name
Descrip
01-Rotor_1
29-Blade_1
29-Blade_2
29-Blade_3
29-Blade_4
29-Blade_5
29-Blade_6
02-Housing_1
09-Back cover_1
11-Cylinder_1
10 - Front cover_1
Filter All
Auto config
Config Name
Suppr...
Suppress
Part Config
Color
Default
<Add a new c...
New config
Copy config
Delete config
Manage key items
Reload
OK
Cancel
Apply


Config Table
Part
Assembly
Filter All
All
Name
Descrip
01-Rotor_1
29-Blade_1
29-Blade_2
29-Blade_3
29-Blade_4
29-Blade_5
29-Blade_6
02-Housing_1
09-Back cover_1
11-Cylinder_1
10 - Front cover_1
Filter All
Auto config
Config Name
Suppr...
Suppress
Part Config
Color
Default
<Add a new c...
New config
Copy config
Delete config
Manage key items
Reload
OK
Cancel
Apply

【Where is it】:

Part/Assembly Environment >> Tools >> Config Table >> Assembly

1.6.5 Pierce Transparency（ZW3D 2025 SP）

To optimize the user selection experience and make it more convenient for users to choose between transparent objects and shaded objects behind them, ZW3D 2025 SP has optimized the function of Ignore Transparency and changed it to Pierce Transparency.

【What can be done】:

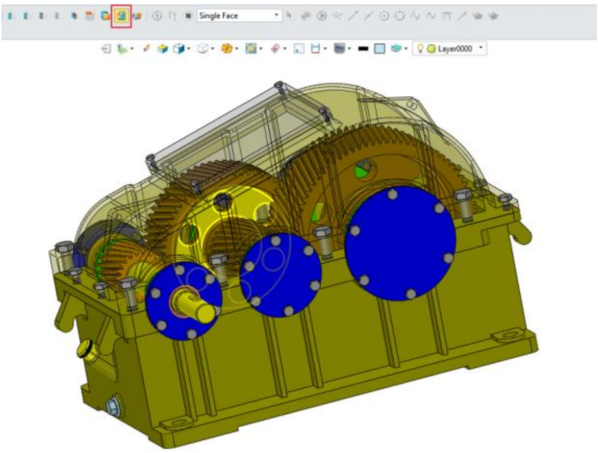
- ✓ When shaded objects are obscured by transparent objects, you can select shaded objects through the transparent objects in ZW3D 2025 SP. If there are no shaded objects behind the transparent objects, the transparent objects can be selected.

【Note】:

- The pierce transparency option is disabled by default. When editing part in assembly, the default non-active object is transparent. If the Pierce Transparency is not enabled, the shaded object behind the transparent object cannot be selected.

【Example】:

After launching the Penetrate Transparent entity control, you can select the shaded component that the transparent component blocks in the drawing area.



【Where is it】:

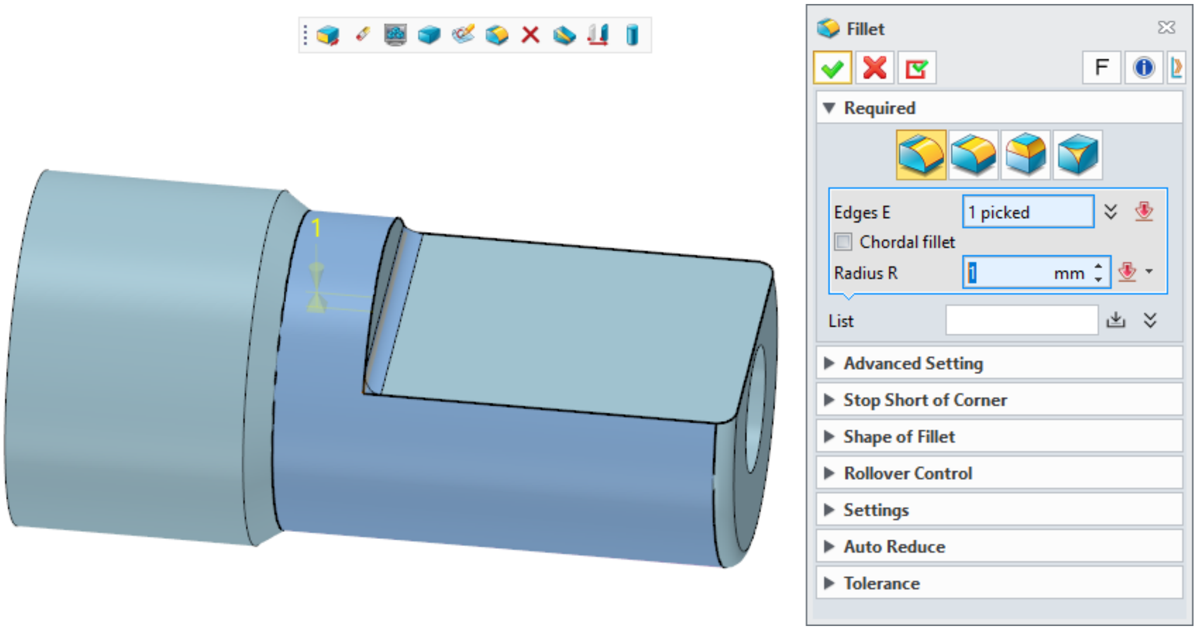
Pick Tools >> Pierce Transparency

1.7 Machine Learning-Based Command Advisor

ZW3D has a multitude of commands to implement a wide range of functions to meet different application scenarios. The enhancement of functionality increases the complexity of the software. To improve user design efficiency, ZW3D 2025 introduces a new feature, the Command Advisor. Based on a machine learning model, it recommends the next operation in real time, according to past cases and current user habits.

【Example】:

- ✓ In the following case, the Command Advisor feature, based on the historical features of the current model, provides ten recommended operations in the figure. Users can quickly perform the fillet operation.


1
Fillet
F
Required
List
Advanced Setting
Stop Short of Corner
Shape of Fillet
Rollover Control
Settings
Auto Reduce
Tolerance

Edges E: 1 picked

☐ Chordal fillet

Radius R: 1 mm

【Where is it】:

Part/Assembly >> Configuration >> General >> Show command advisor

1.8 ★Model Match Function

ZW3D 2025 provides model matching function, supporting the search of models in the path through text, models, attributes, and labels. The search results will be listed in descending order of similarity, and users can view and open models in the result list. The feature can help you quickly find the required model files in copious parts libraries.

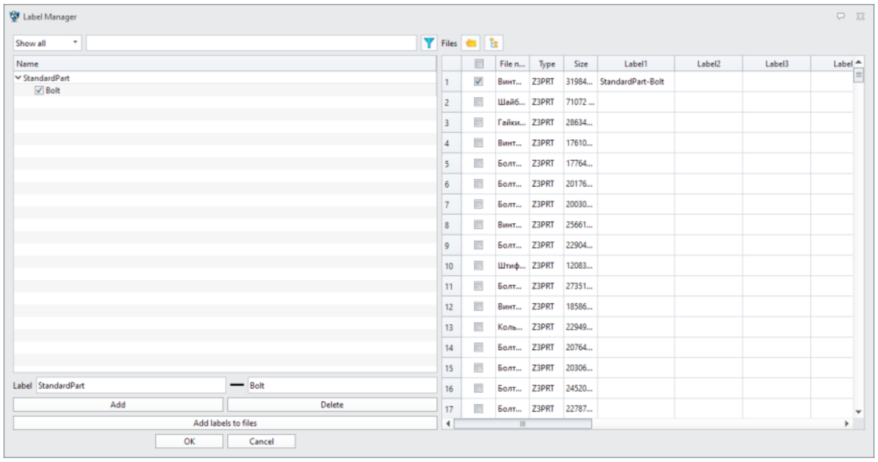
【What can be done】:

- ✓ Search for models through keywords;

- ✓ Search for models through similar models;
- ✓ Search for models through part attributes;
- ✓ Search for models through Labels.
- ✓ Composite search through two or more of the above conditions;
- ✓ Perform a second search in the result list;
- ✓ Right-click in the result list to directly open the model file.

【Note】:

- When adding a new search path, the software needs to extract file information, so there is a waiting process for information extraction; There is no need to extract information again when reusing this path in the future.
- Users can add labels to parts through the Label Manager.



Label StandardPart — Bolt

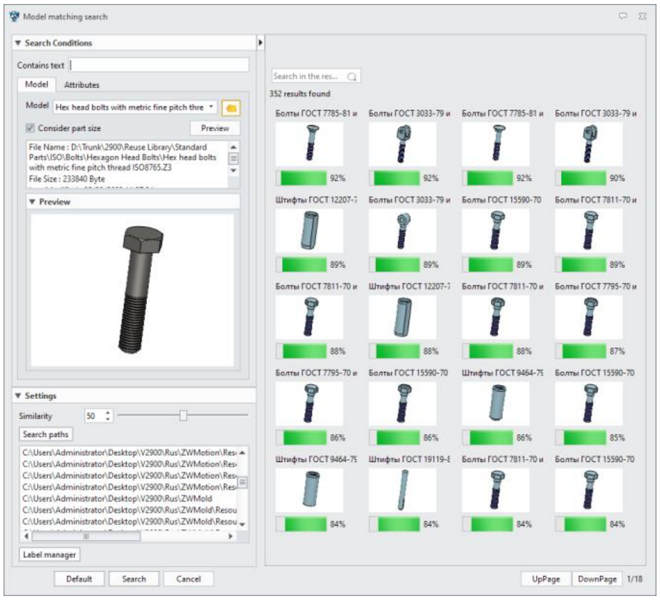
	☐	File n...	Type	Size	Label1	Label2	Label3	Label
1	☑	Винт...	Z3PRT	31984...	StandardPart-Bolt			
2	☐	Шайб...	Z3PRT	71072 ...				
3	☐	Гайки...	Z3PRT	28634...				
4	☐	Винт...	Z3PRT	17610...				
5	☐	Болт...	Z3PRT	17764...				
6	☐	Болт...	Z3PRT	20176...				
7	☐	Болт...	Z3PRT	20030...				
8	☐	Винт...	Z3PRT	25661...				
9	☐	Болт...	Z3PRT	22904...				
10	☐	Штиф...	Z3PRT	12083...				
11	☐	Болт...	Z3PRT	27351...				
12	☐	Винт...	Z3PRT	18586...				
13	☐	Коль...	Z3PRT	22949...				
14	☐	Болт...	Z3PRT	20764...				
15	☐	Болт...	Z3PRT	20306...				
16	☐	Болт...	Z3PRT	24520...				
17	☐	Болт...	Z3PRT	22787...				

- File extraction information and label information are stored in the user folder. If the user folder is not cleaned, you can reuse these two types of information when restarting the software.

【Example】:

1. Open a model in the parts/assembly environment and use it to search for models. The software will find all the parts with similarity exceeding the threshold in the search path and display them

in the result list. When the result list has more than 20 results, you can browse the following results by flipping through the pages.


Model matching search

Search Conditions

Contains text:

Model: Hex head bolts with metric fine pitch thre

☑ Consider part size

File Name : D:\Trunk\2900\Reuse Library\Standard Parts\ISO\Bolts\Hexagon Head Bolts\Hex head bolts with metric fine pitch thread ISO8765.Z3
File Size : 233840 Byte

Settings

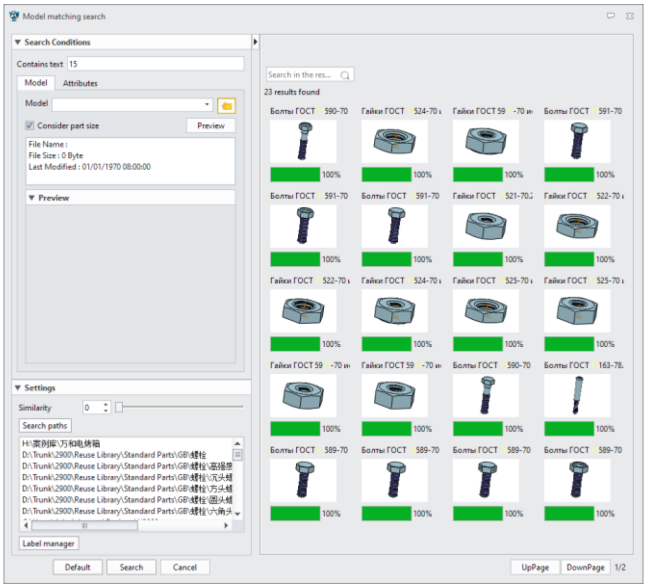
Similarity: 50

Search paths:
C:\Users\Administrator\Desktop\V2900\Rus\ZWMotion\Res
C:\Users\Administrator\Desktop\V2900\Rus\ZWMotion\Res
C:\Users\Administrator\Desktop\V2900\Rus\ZWMotion\Res
C:\Users\Administrator\Desktop\V2900\Rus\ZWMotion\Res
C:\Users\Administrator\Desktop\V2900\Rus\ZWMold
C:\Users\Administrator\Desktop\V2900\Rus\ZWMold\Resou
C:\Users\Administrator\Desktop\V2900\Rus\ZWMold\Resou

352 results found

Model
Attributes
Preview
Label manager
Default
Search
Cancel
Search in the res...
Болты ГОСТ 7785-81 и
Болты ГОСТ 3033-79 и
Болты ГОСТ 7785-81 и
Болты ГОСТ 3033-79 и
92%
92%
92%
90%
Штифты ГОСТ 12207-7
Болты ГОСТ 3033-79 и
Болты ГОСТ 15590-70
Болты ГОСТ 7811-70 и
89%
89%
89%
89%
Болты ГОСТ 7811-70 и
Штифты ГОСТ 12207-7
Болты ГОСТ 7811-70 и
Болты ГОСТ 7795-70 и
88%
88%
88%
87%
Болты ГОСТ 7795-70 и
Болты ГОСТ 15590-70
Штифты ГОСТ 9464-79
Болты ГОСТ 15590-70
86%
86%
86%
85%
Штифты ГОСТ 9464-79
Штифты ГОСТ 19119-8
Болты ГОСТ 7811-70 и
Болты ГОСТ 15590-70
84%
84%
84%
84%
UpPage
DownPage
1/18

2. By searching for input text, the software automatically displays the results containing the input text in the result list and highlights the corresponding text.


Model matching search

Search Conditions

Contains text: 15

Model:

☑ Consider part size

File Name :
File Size : 0 Byte
Last Modified : 01/01/1970 08:00:00

Settings

Similarity: 0

Search paths:
H:\案例库\万和电烤箱
D:\Trunk\2900\Reuse Library\Standard Parts\GB\螺栓
D:\Trunk\2900\Reuse Library\Standard Parts\GB\螺栓\高强度
D:\Trunk\2900\Reuse Library\Standard Parts\GB\螺栓\沉头螺
D:\Trunk\2900\Reuse Library\Standard Parts\GB\螺栓\方头螺
D:\Trunk\2900\Reuse Library\Standard Parts\GB\螺栓\圆头螺
D:\Trunk\2900\Reuse Library\Standard Parts\GB\螺栓\六角头

23 results found

Model
Attributes
Preview
Label manager
Default
Search
Cancel
Search in the res...
Болты ГОСТ 590-70
Гайки ГОСТ 524-70
Гайки ГОСТ 59 -70
Болты ГОСТ 591-70
Болты ГОСТ 591-70
Болты ГОСТ 591-70
Гайки ГОСТ 521-70
Гайки ГОСТ 522-70
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Гайки ГОСТ 59 -70
Гайки ГОСТ 59 -70
Болты ГОСТ 590-70
Болты ГОСТ 163-78
Болты ГОСТ 589-70
Болты ГОСТ 589-70
Болты ГОСТ 589-70
Болты ГОСТ 589-70
100%
UpPage
DownPage
1/2

【Where it is】

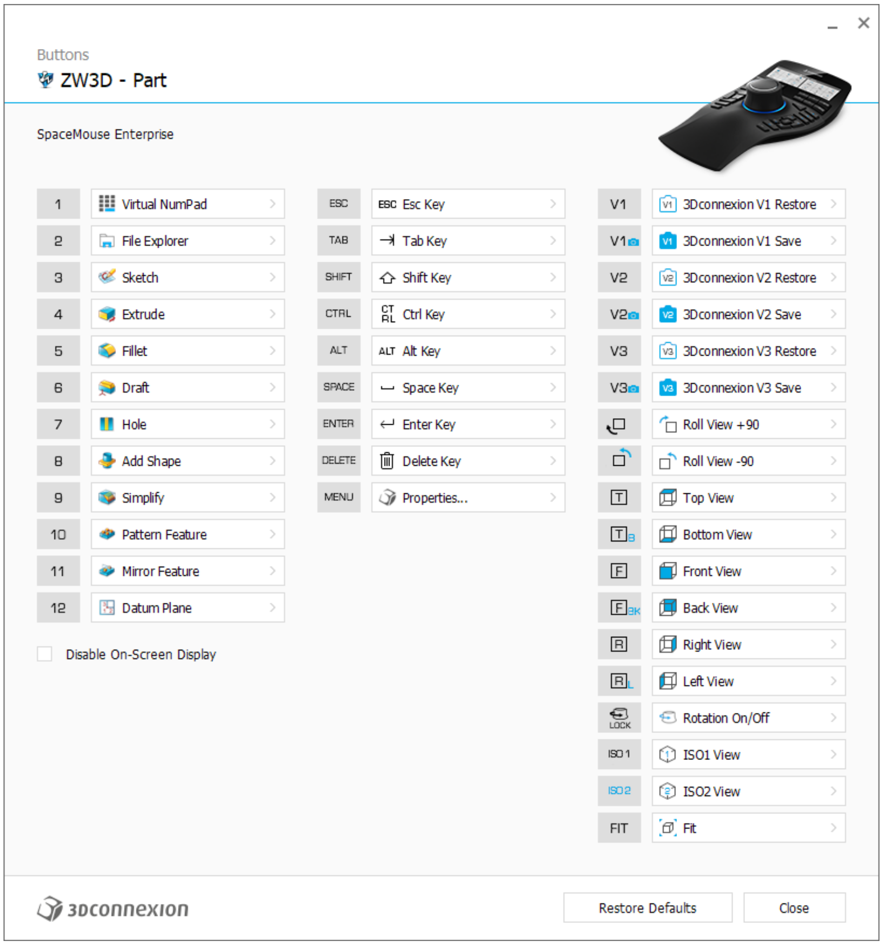
Part/Assembly Environment >> App >> Part Library >> Model Match

1.9 3D Mouse Support Update (ZW3D 2025 SP)

ZW3D 2025 SP has upgraded its support for 3Dconnexion SpaceMouse, allowing users to utilize more SpaceMouse functions to manipulate 3D models, making 3D design easier and more efficient..

【What can be done】:

- ✓ Comprehensive support for rotating, translating, and scaling models with a 3Dconnexion SpaceMouse.
- ✓ Comprehensive support for shortcuts of 3Dconnexion SpaceMouse.



1	Virtual NumPad	ESC	Esc Key	V1	3Dconnexion V1 Restore
2	File Explorer	TAB	Tab Key	V1	3Dconnexion V1 Save
3	Sketch	SHIFT	Shift Key	V2	3Dconnexion V2 Restore
4	Extrude	CTRL	Ctrl Key	V2	3Dconnexion V2 Save
5	Fillet	ALT	Alt Key	V3	3Dconnexion V3 Restore
6	Draft	SPACE	Space Key	V3	3Dconnexion V3 Save
7	Hole	ENTER	Enter Key		Roll View +90
8	Add Shape	DELETE	Delete Key		Roll View -90
9	Simplify	MENU	Properties...	T	Top View
10	Pattern Feature			TB	Bottom View
11	Mirror Feature			F	Front View
12	Datum Plane			FBK	Back View
				R	Right View
				RL	Left View
				LOCK	Rotation On/Off
				ISO1	ISO1 View
				ISO2	ISO2 View
				FIT	Fit

☐ Disable On-Screen Display

【Where is it】:

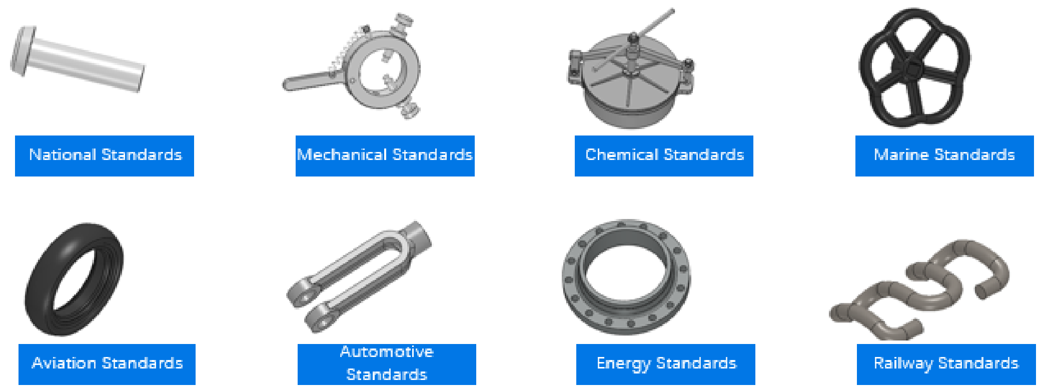
For detailed information, please refer to the function description of the 3Dconnexion SpaceMouse.

1.10 Add Design Library Function (ZW3D 2025 SP)

When designing large product equipment, engineers often use off-the-shelf or standard parts to express the product structure and generate a bill of materials for procurement. The demand for standard parts varies greatly across different industries, and engineers often spend a lot of time selecting and downloading standard parts, which is time-consuming and labor-intensive. ZW3D 2025 SP has added a design library function, which supports the download and selection of standard parts for various industries and can meet the design needs in such scenarios.

【What can be done】:

- ✓ Support free download and selection of standard parts for various industries, including eight major industry standards: national standards, mechanical, chemical, marine, aviation, automotive, energy, and railway.
- ✓ The source library supports downloading resources in various ways (from the ZW3D official website or public resource libraries) to the local library, as well as direct referencing of data in the local resource library.

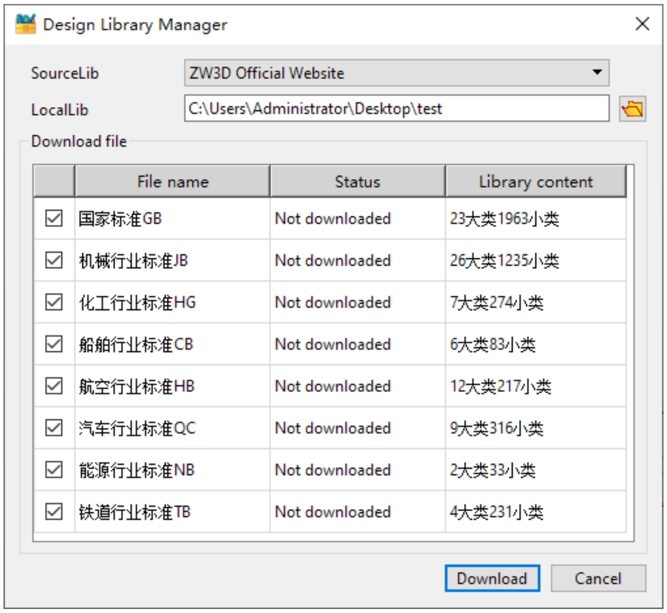

National Standards
Mechanical Standards
Chemical Standards
Marine Standards
Aviation Standards
Automotive Standards
Energy Standards
Railway Standards

【Note】:

- Industry standard library files are large and may take a long time to download. You can click to cancel the interruption download or select a single industry standard library to download.

【Example】:

Open the Design Library function. Select the local storage path of the library, check the libraries you need to download, and click Download. After the download is complete, you can view and use it in the Design Library of the reuse library.



SourceLib: ZW3D Official Website

LocalLib: C:\Users\Administrator\Desktop\test

	File name	Status	Library content
☑	国家标准GB	Not downloaded	23大类1963小类
☑	机械行业标准JB	Not downloaded	26大类1235小类
☑	化工行业标准HG	Not downloaded	7大类274小类
☑	船舶行业标准CB	Not downloaded	6大类83小类
☑	航空行业标准HB	Not downloaded	12大类217小类
☑	汽车行业标准QC	Not downloaded	9大类316小类
☑	能源行业标准NB	Not downloaded	2大类33小类
☑	铁道行业标准TB	Not downloaded	4大类231小类

【Where is it】:

APP Environment >> Part Library >> Design Library

Reuse Library >> Design Library

2 Translator

2.1 ★Input Format Upgrade（ZW3D 2025 SP）

We have upgraded default and alternative translators in ZW3D 2025 SP to support the updated input format versions. The following table lists out the input formats and their versions supported in ZW3D 2025, and the current updated formats are marked in red.

No.	Product	File Extension Name	Default Translator	Alternative Translator
1	Catia V4	.model, .exp, .session	4.1.9 - 4.2.4	4.1.9 - 4.2.4
2	Catia V5	.CATPart, .CATProduct, .CGR	V5R10 --- V5-6R2024	V5R8---V5-6R2024
3	Catia V5_2D	.CATDrawing	V5R10 --- V5-6R2024	V5R8---V5-6R2024
4	3DExperience (CATIA V6)	.CATPart, .CATProduct	R2010x - R2024x	Exported from V6 R2024x and previous version
5	CGR	.cgr	Exported from V6 R2024x and previous version	Exported from V6 R2024x and previous version
6	NX(UG)	.prt	11 - NX 2306	11 - NX 2312
7	Creo(Pro/E)	.prt, .prt.*, .asm, .asm.*	16 - Creo 11.0	16 - Creo 11.0
8	Creo(Pro/E)_2D	.drw .drw.*	2000i - Creo 11.0	/
9	SolidWorks	.sldprt, .sldasm	98 - 2024(only support 64-bit)	98 - 2024(only support 64-bit)

10	SolidWorks_2D	.slddrw	2004 - 2024	2013-2024(only support 64-bit)
11	SolidEdge	.par, .asm, .psm	10 - 2024	18 - 2024
12	Inventor	.ipt, .iam	9 to 2025	.ipt (6 - 2025) .iam (11 - 2025)
13	ACIS	.sat, .sab, .asat, .asab	R1 - 2024 1.0	R1 - 2024 1.0
14	DWG	.dwg	R11 -	R11 -
15	DXF	.dxf	R11 -	R11 -
16	IGES	.ige, .iges		
17	STEP	.stp, .step, .stpz	203, 214, 242	203, 214, 242
18	Parasolid	.x_t, .x_b, .xmt_txt, .xmt_bin	Up to 30.0	Up to 30.0
19	VDA	.vda		
20	Image File	.bmp, .gif, .jpg, .jpeg, .tif, .tiff		
21	Neutral File	.z3n, .vxn		
22	PartSolutions	.ps2, .ps3		
23	STL	.stl		
24	3DXML	.3dxml	4.0 - 4.3	4.0 - 4.3
25	JT	.jt	6.4-10.4	6.4-10.4

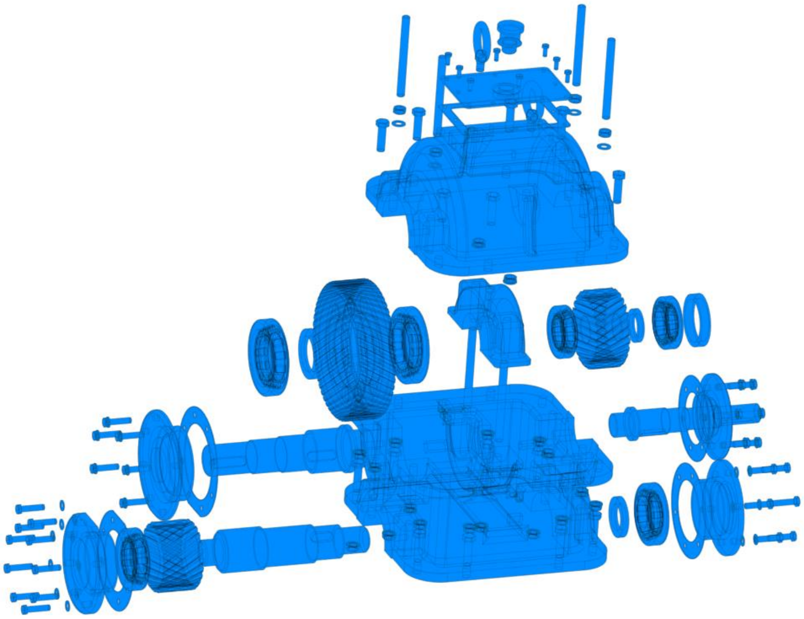
26	OBJ	.obj		
27	Rhino	.3dm	version 2 - 7	version 2 - 7

2.2 Support SVG Format Export

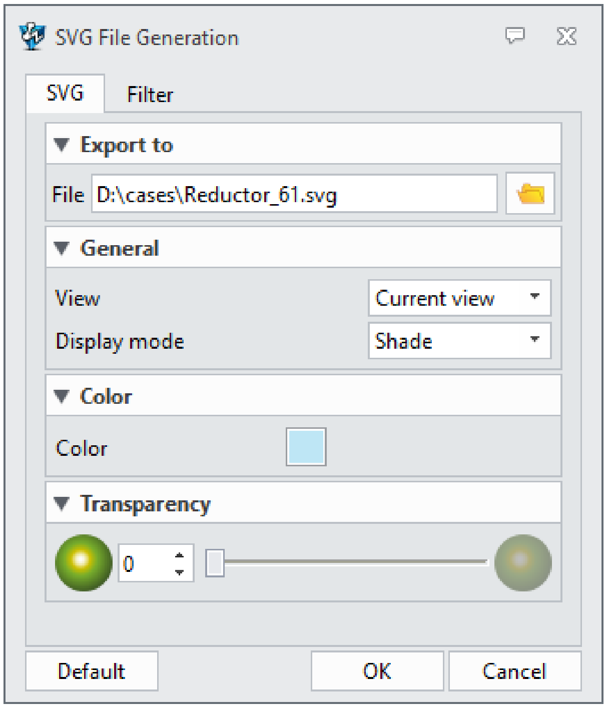
The SVG format describes two-dimensional vector graphics, which supports high quality display when scaling, and is often used in the maintenance manuals of automobile companies to show customers the mechanical structure of the exploded view. ZW3D 2025 supports SVG format export to meet the above usage scenarios.

【What can be done】:

- ✓ The exported SVG can be opened by the commonly used browser and supports highlighting the parts where the mouse hovers.



- ✓ Support selecting views and display modes.
- ✓ Support setting transparency and color.



SVG File Generation

Export to

File D:\cases\Reductor_61.svg

General

View: Current view

Display mode: Shade

Color

Transparency: 0

SVG
Filter
Color
Default
OK
Cancel

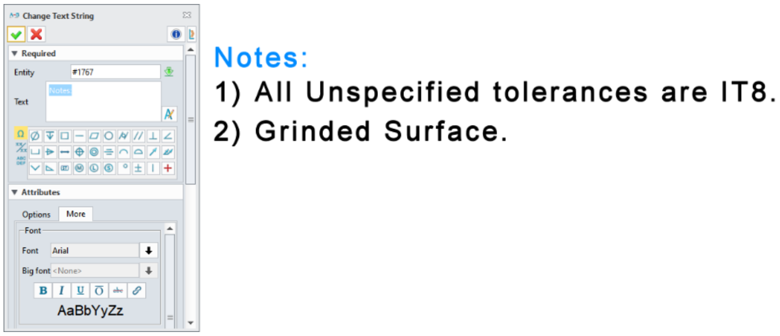
【Where is it】:

Data Exchange Ribbon >> Export >> Export >> SVG File

2.3 Improve Reusability of the Third Party File Data

2.3.1 Support Re-editing Imported Drawing Sheet Text

ZW3D 2025 can help you import text objects, re-edit text contents, and read as well as modify fonts and other styles. The following shows the technical requirements that need to be redefined in the imported SolidWorks engineering sheet.



Change Text String

Entity: #1767

Text: Notes:

Font: Arial

Big font: <None>

Notes:
1) All Unspecified tolerances are IT8.
2) Grinded Surface.

Required
Attributes
Options
More
Font
AaBbYyZz

【What can be done】:

- ✓ Text in third-party drawing sheet can be redefined。

【Where is it】：

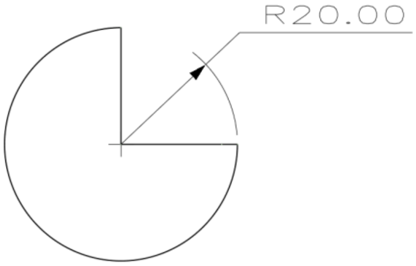
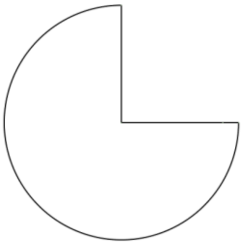
File >> Import >> Third-Party Drawing Sheet

2.3.2 Keep Curve Analyticity

ZW3D 2025 can help you import arcs and ellipses of the third-party drawing sheets as analytic curves and dimension diameters to save more accurate geometric information.

【What can be done】

- ✓ Arc and ellipse object in the drawing sheets of the third party can be dimensioned twice.

ZW3D 2025	ZW3D 2024
	
Support the secondary dimension	Do not support the secondary dimension

【Where is it】：

File >> Import >> Third-Party Drawing Sheet

2.3.3 Support Import Non-Default Coordinate

ZW3D 2025 supports importing the non-default coordinates of 3D files from the third-party as CSYS history features to save more geometric information and improve data reusability of the third-party files.

【What can be done】：

- ✓ Import the non-default coordinates of 3D files from the third-party.

【Where is it】：

File >> Import >> Third-Party 3D File

2.4 STEP Export Enhancement

2.4.1 Reusable Data and Decrease File Size

The same surface data can be reused in ZW3D 2025 to reduce the size of exported STEP files.

【What can be done】:

- ✓ Reduce STEP file volume (Only valid for the model referencing the same surface).

【Where is it】:

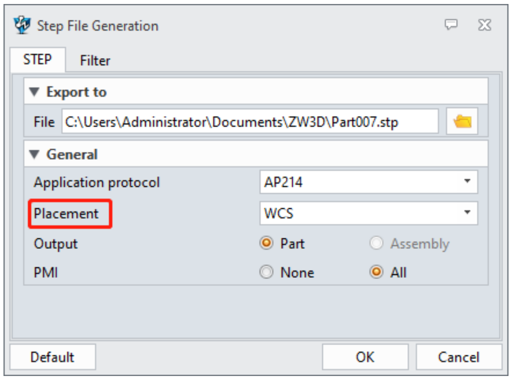
Part/Assembly>>File >> Export >> STEP Format

2.4.2 Support Modifying Export Coordinates

ZW3D 2025 has added “Placement” to the export STEP panel and supports selecting the coordinate system in the feature tree as the export coordinate system of STEP files, so that the geometric model can be located more flexibly.

【What can be done】

- ✓ STEP file coordinates can be changed.



File C:\Users\Administrator\Documents\ZW3D\Part007.stp

Application protocol AP214

Placement WCS

Output ● Part ○ Assembly

PMI ○ None ● All

【Where is it】:

Part/Assembly>> File >> Export >> STEP Format >> Placement

2.5 ★Batch Export Function Optimization

2.5.1 Support Export to Source File Path

ZW3D 2025 supports saving the batch file to the path of source file. Two path settings are supported to meet different archiving requirements and file storage rules.

【What can be done】

- ✓ The files exported in batches are saved to the path of the source file or the same file.

【Where is it】：

Drawing Sheet Environment >> File >> Multi-Export >> Export Settings >> Destination

2.5.2 Support All Drawing Sheets Merging to PDF

ZW3D 2025 supports merging the batch export files into one PDF, helping users to get all printed drawing sheets in one print.

【What can be done】

- ✓ Print all drawing sheets at a time.

【Where is it】：

Drawing Sheet Environment >> File>>Multi-Export >> Export Settings >> PDF Settings>> Export Sheet

2.5.3 Support stitch exporting drawings as DWG/DXF files (ZW3D 2025 SP)

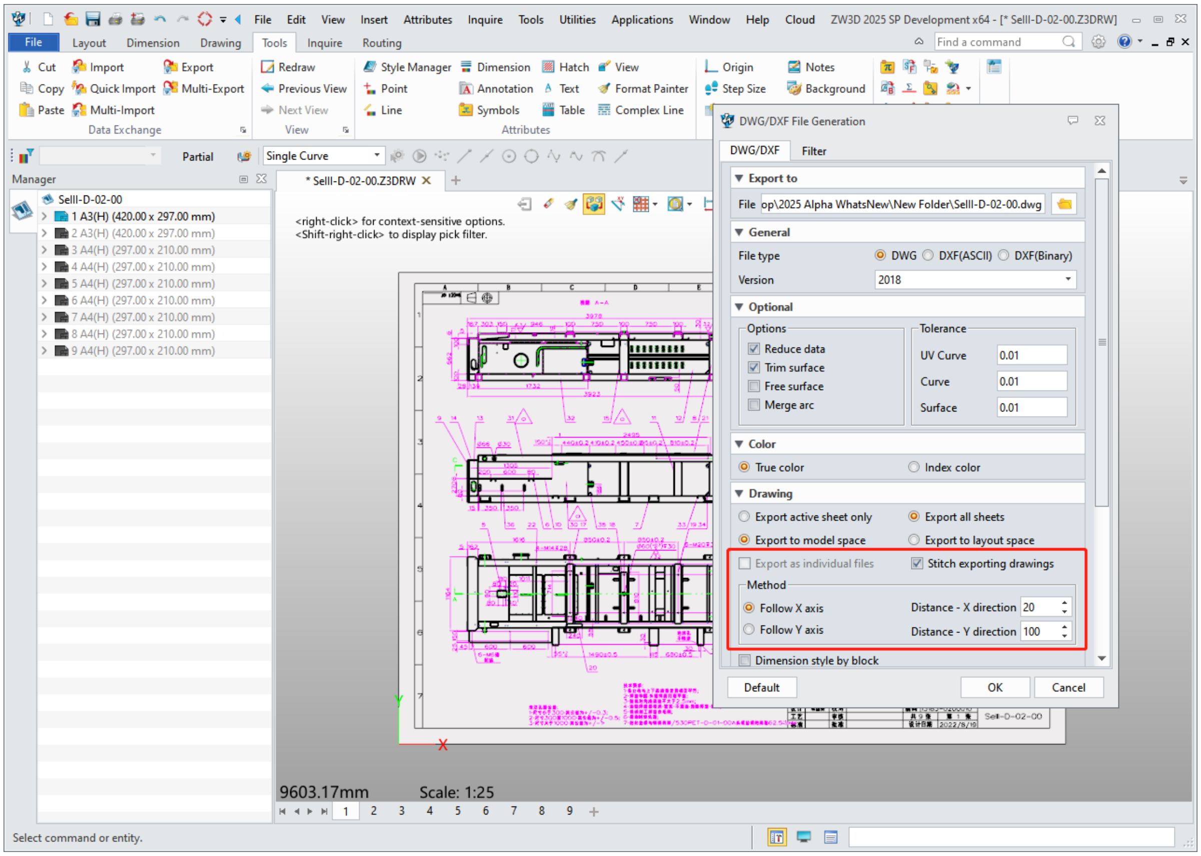
Industries such as connectors and molds commonly stitch together drawing sheets on the same sheet for export. ZW3D 2025 SP provides a stitch export function to meet such requirement.

【What can be done】：

- ✓ Support splicing and exporting a single Z3DRW file containing multiple drawings into DWG/DXF files in model space.
- ✓ Support arranging according to the set distance in the X or Y direction.

【Example】：

Open a drawing file containing multiple sheets, select export as DWG/DXF file, select stitch drawings, set the layout of the drawings along the X direction with a spacing of 20, and click OK.


File
Edit
View
Insert
Attributes
Inquire
Tools
Utilities
Applications
Window
Help
Cloud
ZW3D 2025 SP Development x64 - [* SellI-D-02-00.Z3DRW]
Layout
Dimension
Drawing
Tools
Inquire
Routing
Cut
Copy
Paste
Import
Quick Import
Multi-Import
Export
Multi-Export
Data Exchange
Redraw
Previous View
Next View
View
Style Manager
Point
Line
Dimension
Annotation
Symbols
Hatch
Text
Table
View
Format Painter
Complex Line
Attributes
Origin
Step Size
Notes
Background
Find a command
Partial
Single Curve
Manager
SellI-D-02-00
1 A3(H) (420.00 x 297.00 mm)
2 A3(H) (420.00 x 297.00 mm)
3 A4(H) (297.00 x 210.00 mm)
4 A4(H) (297.00 x 210.00 mm)
5 A4(H) (297.00 x 210.00 mm)
6 A4(H) (297.00 x 210.00 mm)
7 A4(H) (297.00 x 210.00 mm)
8 A4(H) (297.00 x 210.00 mm)
9 A4(H) (297.00 x 210.00 mm)
* SellI-D-02-00.Z3DRW
<right-click> for context-sensitive options.
<Shift-right-click> to display pick filter.
9603.17mm
Scale: 1:25
1 2 3 4 5 6 7 8 9
Select command or entity.
DWG/DXF File Generation
DWG/DXF
Filter
Default
OK
Cancel

DWG/DXF File Generation

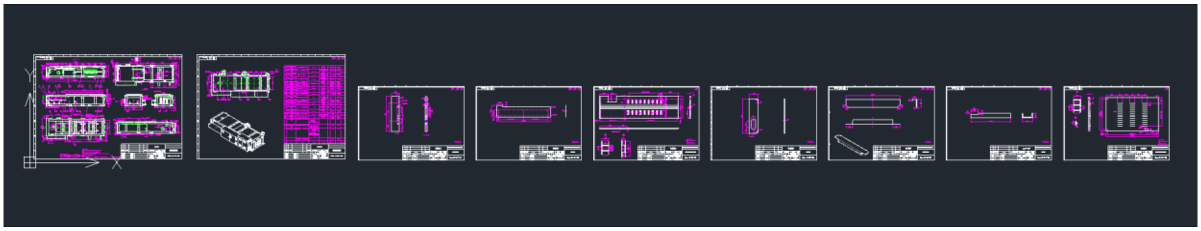
Export to
File op\2025 Alpha WhatsNew\New Folder\SellI-D-02-00.dwg

General
File type ● DWG ○ DXF(ASCII) ○ DXF(Binary)
Version 2018

Optional
Options
☑ Reduce data
☑ Trim surface
☐ Free surface
☐ Merge arc
Tolerance
UV Curve 0.01
Curve 0.01
Surface 0.01

Color
● True color ○ Index color

Drawing
○ Export active sheet only ● Export all sheets
● Export to model space ○ Export to layout space
☐ Export as individual files ☑ Stitch exporting drawings
Method
● Follow X axis Distance - X direction 20
○ Follow Y axis Distance - Y direction 100
☐ Dimension style by block



【Where is it】：

Data Exchange Ribbon >> Export >> Export >> DWG/DXF File

3 CAD

3.1 Sketch & Wireframe Design

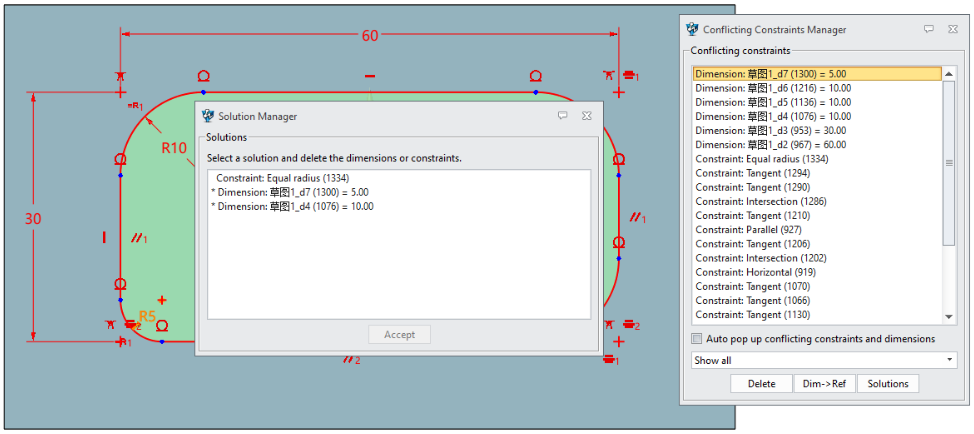
We greatly improve the ability of processing over-constraints in the basic constraint. Smart pick supports selecting point to add constraints. Besides, we increase the command response efficiency when there are multiple references in the large sketch. Moreover, ellipse attributes can be maintained after the ellipse is trimmed. Please check the following chapters and the help documentation for other improvements and detailed descriptions.

3.1.1 ★Sketch Over-defined Process Improvement

In sketch design, when over-constraints occur to the sketch, users must stop drawing and check the constraints step by step before continuing, which reduces productivity. In ZW3D 2025, users are provided with feasible solutions to over-constraints only by clicking Solution. The quick solution to over-constraints helps users draw sketches faster.

【What can be done】：

- ✓ Provide a quick solution when sketch is over-defined.



Select a solution and delete the dimensions or constraints.

Constraint: Equal radius (1334)
* Dimension: 草图1_d7 (1300) = 5.00
* Dimension: 草图1_d4 (1076) = 10.00

Dimension: 草图1_d7 (1300) = 5.00
Dimension: 草图1_d6 (1216) = 10.00
Dimension: 草图1_d5 (1136) = 10.00
Dimension: 草图1_d4 (1076) = 10.00
Dimension: 草图1_d3 (953) = 30.00
Dimension: 草图1_d2 (967) = 60.00
Constraint: Equal radius (1334)
Constraint: Tangent (1294)
Constraint: Tangent (1290)
Constraint: Intersection (1286)
Constraint: Tangent (1210)
Constraint: Parallel (927)
Constraint: Tangent (1206)
Constraint: Intersection (1202)
Constraint: Horizontal (919)
Constraint: Tangent (1070)
Constraint: Tangent (1066)
Constraint: Tangent (1130)

☐ Auto pop up conflicting constraints and dimensions

【Notes】：

- When checking sketch over-constraints, all associated constraints will be recognized. When there are copious constraint groups, multiple solutions to over-constraints will be provided. You can only focus on the solutions ahead of the list.

- Automatically pops up when checking Configuration—2D—Sketch—Enable 2D Constraint Solver.

【Where is it】:

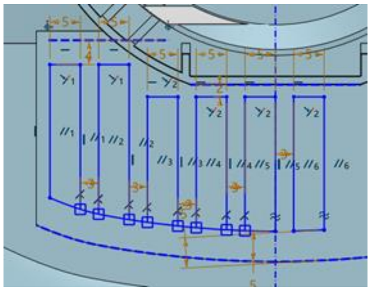
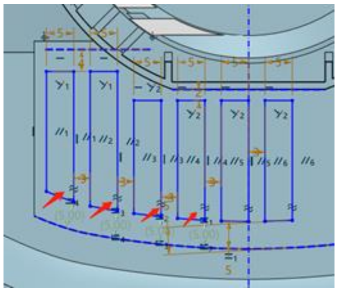
Configuration >> 2D >> Sketch >> Enable 2D Constraint Solver

3.1.2 Keep Constraints after Sketch Trimming

In sketch design, sometimes the derived curve needs to be trimmed after creating offsets or patterns. In this case, users want to keep the constraints to maintain the trimmed curve position.

【What can be done】:

✓ Maintain offset constraints and pattern constraints after the derived curve being trimmed.

Before trimming in ZW3D 2025	After trimming in ZW3D 2025
	

【Note】:

- Directly trim the original curve will change the offset result.

【Where is it】:

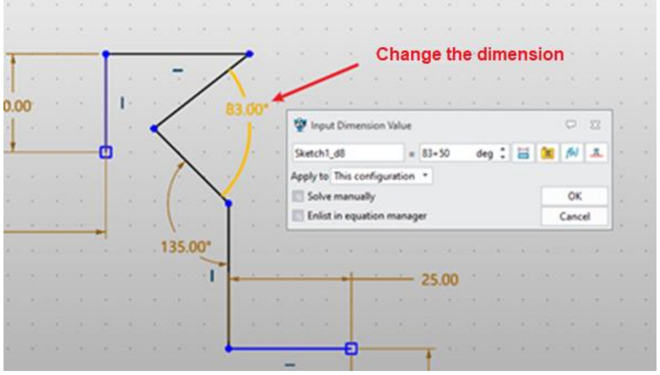
Sketch >> Edit Curve

3.1.3 Angle Dimension Improvement

When users create angular dimension, the expected dimensioned angle is the actual angle, not the supplementary angle. The improved ZW3D 2025, the consistent dimension improves user experience.

【What can be done】:

- ✓ The angular dimension value is consistent with the result (In the figure below, 83 degrees is changed to 133 degrees, and the actual angle is 133 degrees, not 47 degrees).



Change the dimension

Input Dimension Value

Sketch1_d8 = 83+50 deg

Apply to This configuration

☐ Solve manually

☐ Enlist in equation manager

【Where is it】:

Sketch >> Dimension >> Angular Dimension

3.1.4 Directly Pick Feature Point and External Geometry to Create Constraint

When adding constraints during sketch drawing, users are used to selecting the corresponding geometry, choosing what they need from the pop-up constraints, and adding it quickly. ZW3D 2025's new feature of creating constraints from feature points and external geometries helps users shorten the constraint path, increasing the efficiency of creating constraints.

【What can be done】:

- ✓ Support adding constraints to the feature points (such as end points, circle centers, spline control points, spline through points) or the external geometries (and any other primitives that you want to constrain).

【Note】:

- Do not support selecting the middle points to add constraints.

【Where is it】:

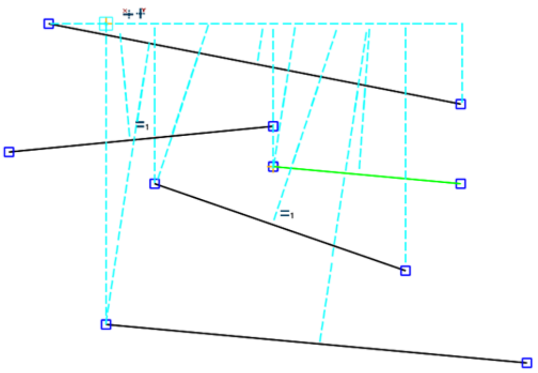
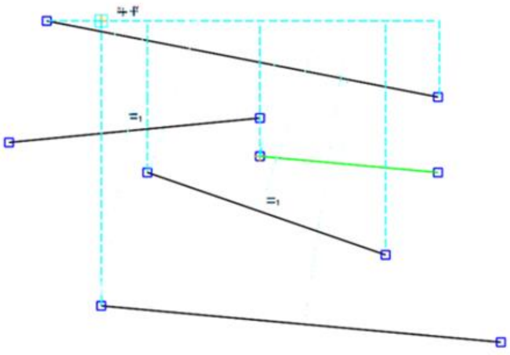
Sketch >>Constraint >> Add Constraints

3.1.5 Smart Capture Optimization

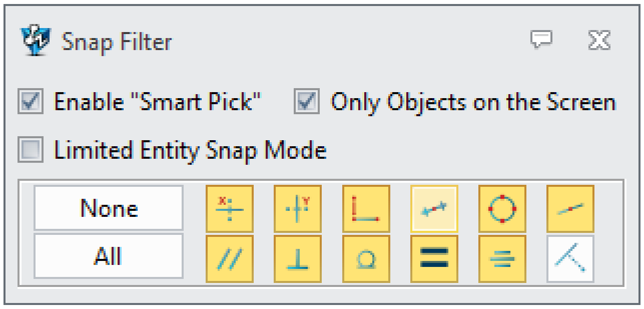
When users draw sketch, too many snap lines will make the sketch less concise, and users cannot quickly place the cursor at any position during drawing process. In the improved ZW3D 2025, vertically align is closed by default, which can greatly reduce the snapping of aligned lines during drawing. When there are too many sketch primitives, users can turn on the new Limited Entity Snap Mode to avoid the interference of irrelevant items and facilitate the user’s drawing.

【What can be done】：

✓ Reduce the quantity of snap lines and decrease interference of irrelevant items during drawing.

Vertically align is enabled in ZW3D 2024 by default	**Vertically align is disabled in ZW3D 2025 by default**
	

✓ The new drawing mode does not capture any object by default. Only when the target is touched can it be snapped.



Snap Filter

☑ Enable "Smart Pick" ☑ Only Objects on the Screen

☐ Limited Entity Snap Mode

【Note】：

- Vertically align can be reactivated by manual.

【Where is it】：

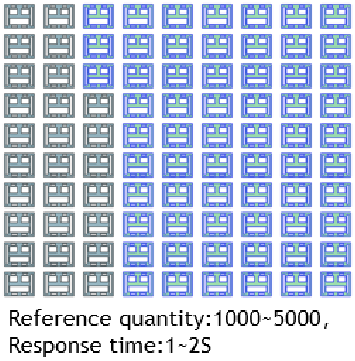
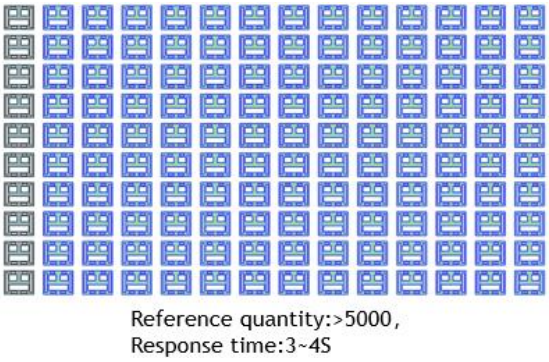
Sketch >> DA Toolbar >> Snap Filter >> Limited Entity Snap Mode

3.1.6 Large Sketch Efficiency Improvement

Some users need to smoothly create thousands or even ten-thousands of references in a single sketch. ZW3D 2025 has greatly increased the sketch drawing productivity by improving large sketch's reference and shade ring efficiency.

【What can be done】：

✓ It only takes 3-4s to continue creating references under 5,000 reference lines.

Respond in 1-2s with 1k-5k sketch references	Respond in 3-4s with over 5k sketch references
 Reference quantity:1000~5000, Response time:1~2S	 Reference quantity:>5000, Response time:3~4S

✓ The response time of turning on sketch ring shade is reduced by 5 times.

ZW3D 2024	ZW3D 2025
Respond in over 25s with 1,500 closed rectangle shades when continue drawing	**Respond in less than 5s with 1,500 closed rectangle shades when continue drawing**

【Note】：

- It has only increased the efficiency of creating references within the sketch. The regeneration efficiency of sketch with a large amount of reference lines is pending to improve.

【Where is it】：

Reference: Sketch >> Reference >> Reference

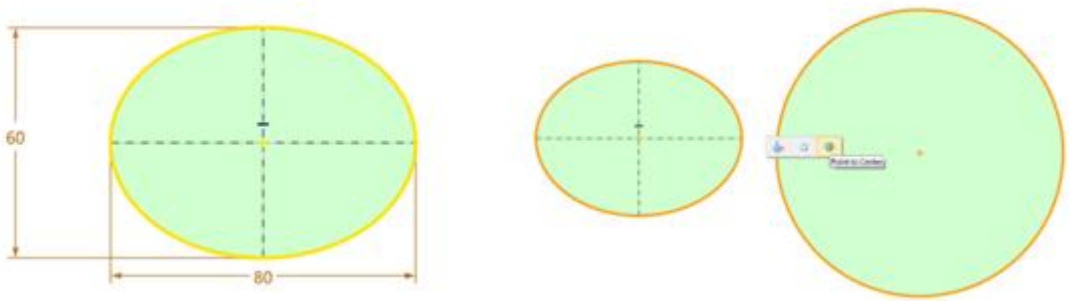
Open Closed Ring: Sketch >> DA Toolbar >> Closed Rings On/Off

3.1.7 Ellipse Analytical Expression Improvement

There are some advanced curve shapes commonly used in ellipse. The ellipse analytical expressions can make an ellipse maintain ellipse features even after trimming and patterning, so that the ellipse can still be constrained.

【What can be done】：

- ✓ After trimming, moving, rotating, mirroring, or patterning an ellipse, the derived curve is still an ellipse which can be applied concentric constraints.



【Note】：

- The ellipse undergone analytical expression does not support radius and perimeter dimensions.

【Where is it】：

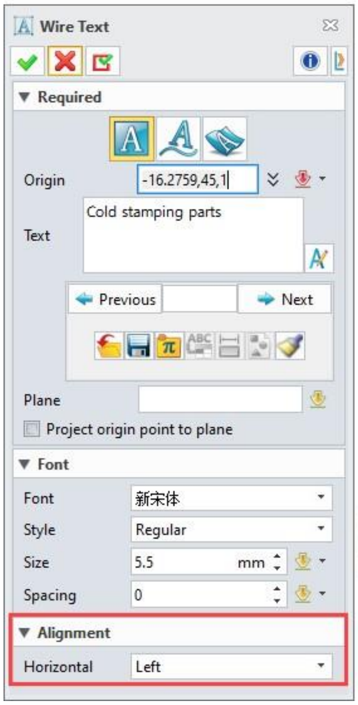
Sketch >> Drawing >> Ellipse

3.1.8 Wireframe Text Optimization（ZW3D 2025 SP）

ZW3D 2025 SP optimizes the alignment of wireframe text by adding three new horizontal left, middle and right alignment methods, which provides faster and more convenient text alignment and improves the efficiency of users in creating wireframe text.

【What can be done】:

- ✓ Quickly align text horizontally left, center and right


Wire Text
Required
Previous
Next
Font
Alignment

Origin: -16.2759,45,1

Text: Cold stamping parts

Plane:

☐ Project origin point to plane

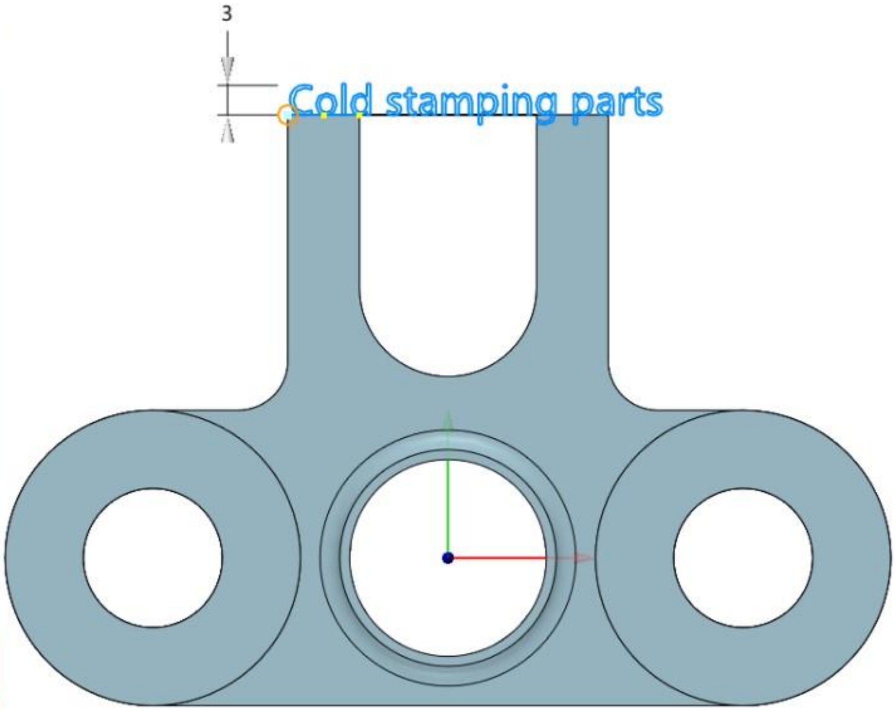
Font: 新宋体

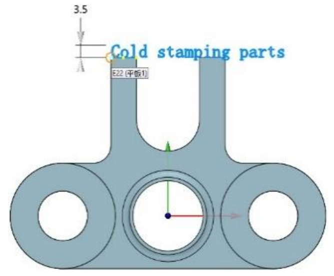
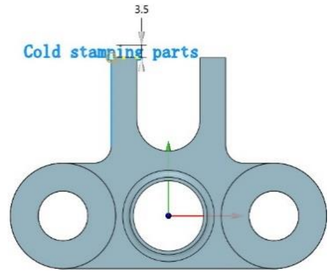
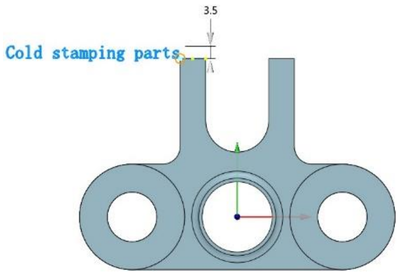
Style: Regular

Size: 5.5 mm

Spacing: 0

Horizontal: Left


3
Cold stamping parts

Left	Center	Right
		

【Where is it】:

Part/Assembly >> Wireframe >> Wire Text

3.2 Shape Design

3.2.1 ★New “To Next” Extrude Function

ZW3D 2025 has added the “To next” option for terminal faces to the start and end methods in the extrude feature. Instead of picking any face by manual, the function can automatically find and pick the nearest entity or surface to the sketch as the terminal face of extrusion, which enables simpler operations. This function is compatible with the existing bool, draft, offset, convert, and profile gap.

【What can be done】:

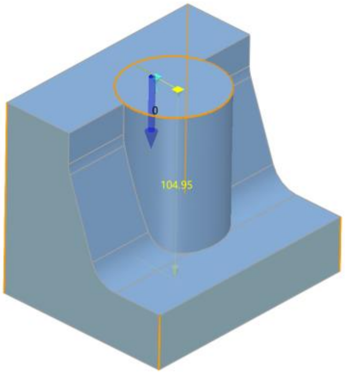
- ✓ Find entity or surface as terminal face.
- ✓ The corresponding terminal face can still be found even after adding draft, offsetting, and converting (twisting).
- ✓ Supports hybrid modeling, such as stretching solid and surface terminating surfaces for Boolean operations.

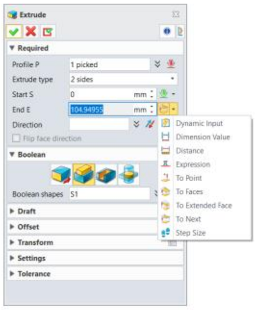
【Notes】:

- The terminal surface cannot be pieced together by multiple shapes, and it must be a single shape.
- After patterning or mirroring extrude features, it does not support finding the terminal surface based on pattern position changes currently.

【Example】:

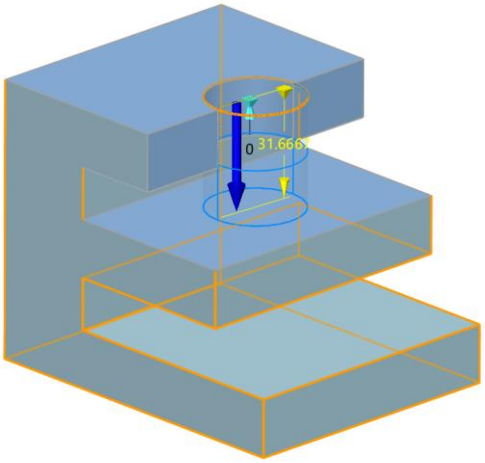
1. An extruded boss that attaches to and ends at the side wall.

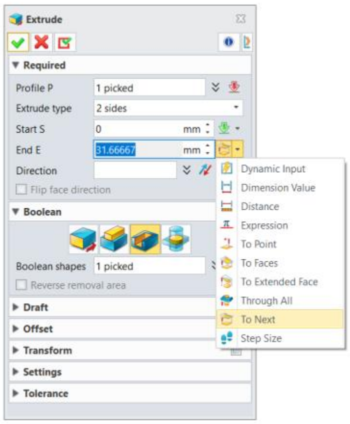



Extrude
Required
Boolean
Draft
Offset
Transform
Settings
Tolerance
Dynamic Input
Dimension Value
Distance
Expression
To Point
To Faces
To Extended Face
To Next
Step Size

Profile P: 1 picked
Extrude type: 2 sides
Start S: 0 mm
End E: 104.94955 mm
Direction:
☐ Flip face direction
Boolean shapes: S1

2. The extrude cut that terminates to the next surface.




Extrude
Required
Boolean
Dynamic Input
Dimension Value
Distance
Expression
To Point
To Faces
To Extended Face
Through All
To Next
Step Size
Draft
Offset
Transform
Settings
Tolerance

Profile P: 1 picked
Extrude type: 2 sides
Start S: 0 mm
End E: 31.66667 mm
Direction:
☐ Flip face direction
Boolean shapes: 1 picked
☐ Reverse removal area

【Where is it】:

Shape>>Basic Shape>>Extrude

3.2.2 Spiral Sweep Function Improvement

In ZW3D 2025, the Spiral Sweep function now can define two out of the three parameters, including total height, pitch, and revolution of the spiral sweep body. It also supports the definition of variable pitch through various forms, such as linear, 2 degrees, 3 degrees, etc. The sweep radius of a spiral sweep can now be set to linear change or controlled by a law curve. In addition, the UI and ease of use of the features have also been improved. The anti-spiral direction, taper and other functions have been integrated into the rotation axis direction switching and radius control to provide a simpler UI operation interface.

【What can be done】:

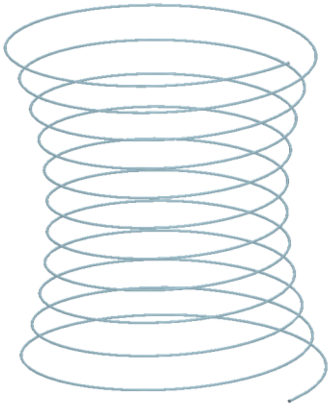
- ✓ You can define spiral sweep features through pitch and revolution, pitch and height, as well as height and revolution.
- ✓ You can change the spiral sweep radius by drawing a 2D sketch.
- ✓ In the Spiral Sweep command panel, the in-place profile sketch is supported, and all in-place sketch will be built in the spiral sweep features to simplify the display of the feature tree.

【Note】:

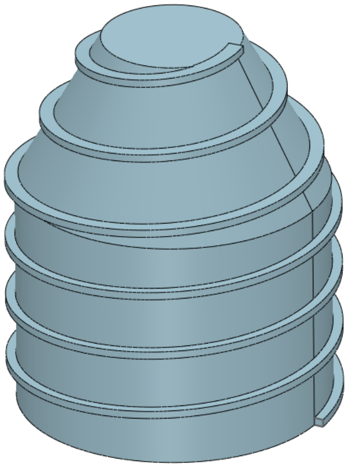
- Currently, control curves of sweep profile and radius variable do not support 3D sketches.

【Example】：

1. Draw a variable radius spring of a circle outline.



2. Generate a thread whose radius changes according to the profile of the rotating body.



【Where is it】：

Shape>>Basic Shape>>Spiral Sweep

3.2.3 Pattern Function and Efficiency Improvement

The efficiency of pattern features is increased by 30% on average. Special optimization is made for the scene where the pattern instances interfere with each other, and the efficiency has been improved by up to 90%.

Users can perform fillet, Boolean or hold design on any pattern instance, and then use the reference pattern to implement the dependent features on all pattern instances.

【What can be done】:

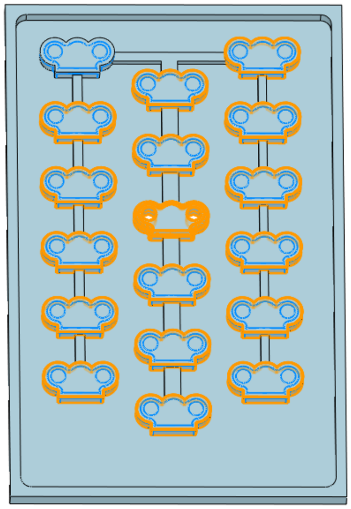
1. The scene efficiency optimization of pattern instance interference can reach 90%.
2. The spacing and number of circular patterns now support multiple mode definitions. You can define the size of the circular pattern in three modes: pattern angle and number, pattern number and range, and pattern angle and range in the **Pattern Feature >> Circular >> Mode >> First Direction**.
3. By number, users can easily find, select, or deselect a pattern item that has been excluded in the **Pattern Feature >> Instances to Toggle**. When an item is selected in the list, it will be highlighted in the drawing area.

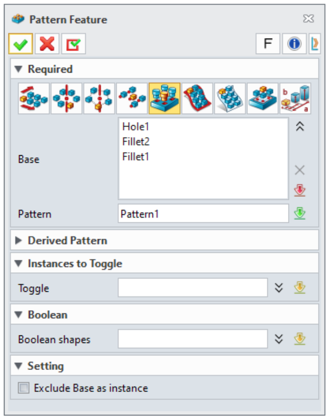
【Notes】:

- Features that both refer to patterns with and without dependencies are not supported.
- If the pattern has engineering features such as circular chamfer, draft, and face offset, the efficiency improvement cannot reach 90%.

【Example】:

1. If users hole and fillet any pattern instance, the holed and filleted pattern can be applied to all other associated pattern instances by referring to the pattern function.




Pattern Feature
F
Required
Derived Pattern
Instances to Toggle
Boolean
Setting

Base: Hole1, Fillet2, Fillet1

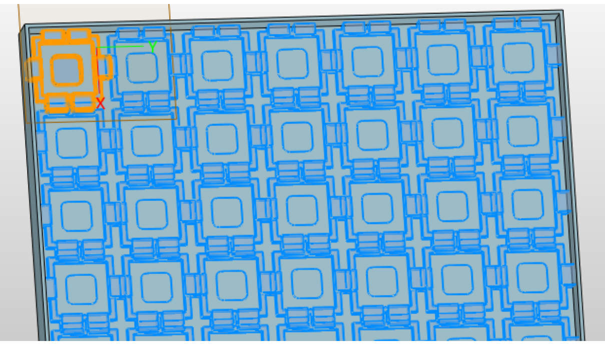
Pattern: Pattern1

Toggle:

Boolean shapes:

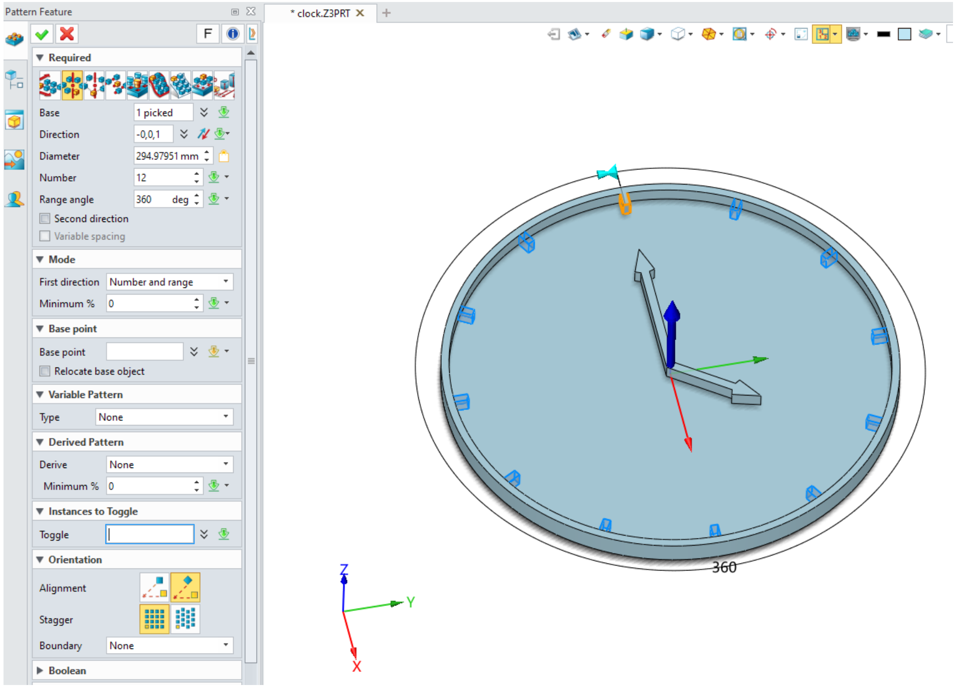
☐ Exclude Base as instance

2. Perform an extrude cut on a plane and draft the cutting faces. Finally, pattern the designed structure to the whole plane and the efficiency is greatly improved.



ZW3D 2024 Effect	ZW3D 2025 Effect
165s	39s

3. In the circular pattern, the pattern interval and quantity are controlled by the number and range angle. In the clock example, the range angle (360°) and number (12 scale) modes are set to assign a specified number of features evenly across a specified range of angles, eliminating the need to calculate the angles between the pattern items.



Pattern Feature

Required

Base: 1 picked

Direction: -0,0,1

Diameter: 294.97951 mm

Number: 12

Range angle: 360 deg

☐ Second direction

☐ Variable spacing

Mode

First direction: Number and range

Minimum %: 0

Base point

Base point:

☐ Relocate base object

Variable Pattern

Type: None

Derived Pattern

Derive: None

Minimum %: 0

Instances to Toggle

Toggle:

Orientation

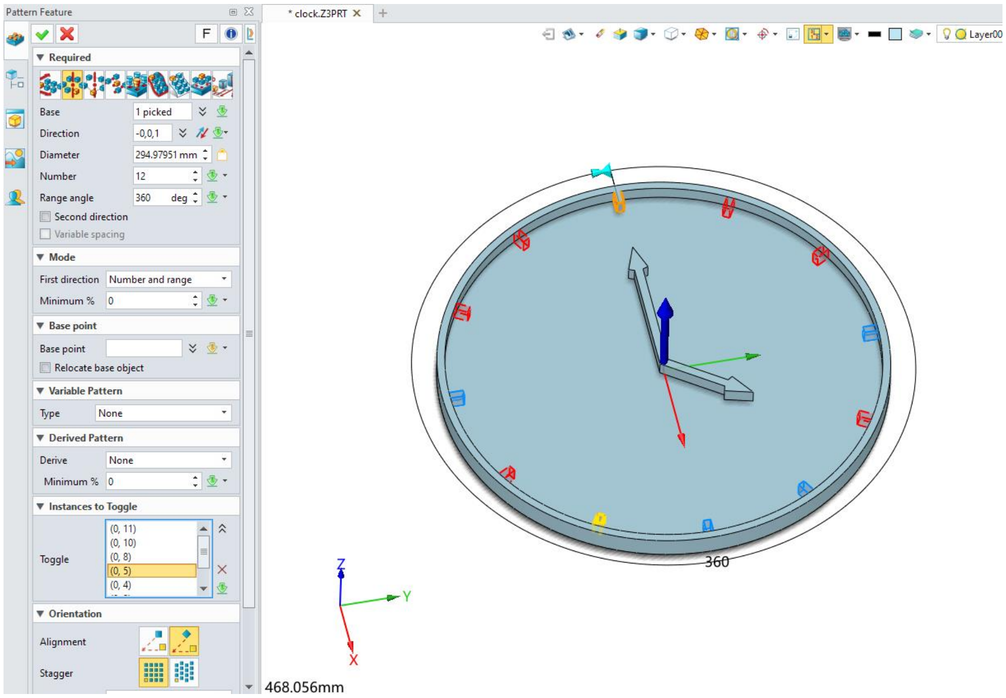
Alignment

Stagger

Boundary: None

Boolean

4. After finding all excluded pattern items, you can select and eliminated them.



Pattern Feature

Required

Base: 1 picked

Direction: -0,0,1

Diameter: 294.97951 mm

Number: 12

Range angle: 360 deg

☐ Second direction

☐ Variable spacing

Mode

First direction: Number and range

Minimum %: 0

Base point

Base point:

☐ Relocate base object

Variable Pattern

Type: None

Derived Pattern

Derive: None

Minimum %: 0

Instances to Toggle

Toggle: (0, 11)
(0, 10)
(0, 8)
(0, 5)
(0, 4)

Orientation

Alignment

Stagger

【Where is it】:

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

3.2.4 Direct Edit Efficiency Improvement

ZW3D 2025 has greatly improved the efficiency and correctness of some direct editing (hereinafter referred to as DE) commands, such as DE move, replace, simplify, etc., and the execution time of the command can be reduced by up to 80%. So now you can use the DE command more efficiently to edit the shape of an existing model.

【What can be done】:

- ✓ The execution time of DE face offset, DE move, and replace commands can be reduced by up to 80%.

【Note】:

- The rib and boss features are not supported by Simplify function.

【Where is it】:

Part/Assembly Environment>>Direct Edit>>Modify

3.2.5 New Major Diameter Preview

The major diameter of a thread hole can now be previewed in the 3D model. You can directly see whether there is interference between the thread holes in the 3D model.

【What can be done】:

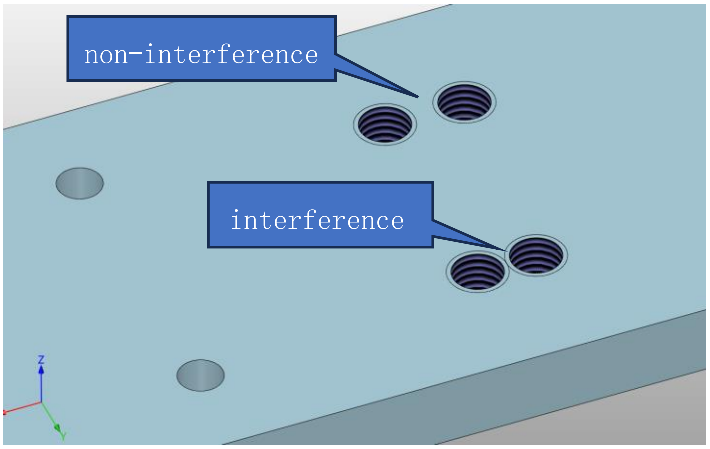
1. Correctly display the major diameter of a thread according to its specification.
2. Globally hide the preview of major diameter.
3. The preview of the major diameter will be generated in the start and end of thread.

【Notes】:

- The minor diameter of a thread is not supported.
- The major diameter preview cannot be exported as cable.

【Example】:

1. In 3D modeling environment, check whether there is interference in thread hole.



【Where is it】:

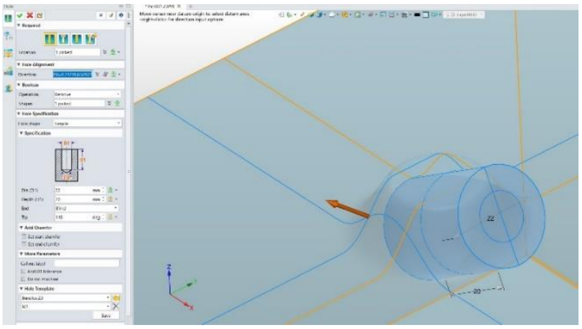
Configuration>>Display>>Toggle settings>>Display Major Diameter

3.2.6 Hole Feature UI Upgrade

ZW3D 2025 has optimized the operation logic of the hole command, and now the direction of a hole can be defined by surface normal, edge lines, and curves.

【What can be done】:

1. Define the hole direction by picking plane/surface.



【Note】:

- The hole alignment >> face normal input has been removed, and it is no longer possible to control the position range of the point by selecting the region of the face.

【Where is it】:

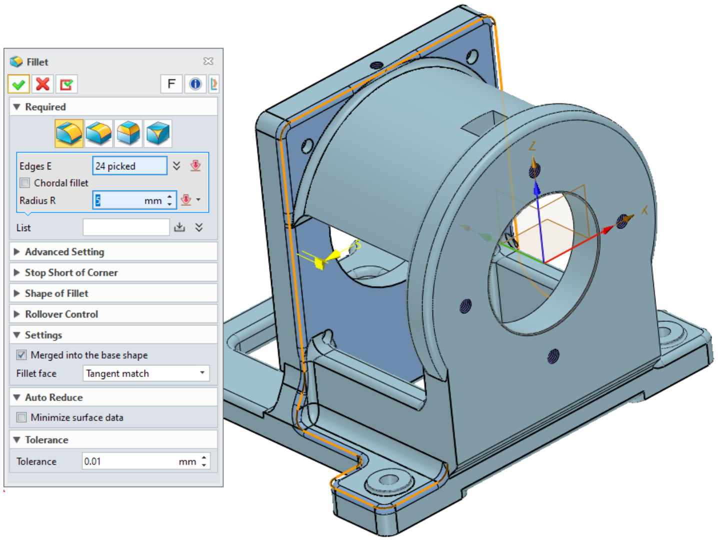
Part/Assembly Environment>>Shape>> Basic Editing

3.2.7 Fillet

In the design process of castings or injection molded parts, long fillet chains are often used to generate fillets, and new fillets often cross the old fillet surface. The new version has improved the support for such scenarios, on the one hand, it has improved the success rate and efficiency of fillet, on the other hand, the generation algorithm of the new and old fillet bridge surface has been improved, which is conducive to subsequent modeling operations.

【What can be done】:

- ✓ By selecting the entire tangent chain at once and setting the fillet radius, you can create the correct fillet.
- ✓ The offset of the old and new fillet faces is set to be close to the fillet radius and can be successfully implemented.



Fillet
F
Required
Advanced Setting
Stop Short of Corner
Shape of Fillet
Rollover Control
Settings
Auto Reduce
Tolerance

Edges E: 24 picked

☐ Chordal fillet

Radius R: 5 mm

List:

☑ Merged into the base shape

Fillet face: Tangent match

☐ Minimize surface data

Tolerance: 0.01 mm

【Note】:

- If the fillet radius is too large, that is, if the fillet is completely outside the support surface, check the “Hold fillet to edge” option.

【Where is it】:

Part>>Shape>>Fillet

3.2.8 New morph function - "Curved Bend"（ZW3D 2025 SP）

ZW3D 2025 SP adds new function " Curved Bend ", which can morph selected shapes to the curve by specifying a target curve. This function supports automatic calculation or custom definition of the original curve, and allows scaling the shape to match the length of the target curve while bending, achieving flexible modeling requirements.

【What can be done】:

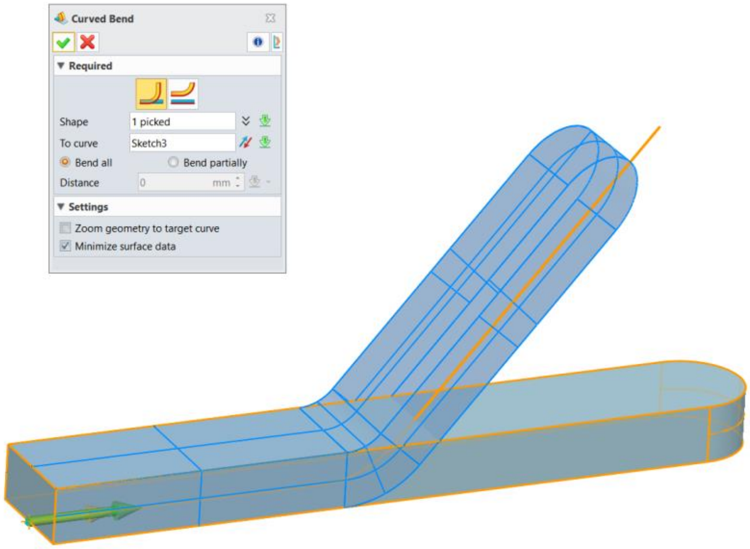
- ✓ When "From curve" is automatically generated, the shapes will be segmented according to the target curve.
- ✓ Support bending entire shapes or specified length.
- ✓ " From curve " can be set to either automatically calculate or customize.

【Notes】:

- If the shapes are longer than "To curve" and "Zoom geometry to target curve" is unchecked, ZW3D may provide an unexpected result.
- When using "Customized from curve" mode, the shapes will not be separation.

【Example】:

1. When automatically calculating the original curve, you can directly bend the sheet or tubular object according to the target curve.



Curved Bend

Required

Shape 1 picked

To curve Sketch3

● Bend all ○ Bend partially

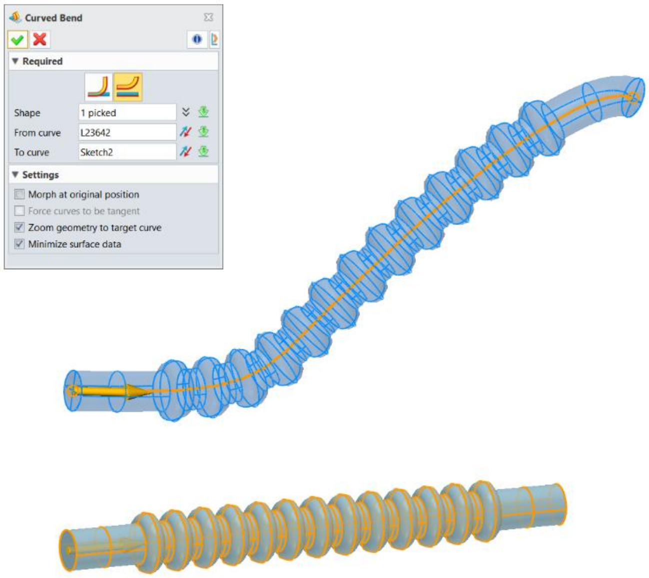
Distance 0 mm

Settings

☐ Zoom geometry to target curve

☑ Minimize surface data

2. When customizing the original curve, you can deform the tubular object and move it to the target curve position.



Curved Bend

Required

Shape 1 picked

From curve L23642

To curve Sketch2

Settings

☐ Morph at original position

☐ Force curves to be tangent

☑ Zoom geometry to target curve

☑ Minimize surface data

【Where is it】:

Part/Assembly Environment>>Shape>>Morph>>Curved Bend

3.2.9 Multi-entities efficiency improvement（ZW3D 2025 SP）

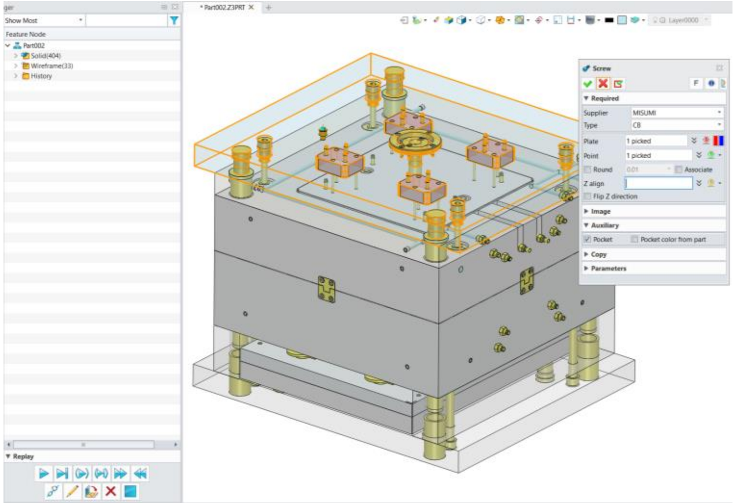
ZW3D 2025 SP greatly improved the efficiency of command execution and preview in multi-entities environment. When editing a design in multi-entities environment, the efficiency is nearly same as single-entity environment. Additionally, the preview effect has also been optimized, making the drawing area mouse movement and dimension dragging smoother when the model is highlighted.

【What can be done】:

- ✓ Multi-entity design efficiency has been improved by over 50%, and can increase by up to 90% with larger quantities.
- ✓ Most efficiency of modeling commands have been greatly improved. The efficiency of Mold and Direct edit module has also been greatly improved.
- ✓ When dragging DDD control in the drawing area, the preview is much smoother.

【Example】:

1. In scenarios with a large number of entities, such as mold design, smooth design and editing are performed.


ger
Show Most
Feature Node
Part002
Solid(404)
Wireframe(33)
History
* Part002.Z3PRT
Layer0000
Screw
F
Required
Image
Auxiliary
Copy
Parameters
Replay

Supplier: MISUMI
Type: CB
Plate: 1 picked
Point: 1 picked
☐ Round 0.01 ☐ Associate
Z align:
☐ Flip Z direction
☑ Pocket ☐ Pocket color from part

【Where is it】

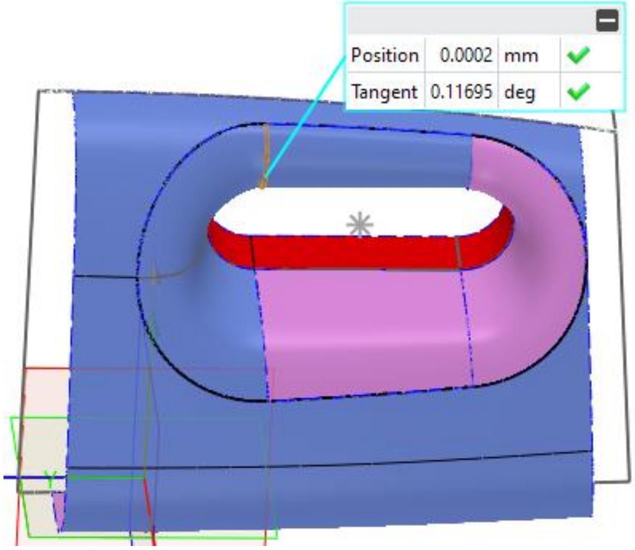
Part/Assembly Environment

3.3 Free Form Design

In the 3C product design process, a large surface is often created first, then the surface with a desired shape is trimmed by Boolean pruning, and the surfaces between different large surfaces are linked by creating small surfaces. ZW3D 2025 can not only ensure the G1 continuity between the large surface and the small surface, but also ensure that the boundaries between the generated small surface and the large surface coincide.

【What can be done】:

- ✓ The generated surface can not only satisfy the set G1 boundary continuity, but also ensure that the generated surface and the input edge line coincide completely.



Position	0.0002	mm	✓
Tangent	0.11695	deg	✓

【Note】:

- When using UV surface, if the boundary continuity is set to G2, you should pay attention to check whether there is a detachment of the boundary.

【Where is it】:

Part>>Surface>> UV Surface

3.4 Assembly Design

In assembly design, ZW3D has optimized large assembly efficiency, association update, assembly constraints, assembly insertion, and so on, which can provide a correct and efficient design experience for users in the main assembly design scenarios and realize “One Click Regeneration”. We make new UI for insert, which allows users to quickly find objects that need to be inserted. Besides, Assembly Insert supports quickly inserting specified file objects from assembly tree/folder. Please check the following chapters and the help documentation for other improvements and detailed descriptions.

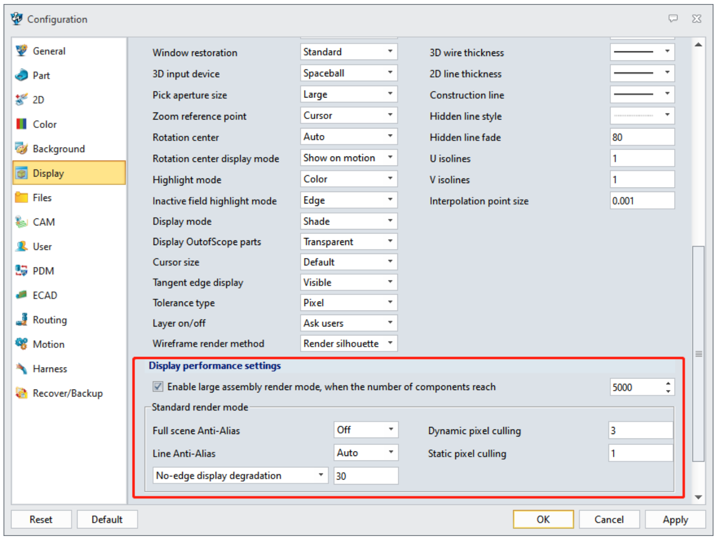
3.4.1 ★Large Assembly Efficiency Improvement

ZW3D 2025 has optimized the efficiency of common commands for large assembly scenes, such as Open/Close, View Model, Insert, Add Constraints, Pattern/Mirror, Show/Hide, Drag/Move/Fix, Interference Check, and Open 3D BOM.

【**What can be done**】:

- ✓ Assembly design efficiency of 5,000 to 10,000 components is increased by more than 50%, and the common commands respond smoothly, which can provide a stable and smooth assembly design experience for machinery and equipment industry customers.
- ✓ Supports smooth design and preview of assemblies containing tens of thousands of components. For large engineering equipment, provide a stable assembly design platform with great improvements in assembly design experience.
- ✓ Compared to ZW3D 2024, file opening efficiency improves by more than 10%, file closing and renaming efficiency improves by more than 80%, and users can smoothly preview the assembly containing more than 5,000 components. Common assembly commands such as Open/Close, View Model, Insert, Add Constraints, Pattern/Mirror, Show/Hide, Drag/Move/Fix, Interference Check, and Open 3D BOM can respond smoothly under 5,000 component assembly, and the efficiency improves by more than 50% and by more than 80% in some scenarios.
- ✓ When open a large assembly model, users need to wait a minute (usually within 5s) to complete the rendering recommendation calculation and intelligent display scheme recommendation to enjoy a smooth model browsing experience. Users can manage display optimization-related

content in Configuration.



Window restoration	Standard	3D wire thickness	
3D input device	Spaceball	2D line thickness	
Pick aperture size	Large	Construction line	
Zoom reference point	Cursor	Hidden line style	
Rotation center	Auto	Hidden line fade	80
Rotation center display mode	Show on motion	U isolines	1
Highlight mode	Color	V isolines	1
Inactive field highlight mode	Edge	Interpolation point size	0.001
Display mode	Shade		
Display OutofScope parts	Transparent		
Cursor size	Default		
Tangent edge display	Visible		
Tolerance type	Pixel		
Layer on/off	Ask users		
Wireframe render method	Render silhouette		

Display performance settings

☑ Enable large assembly render mode, when the number of components reach 5000

Standard render mode

Full scene Anti-Alias	Off	Dynamic pixel culling	3
Line Anti-Alias	Auto	Static pixel culling	1
No-edge display degradation	30		

【Note】:

- As you rotate a model in preview, the large assembly may get slightly stuck when carrying out intelligent calculation on the rotation center.

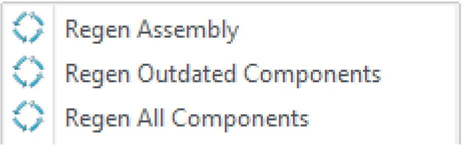
【Where is it】:

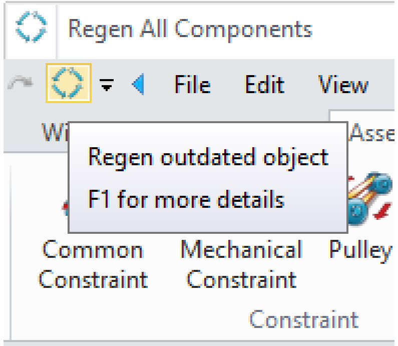
Configuration>>Display>> Display performance settings

3.4.2 ★One Click Regeneration & Auto Regeneration

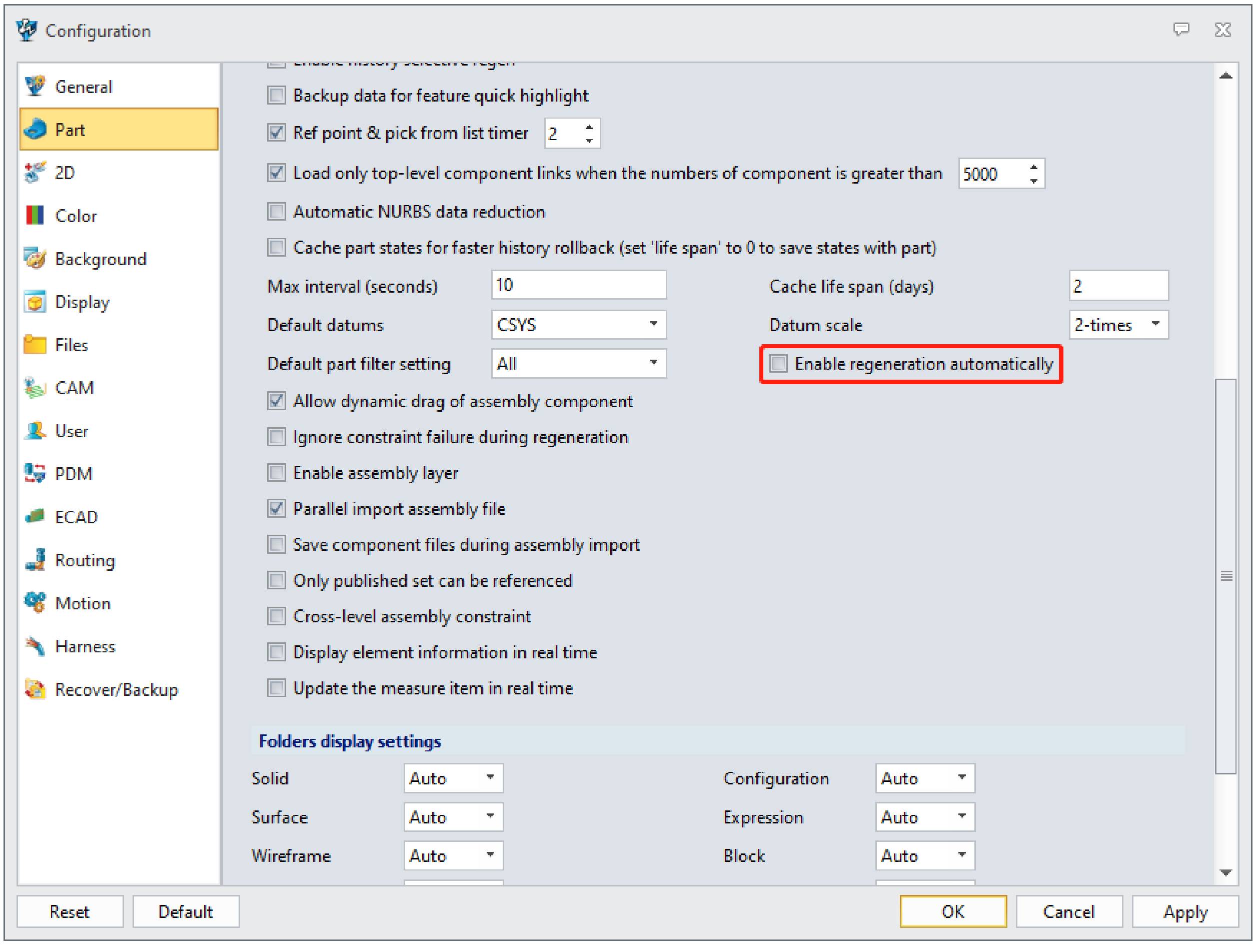
Assembly regeneration gets fully upgrade in ZW3D 2025. “**Regen Outdated Components**” and “**Regen Assembly**” have been optimized and combined into the “**Regen Outdated Object**” function. Users can use the update button to refresh all outdated objects with one click.

ZW3D 2024	ZW3D 2025


Regen Assembly
Regen Outdated Components
Regen All Components


Regen All Components
File
Edit
View
Wi
Asse
Regen outdated object
F1 for more details
Common Constraint
Mechanical Constraint
Pulley
Constraint

In addition, ZW3D 2025 realizes Auto Regeneration. After the “Enable regeneration automatically” option is enabled in the Configuration panel, the system will update the model according to the outdated status automatically when the model is modified, instead of refreshing by manual.


Configuration
General
Part
2D
Color
Background
Display
Files
CAM
User
PDM
ECAD
Routing
Motion
Harness
Recover/Backup
Reset
Default
OK
Cancel
Apply

☐ Backup data for feature quick highlight
☑ Ref point & pick from list timer 2
☑ Load only top-level component links when the numbers of component is greater than 5000
☐ Automatic NURBS data reduction
☐ Cache part states for faster history rollback (set 'life span' to 0 to save states with part)
Max interval (seconds) 10
Cache life span (days) 2
Default datums CSYS
Datum scale 2-times
Default part filter setting All
☐ Enable regeneration automatically
☑ Allow dynamic drag of assembly component
☐ Ignore constraint failure during regeneration
☐ Enable assembly layer
☑ Parallel import assembly file
☐ Save component files during assembly import
☐ Only published set can be referenced
☐ Cross-level assembly constraint
☐ Display element information in real time
☐ Update the measure item in real time

Folders display settings
Solid Auto
Configuration Auto
Surface Auto
Expression Auto
Wireframe Auto
Block Auto

【What can be done】:

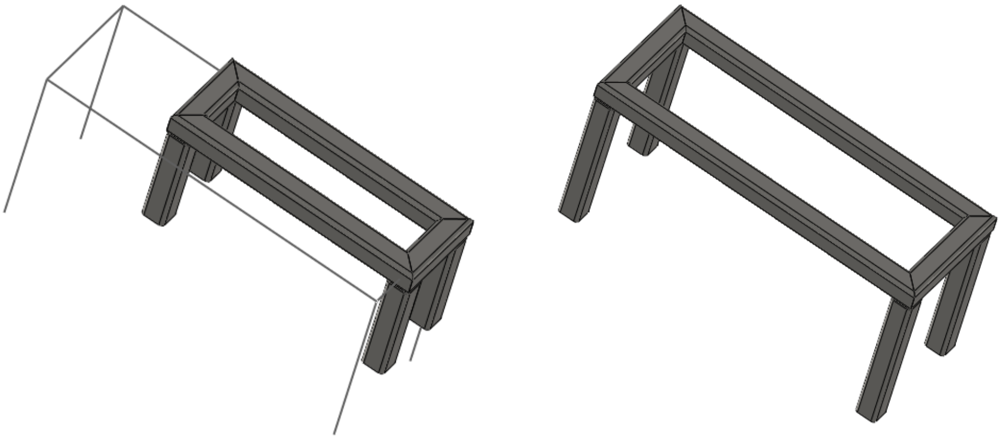
- ✓ Without considering the assembly update mode, one click updates all outdated objects in the current file in the Assembly environment.
- ✓ Without the manual assembly update, the software can recognize and refresh outdated objects automatically.

【Notes】:

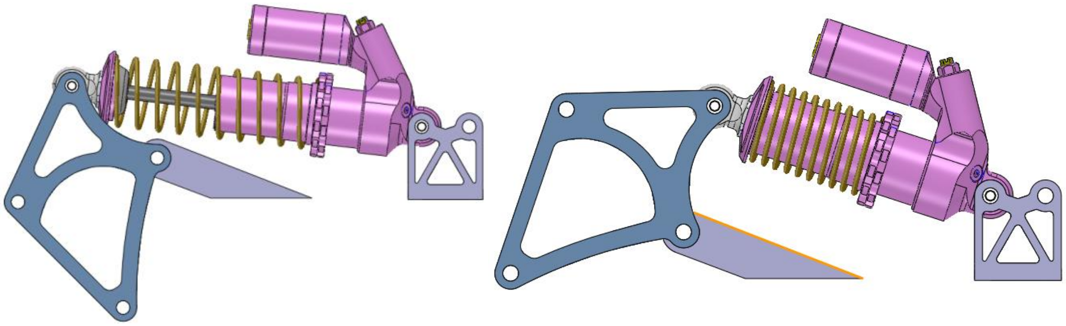
- When the assembly file contains features, including assembly fillet, assembly chamfer, and assembly hole, if “Regen Outdated Object” cannot refresh completely, you can use "Regen All Components" in the right-click menu to update the model.
- In the design of a large complex assembly model, as Automatic Regeneration is enabled, it may cause the problem of waiting too long after modifying.

【Example】:

1. When designing a steel frame, after modifying the central line size of the frame, use the “Regen” button to update the model in one click.



2. After automatic regeneration is enabled, drag the component to make the spring flexible part refresh automatically.

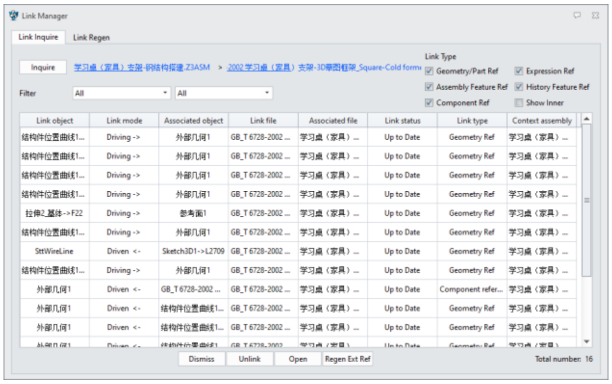
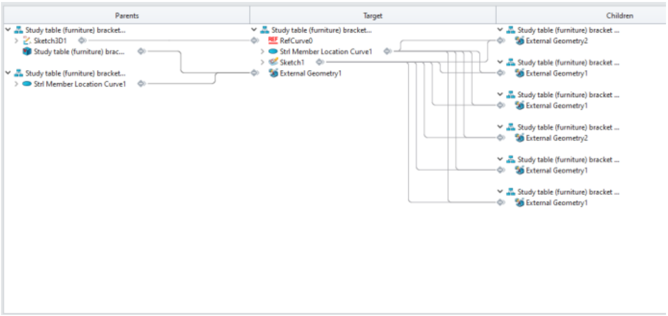


【Where is it】:

Configuration>>Part>> Enable regeneration automatically

3.4.3 ★Link Manager

The presentation of the link relationship gets fully upgraded in ZW3D 2025. The link relationship is displayed in the form of a tree structure network diagram, by which users can quickly and clearly understand the information, such as which feature and which part the link object belongs to. The specific form is as follows:

ZW3D 2024	ZW3D 2025
	

Link Manager

Link Inquire | Link Regen

Inquire 学习桌（家具）支架-钢结构搭建.Z3ASM > 2002 学习桌（家具）支架-3D草图框架_Square-Cold form

Filter: All / All

Link Type: ☑ Geometry/Part Ref ☑ Expression Ref ☑ Assembly Feature Ref ☑ History Feature Ref ☑ Component Ref ☐ Show Inner

Link object	Link mode	Associated object	Link file	Associated file	Link status	Link type	Context assembly
结构件位置曲线1...	Driving ->	外部几何1	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
结构件位置曲线1...	Driving ->	外部几何1	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
结构件位置曲线1...	Driving ->	外部几何1	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
结构件位置曲线1...	Driving ->	外部几何1	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
拉伸2_基体->F22	Driving ->	参考面1	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
结构件位置曲线1...	Driving ->	外部几何1	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
SttWireLine	Driven <-	Sketch3D1->L2709	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
结构件位置曲线1...	Driving ->	外部几何1	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
外部几何1	Driven <-	GB_T 6728-2002 ...	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Component refer...	学习桌（家具）...
外部几何1	Driven <-	结构件位置曲线1...	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
外部几何1	Driven <-	结构件位置曲线1...	GB_T 6728-2002 ...	学习桌（家具）...	Up to Date	Geometry Ref	学习桌（家具）...
外部几何1	Driven <-	结构件位置曲线1	GB_T 6728-2002	学习桌（家具）	Up to Date	Geometry Ref	学习桌（家具）

Dismiss | Unlink | Open | Regen Ext Ref

Total number: 16

Parents
Target
Children
Study table (furniture) bracket...
Sketch3D1
Study table (furniture) brac...
Study table (furniture) bracket...
Strl Member Location Curve1
Study table (furniture) bracket...
RefCurve0
Strl Member Location Curve1
Sketch1
External Geometry1
Study table (furniture) bracket ...
External Geometry2
Study table (furniture) bracket ...
External Geometry1
Study table (furniture) bracket ...
External Geometry1
Study table (furniture) bracket ...
External Geometry2
Study table (furniture) bracket ...
External Geometry1
Study table (furniture) bracket ...
External Geometry1

In addition, the query mode of link relationship gets fully upgraded in ZW3D 2025. Users can quickly inquire the link relationship range through different query modes. Details are as follows:

- Inquire the link relationships for a specified object.

The target object can be a part, assembly, feature, variable, and so on. After selection, the interface will display all the reference relationships of the object, including other objects referring to the object or the object referring to other objects.

- Inquire about the link relationships between two objects.

  Inquire about and display the reference relationship between two objects.

- Inquire about all link relationships for an assembly file.

  The target object is an assembly file. Users can inquire about the reference relationships between all components in this assembly file and understand the reference of the whole file quickly.

- Inquire about circular reference relationships.

  Check all the currently loaded link relationships and show the circular reference relationships.

【What can be done】:

✓ Quickly sort out the reference relationship in the model and correct the wrong reference relationship can be.

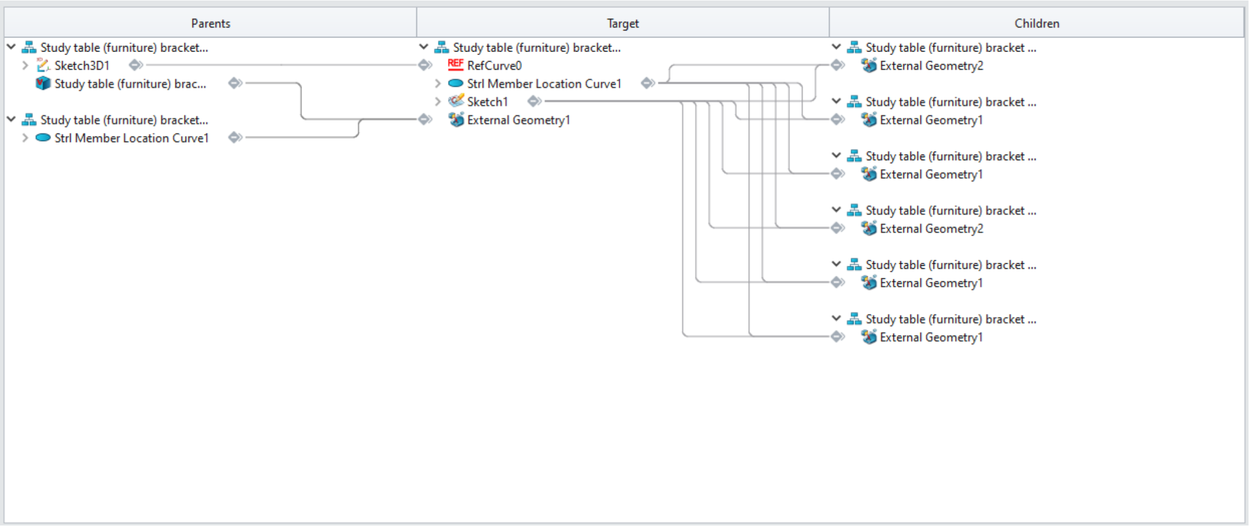
✓ Quickly understand the circular reference relationship in the loaded file and solve the circular reference.

【Note】:

- When there are too many link relationships, it will take longer to construct a tree structure network diagram of the link relationship.

【Example】

1. Inquire about the reference range of the modified object to ensure the correctness of the model.


Parents
Target
Children
Study table (furniture) bracket...
Sketch3D1
Study table (furniture) brac...
Study table (furniture) bracket...
Strl Member Location Curve1
Study table (furniture) bracket...
RefCurve0
Strl Member Location Curve1
Sketch1
External Geometry1
Study table (furniture) bracket ...
External Geometry2
Study table (furniture) bracket ...
External Geometry1
Study table (furniture) bracket ...
External Geometry1
Study table (furniture) bracket ...
External Geometry2
Study table (furniture) bracket ...
External Geometry1
Study table (furniture) bracket ...
External Geometry1

【Where is it】：

Utilities>>Tools>>Link Manager

3.4.4 Assembly Popup Optimization

ZW3D 2025 has improved the assembly pop-up window. When deleting or modifying objects, such as assembly components and assembly features, the affected objects will be displayed more clearly in the pop-up window, which is helpful for users to operate. The specific categories are as follows:

- Affected features

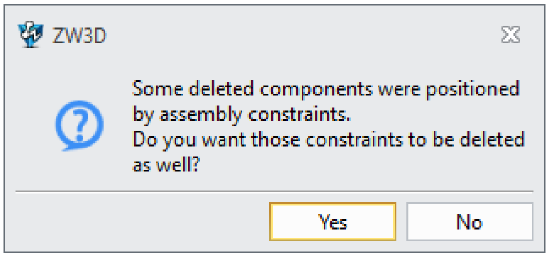
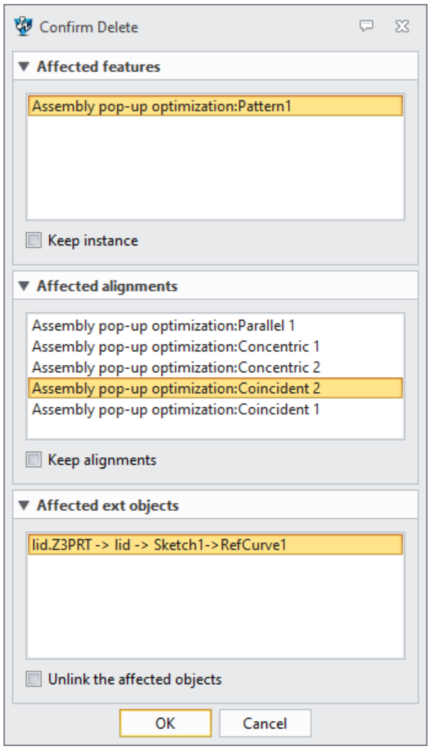
  The features affected by deleting the object, such as assembly pattern, assembly mirror, etc. Users can check the “Keep instance” option to dissolve the features relation and keep the result instance.

- Affected alignments

  The constraints affected by deleting the object. Users can check the “Keep alignments” option to remain the affected constraints.

- Affected external objects

  The objects referencing the deleted object. Users can check the “Unlink the affected objects” option to disconnect these references.

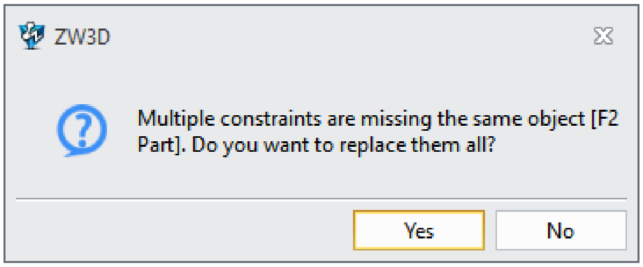
ZW3D 2024	ZW3D 2025
 ZW3D Some deleted components were positioned by assembly constraints. Do you want those constraints to be deleted as well? 	 Confirm Delete Affected features Assembly pop-up optimization:Pattern1 ☐ Keep instance Affected alignments Assembly pop-up optimization:Parallel 1 Assembly pop-up optimization:Concentric 1 Assembly pop-up optimization:Concentric 2 Assembly pop-up optimization:Coincident 2 Assembly pop-up optimization:Coincident 1 ☐ Keep alignments Affected ext objects lid.Z3PRT -> lid -> Sketch1->RefCurve1 ☐ Unlink the affected objects

3.4.5 ★Assembly Constraints

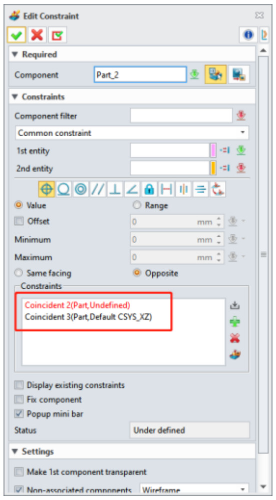
ZW3D 2025 has optimized the stability of the assembly constraint, providing a more stable and correct result, including the correctness of the flexible sub-assembly. The solutions to the lost constraints have been optimized, allowing them to be retrieved in batches.

【What can be done】:

- ✓ Flexible sub-assembly dragging for verification.
- ✓ Flexible sub-assembly and its pattern or mirror results can be associated.
- ✓ When multiple constraints lose the same object, all the constraints at the same level that lose the same object can be found at once by redefining one of them. Besides, users can see the missing constraints of the corresponding component by the Edit Constraint command.



Multiple constraints are missing the same object [F2 Part]. Do you want to replace them all?


Edit Constraint
Required
Constraints
Component filter
1st entity
2nd entity
Settings

Component: Part_2

Common constraint

● Value ○ Range

☐ Offset 0 mm

Minimum 0 mm

Maximum 0 mm

○ Same facing ● Opposite

Constraints:

Coincident 2(Part,Undefined)

Coincident 3(Part,Default CSYS_XZ)

☐ Display existing constraints

☐ Fix component

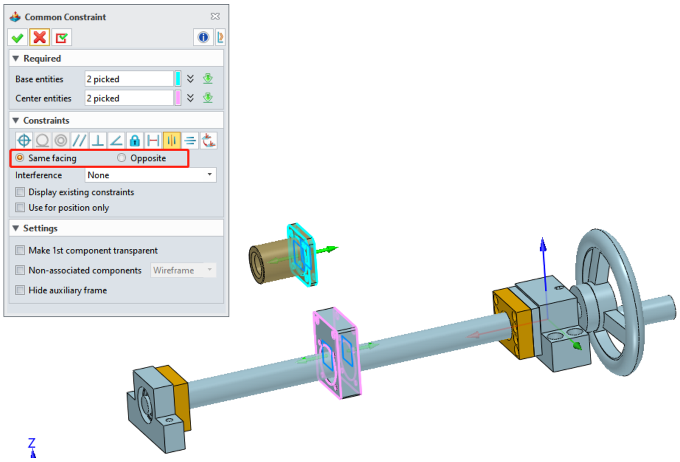
☑ Popup mini bar

Status: Under defined

☐ Make 1st component transparent

☑ Non-associated components: Wireframe

- ✓ Middle constraint can be flipped when selecting two component objects.


Common Constraint
Required
Constraints
Settings
Z

Base entities: 2 picked

Center entities: 2 picked

● Same facing ○ Opposite

Interference: None

☐ Display existing constraints

☐ Use for position only

☐ Make 1st component transparent

☐ Non-associated components: Wireframe

☐ Hide auxiliary frame

【Note】:

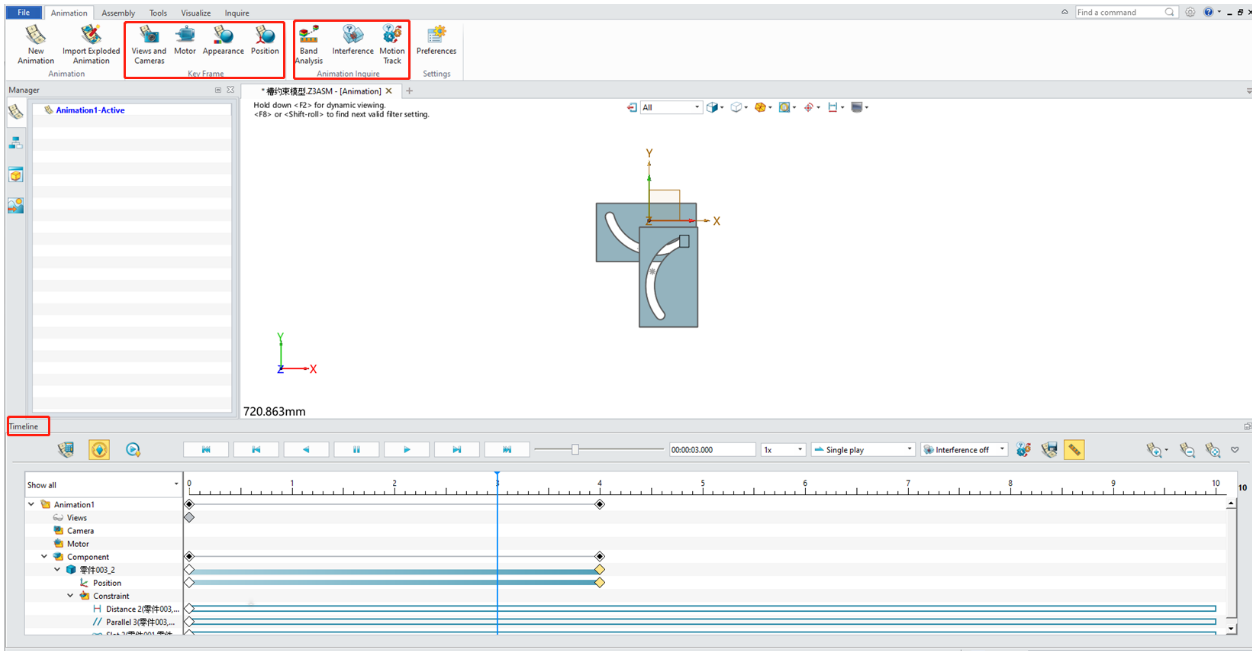
- Constraint batch replace can only help the same-level constraints that are missing the same object to find the lost object.

【Where is it】:

Assembly Environment>>Constraint

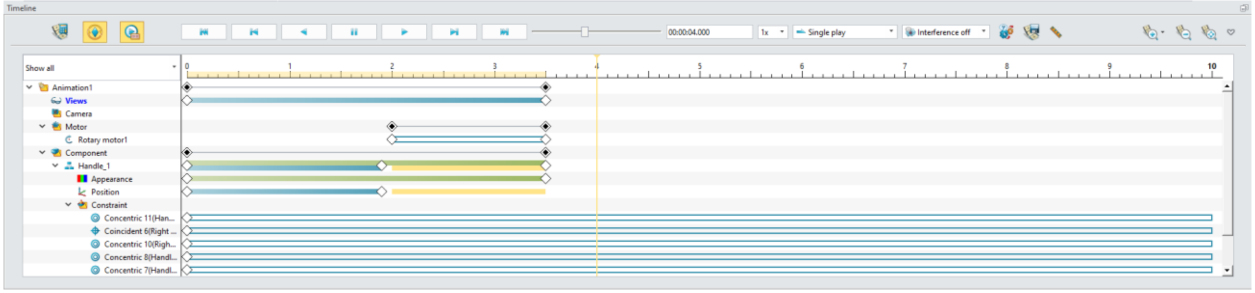
3.4.6 ★Assembly Animation Reconstruction

ZW3D 2025 reconstructs the assembly animation module, including: introducing the timeline to change the management mode of keyframes, supporting flexible sub-assembly in creating animations, realizing the real-time transmission of data in the Assembly environment to the Animation environment, and supporting animations under different configurations for rapid reuse. These improvements have greatly improved the efficiency and ease of use of creating and editing animations. Besides, the improvement of the interface makes it more in line with the style of mainstream animation design software and reduces the learning cost. Please check the following chapters and the help documentation for other improvements and detailed descriptions. The reconstructed assembly animation interface is as follows:


File
Animation
Assembly
Tools
Visualize
Inquire
New Animation
Import Exploded Animation
Animation
Views and Cameras
Motor
Appearance
Position
Key Frame
Band Analysis
Interference
Motion Track
Animation Inquire
Preferences
Settings
Find a command
Manager
Animation1-Active
Hold down <F2> for dynamic viewing.
<F8> or <Shift-roll> to find next valid filter setting.
All
Y
X
720.863mm
Timeline
00:00:03.000
1x
Single play
Interference off
Show all
Animation1
Views
Camera
Motor
Component
零件003_2
Position
Constraint
Distance 2(零件003,...
Parallel 3(零件003,...
0
1
2
3
4
5
6
7
8
9
10

3.4.6.1 New Timeline

ZW3D 2025 assembly animation module introduces a timeline to manage keyframes and classifies different types of keyframes in the form of nodes. Keyframes can be reused in batches and edited, which enhances the efficiency of creating animations. Different types of keyframes (view, motor, position, and appearance) are classified and stored, which makes animation production, editing, and preview more intuitive and convenient. The timeline interface is as follows:


Timeline
00:00:04.000
1x
Single play
Interference off
Show all
0
1
2
3
4
5
6
7
8
9
10
Animation1
Views
Camera
Motor
Rotary motor1
Component
Handle_1
Appearance
Position
Constraint
Concentric 11(Han...
Coincident 6(Right ...
Concentric 10(Righ...
Concentric 8(Handl...
Concentric 7(Handl...

【What can be done】：

- ✓ Calculate and play animation, switch to real-time play and frame-by-frame play, and enable/disable automatic creation of keyframes.
- ✓ Modify playback mode, control playback progress, and adjust playback speed.
- ✓ Enable/disable the analysis functions, such as animation measurement, interference checks, motion track, etc.
- ✓ Zoom in, zoom out, and reset the timeline.
- ✓ Manage and edit different types of keyframes (copy/paste, delete, mirror, reverse, modify interpolation forms, edit time, edit parameters and other operations).

【Where is it】：

Assembly Environment >> Animation>> Timeline

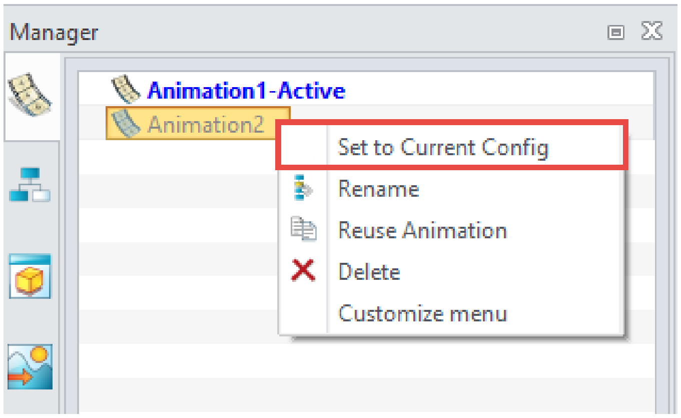
3.4.6.2 Create Animation for Flexible Sub-Assembly, Update and Transfer Assembly Data, and Reuse Animation for Different Configurations

ZW3D 2025 supports users making animations for multi-level flexible sub-assembly and covers more animation production scenes at multiple-level sub-assembly. It also supports the fast reuse of animations for different configurations. On this basis, the edited assembly data can be passed to the animation environment in real time, and the secondary editing of unaffected animation data is reduced, which improves the operation efficiency and reduces the editing costs.

【What can be done】：

- ✓ Create animations for multi-level flexible sub-assembly.

- ✓ Quickly reuse animation for different configurations.
- ✓ Real-time update and transmit assembly data.

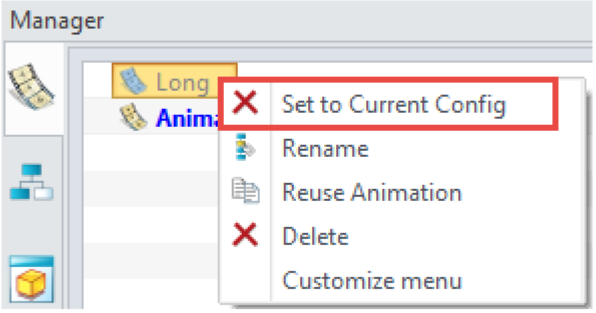


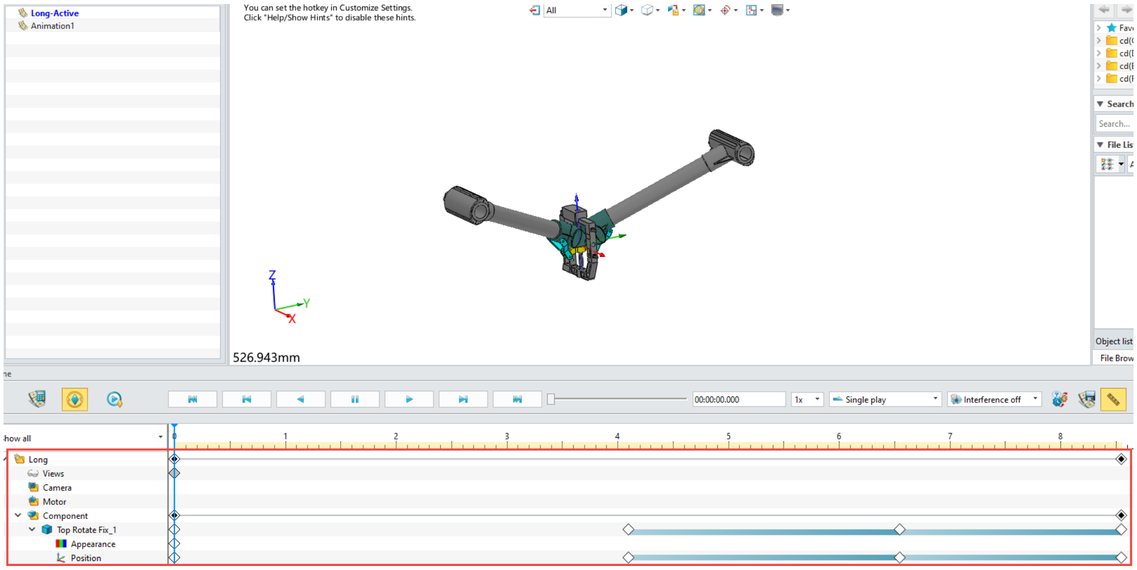
【Notes】：

- The motor or position keyframes created in the rigid/flexible state in the animation will be automatically suppressed after the rigid/flexible state of sub-assembly in the Assembly environment is switched. When the rigid/flexible status is switched back, it will be unsuppressed automatically.
- To avoid frequent changes in component position in the Assembly environment, which will affect the position of the initial moment of the created animation, the modification of the component position in the Assembly environment will not be passed to the existing animation.

【Example】：

1. In the Animation environment, you can see a gray animation instance named "Long" in the animation manager. By right-clicking this instance and choosing “Set to Current Config”, the animation created in the “Long” configuration of the assembly model can be reused in the Default configuration.


Manager
Long
Anima
Set to Current Config
Rename
Reuse Animation
Delete
Customize menu



You can set the hotkey in Customize Settings.
Click "Help/Show Hints" to disable these hints.

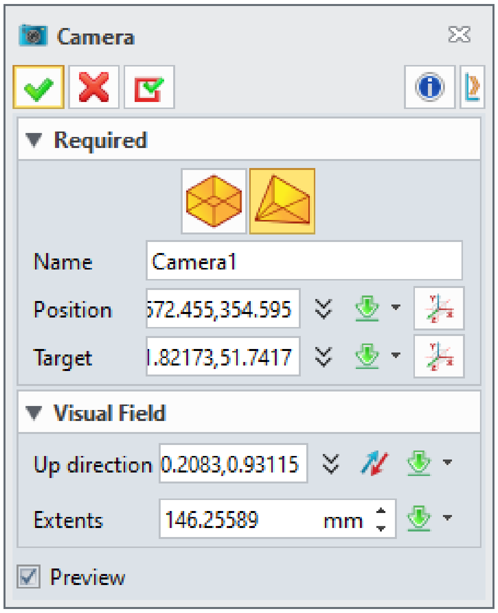
Long-Active
Animation1
526.943mm
00:00:00.000
1x
Single play
Interference off
Show all
Long
Views
Camera
Motor
Component
Top Rotate Fix_1
Appearance
Position
Object list
File Brow

【Where is it】：

Assembly Environment >> Animation

3.4.6.3 New Camera Keyframe

ZW3D2025 has optimized the original camera function, adding the perspective view based on the existing parallel view. At the same time, it has added the preview and operation window, which provides quick operation to check and adjust the camera view. It covers scenes of local viewing and the model’s global-to-local perspective switching for users to see global or detailed views.



Camera

Required

Name: Camera1

Position: 572.455,354.595

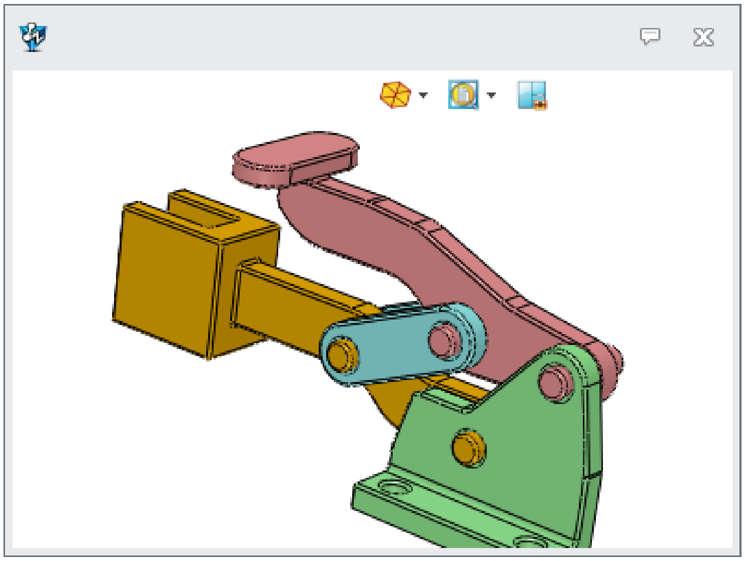
Target: 1.82173,51.7417

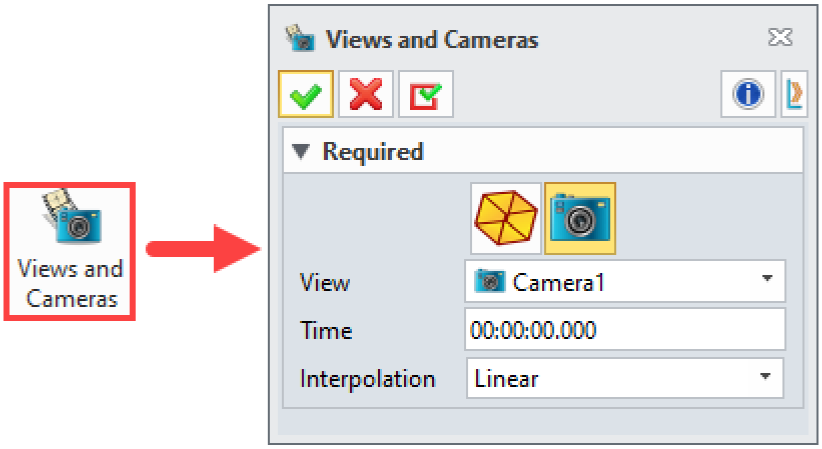
Visual Field

Up direction: 0.2083,0.93115

Extents: 146.25589 mm

☑ Preview





Views and Cameras

Required

View: Camera1

Time: 00:00:00.000

Interpolation: Linear

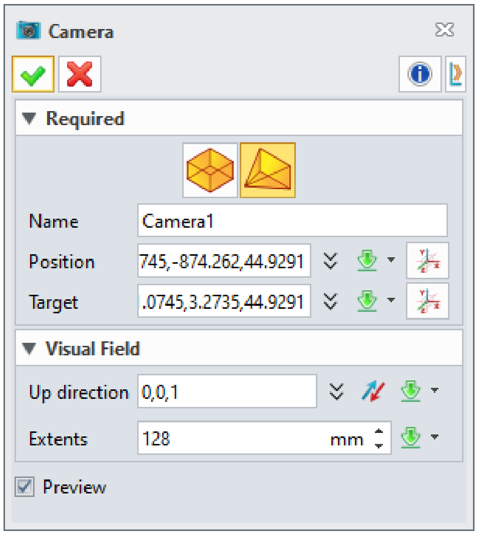
【What can be done】:

- ✓ Create camera keyframes of the same camera that changes at different times.
- ✓ Create view keyframes of different cameras that changes at different times.
- ✓ Support to modify the interpolation form of keyframes (linear and none).

【Example】:

1. Select “New camera” from the context menu of “Camera view” under the View Manager, input “Camera 1” as the camera name, and put the model in the position as shown in the following figure in the preview window. Click OK, and a camera named “Camera 1” is created under the camera view folder in View

Manager.



Camera

Required

Name Camera1

Position 745,-874.262,44.9291

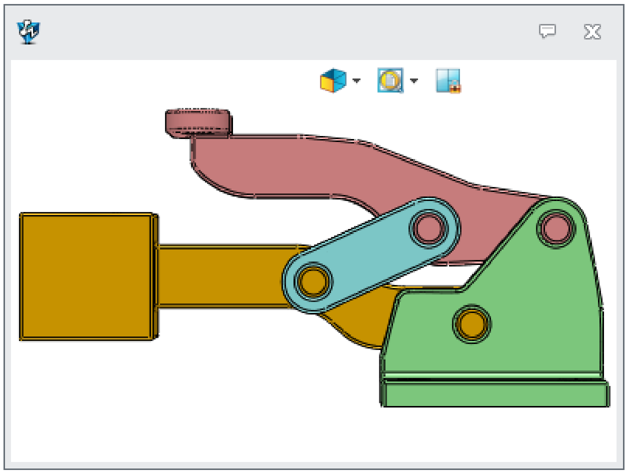
Target .0745,3.2735,44.9291

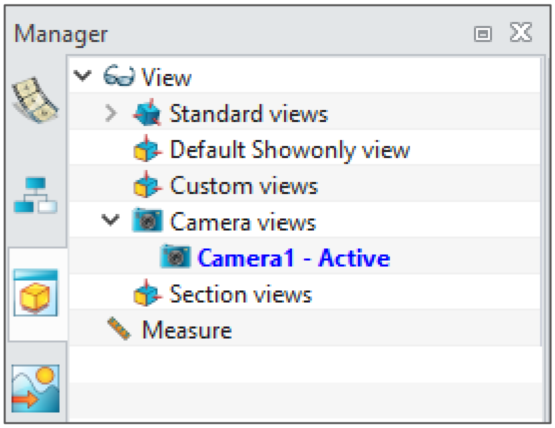
Visual Field

Up direction 0,0,1

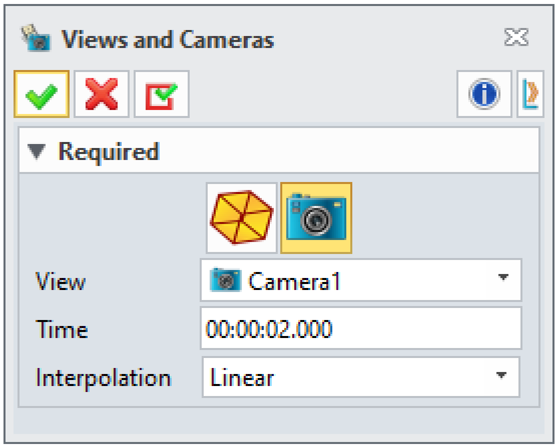
Extents 128 mm

☑ Preview




Manager
View
Standard views
Default Showonly view
Custom views
Camera views
Camera1 - Active
Section views
Measure

2. Open the “Views and Cameras” command, select “Camera keyframe”, select “Camera 1”, enter 2s in the time input box, and select "Linear" in the interpolation form. So, we can insert the keyframe to the arrow of camera 1 in the 2nd second in the timeline, which shows the view of the preview window when created.



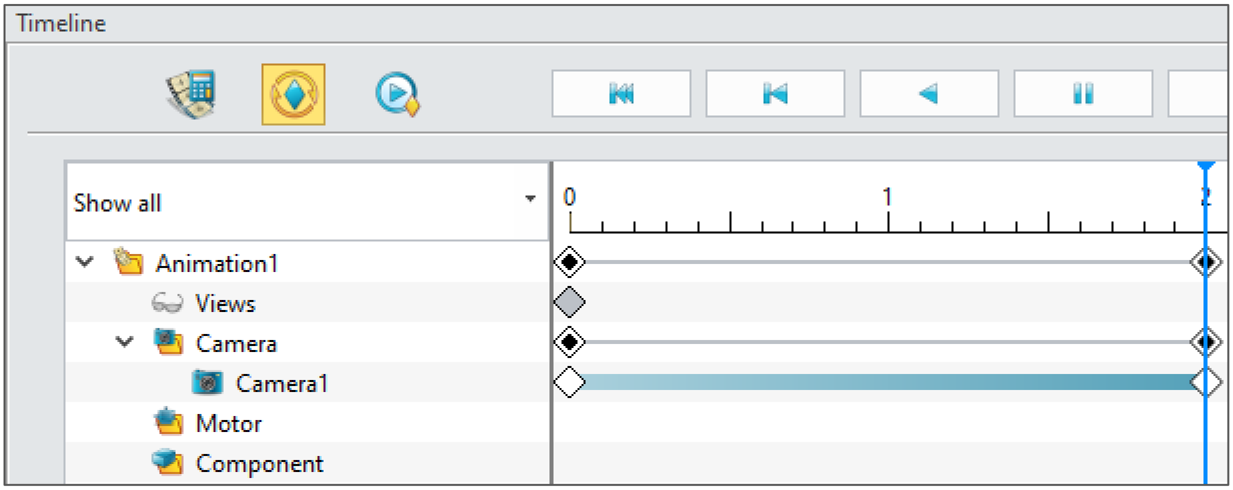
Views and Cameras

Required

View: Camera1

Time: 00:00:02.000

Interpolation: Linear



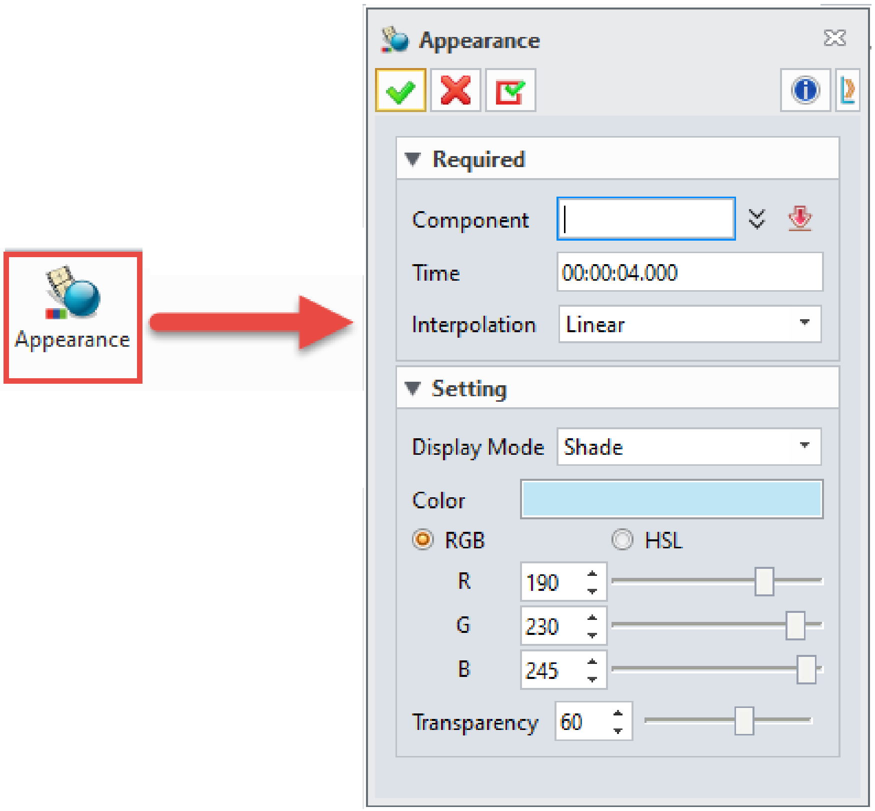
【Where is it】：

View Manager >> Camera Views >> New Camera

Assembly Environment >> Animation >> Key Frame >> Views and Cameras>>Camera Keyframe

3.4.6.4 New Appearance Keyframe

ZW3D 2025 can create the animation effect of appearance changes, including color, transparency, display style, and display/hide dynamic changes. It covers the scenarios where the user needs to change the transparency of shell parts or carry out dynamic color identification of highlighted parts when making product demonstration animations or installation instructions. It also covers the scenario where the internal structure of the device is viewed at different times by switching to display/hide.



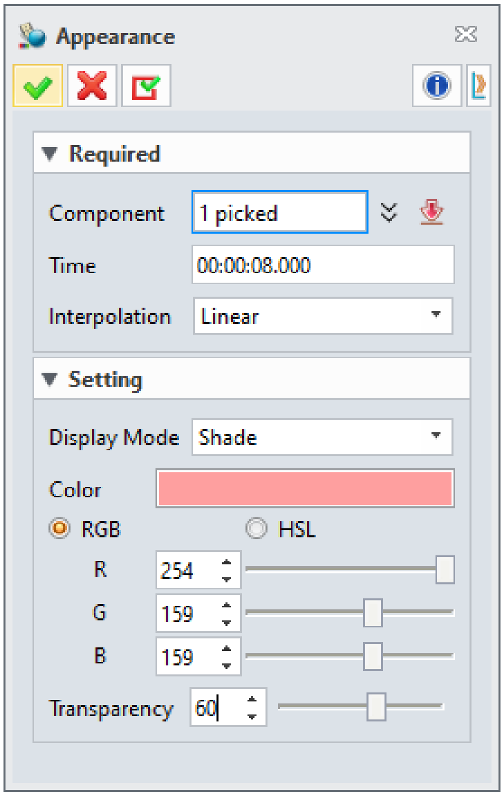
Component
Time 00:00:04.000
Interpolation Linear
Display Mode Shade
Color
● RGB ○ HSL
R 190
G 230
B 245
Transparency 60

【What can be done】:

- ✓ Create appearance keyframes for the change of display mode, color, and transparency.
- ✓ Support to modify the interpolation form of keyframes (linear and none).

【Example】:

1. Select a component shown in the picture, modify the transparency to 60, and enter 8s in the time input box, and select "Linear" in the interpolation form, which can make the transparency of the selected component change from 0 to 60 in the 8th second.



Appearance

Required

Component: 1 picked

Time: 00:00:08.000

Interpolation: Linear

Setting

Display Mode: Shade

Color

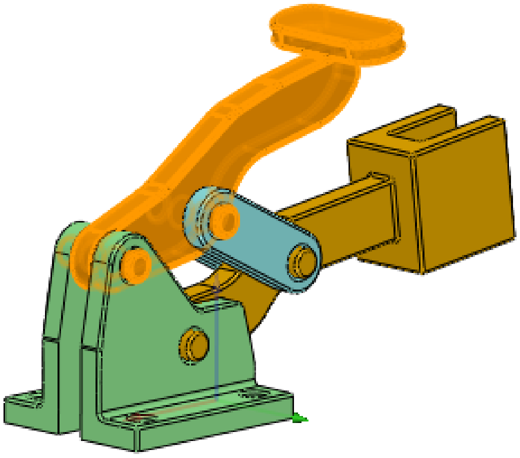
● RGB ○ HSL

R: 254

G: 159

B: 159

Transparency: 60

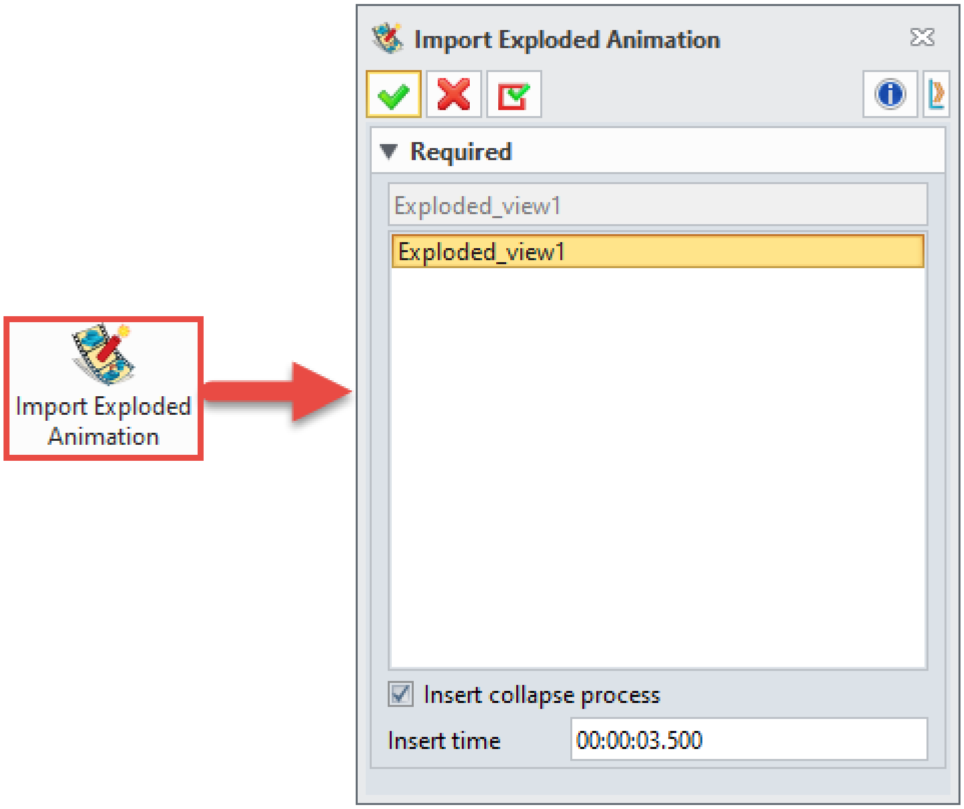


【Where is it】：

Assembly Environment >> Animation >> Key Frame >> Appearance

3.4.6.5 Support Importing Exploded Animation

Importing exploded animation is supported in ZW3D 2025. Users can insert explosion and collapse processes and define perspective and part appearance during the exploding process. It completely covers the production scenes of exploding animation and product internal structure display animation.



Exploded_view1

Exploded_view1

☑ Insert collapse process

Insert time 00:00:03.500

【What can be done】:

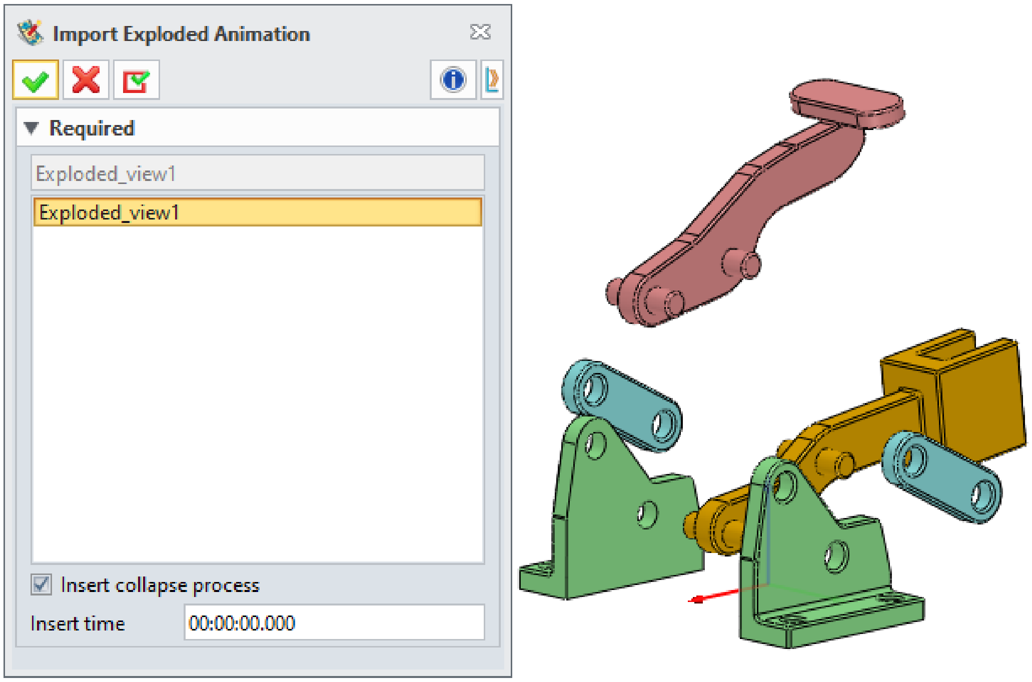
- ✓ Import the exploded animation created with the current configuration in the Exploded View.
- ✓ Insert the collapse process and the explosion process at the same time.
- ✓ Adjust the exploded time through time scaling in the timeline.

【Note】:

- It is not supported to edit the exploded data in the animation environment. Users need to edit it in the Explosion module before importing it.

【Example】:

1. Click “Import Exploded Animation”, select the exploded view to be imported in the list, check “Insert collapse process”, set the insert time to "3.5s", and click OK to import the exploded view into the current animation, as the figure shows below:



Exploded_view1

Exploded_view1

☑ Insert collapse process

Insert time 00:00:00.000

【Where is it】:

Assembly Environment >> Animation >> Import Exploded Animation

3.4.6.6 New Motion Analysis

ZW3D 2025 supports motion analysis, including measurement, band analysis, interference check, and motion track. Users can carry out simple design verification of the assembly model, which can let designers evaluate the accuracy of the model design more intuitively, reduce design mistakes, and lower prototype production costs.

【What can be done】:

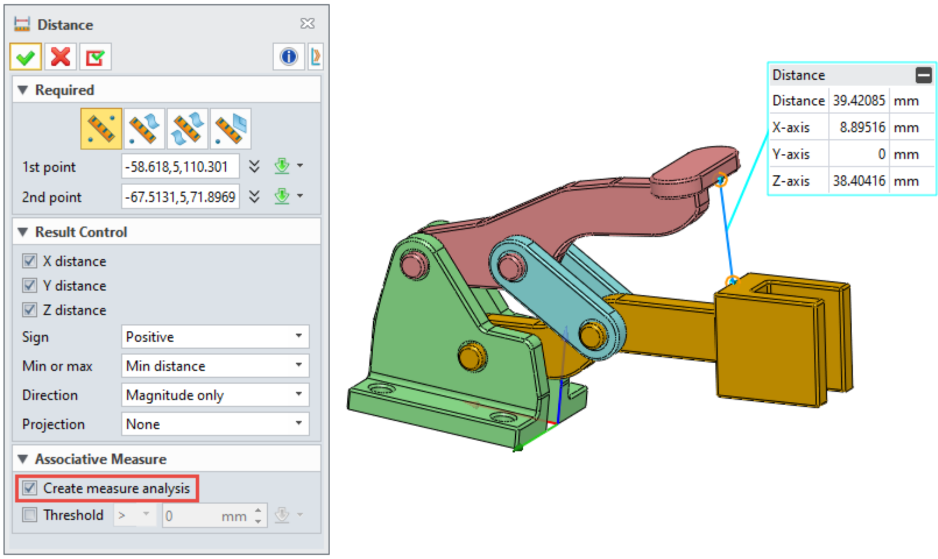
- ✓ Support real-time distance and angle measurement when the animation is playing.
- ✓ Support band analysis for the model components at any time in the animation.
- ✓ Support the real-time motion track drawing of selected points or lines when the animation is playing.
- ✓ Support inquiring about the interference check result of each keyframe.

【Note】:

- The distance between entities cannot be measured in real time.

【Example】:

1. Click the "Distance" command, pick two points at the component in drawing area, check "Create measure analysis", set other options as needed, and click "OK" to inquire about the result.



Distance

Required

1st point -58.618,5,110.301

2nd point -67.5131,5,71.8969

Result Control

☑ X distance

☑ Y distance

☑ Z distance

Sign: Positive

Min or max: Min distance

Direction: Magnitude only

Projection: None

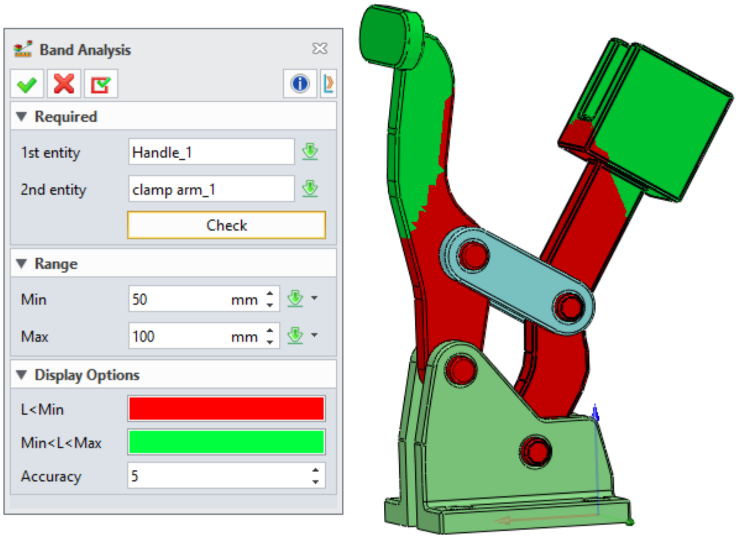
Associative Measure

☑ Create measure analysis

☐ Threshold > 0 mm

Distance		
Distance	39.42085	mm
X-axis	8.89516	mm
Y-axis	0	mm
Z-axis	38.40416	mm

2. Click the "Band Analysis" command, pick two components as shown in the pictures, set "min" to 50mm and "max" to 100mm, and click the "Check" button to view the analysis result. The red region indicates that the distance between two components is less than 50mm, while the green region suggests that the distance between two components is between 50mm and 100mm.



Band Analysis

Required

1st entity: Handle_1

2nd entity: clamp arm_1

Range

Min: 50 mm

Max: 100 mm

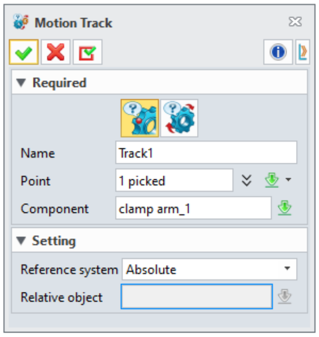
Display Options

L<Min: 🟥

Min<L<Max: 🟩

Accuracy: 5

3. Click the “Motion Track” command, select the point as shown in the following picture, and click OK to calculate and play the animation. The final track drawing is as follows:


Motion Track

Required

Name: Track1

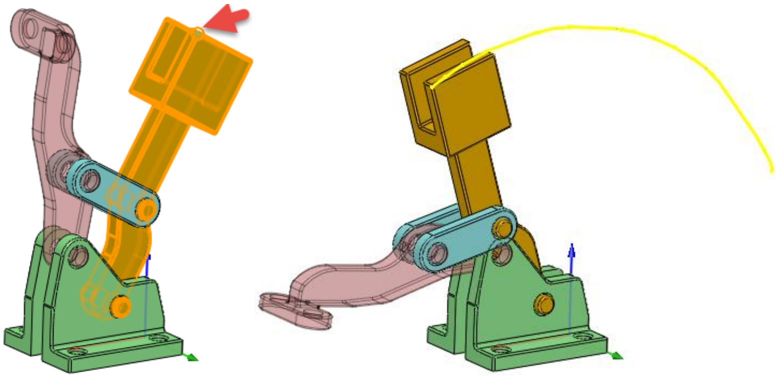
Point: 1 picked

Component: clamp arm_1

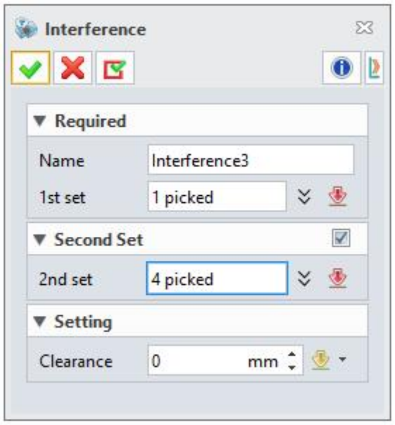
Setting

Reference system: Absolute

Relative object:



4. Click the “Interference Check” command, select the objects in the 1st set shown in the following picture as the first group objects for interference check and the other as the second group objects, and click "OK". Click the calculate button on the timeline to calculate the current interference item, switch the interference check drop-down box to "Interference on", and then the interfering components will be highlighted when the animation is playing.


Interference

Required

Name: Interference3

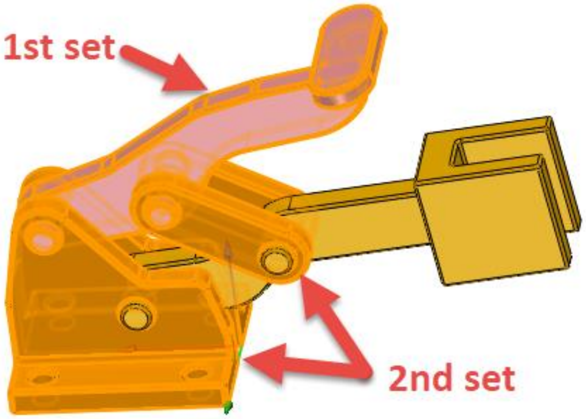
1st set: 1 picked

Second Set ☑

2nd set: 4 picked

Setting

Clearance: 0 mm



【Where is it】:

Assembly Environment >> Animation >> Animation Inquire >> Band Analysis/ Interference/ Motion Track

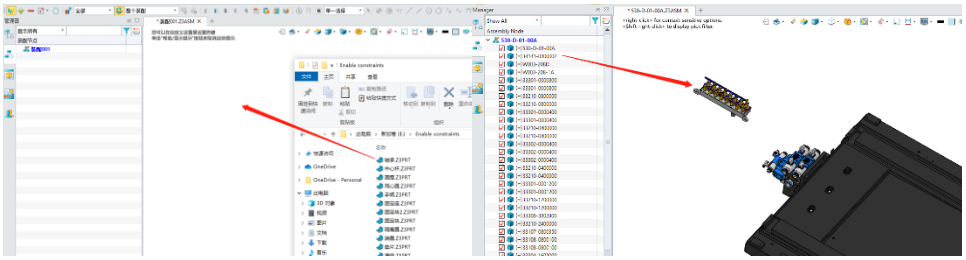
Assembly Environment >> Animation >>Inquire >> Distance/Angle

3.4.7 ★Assembly Component Quick Insertion

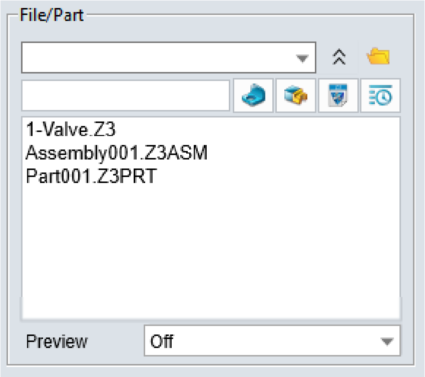
ZW3D 2025 has optimized the assembly insertion function, improved the display effect of UI, and provided a quick file insert function without activating any command.

【What can be done】:

- ✓ Quickly insert components by dragging them directly from the assembly tree/folder to the drawing area. Users can copy and paste components to the drawing area by pressing Ctrl + C and Ctrl + V.



- ✓ Recently used objects can be found in the Insert command drop-down list. Selectable objects can be displayed in the list box (the list box displays objects already opened in ZW3D by default). Besides, users can search for selectable objects in the list box, filter the corresponding component types, and display the files currently loaded in memory.



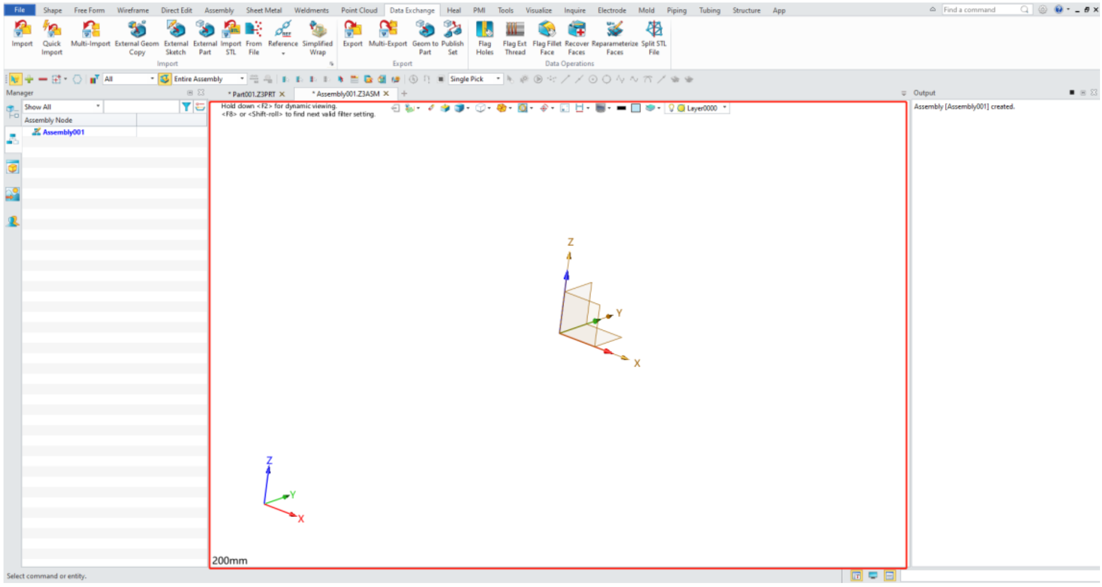
File/Part

1-Valve.Z3
Assembly001.Z3ASM
Part001.Z3PRT

Preview: Off

【Note】:

- Quick Insert only works by dragging/pasting components directly from the assembly to the drawing area.

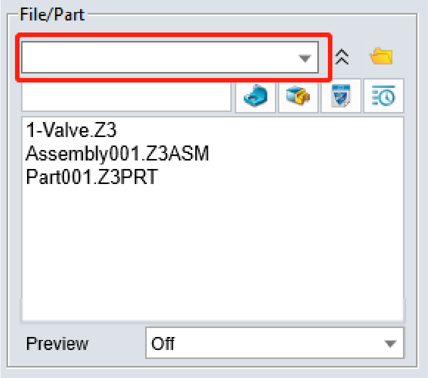


Hold down <F2> for dynamic viewing.
<F8> or <Shift-roll> to find next valid filter setting.

Output: Assembly [Assembly001] created.

File
Shape
Free Form
Wireframe
Direct Edit
Assembly
Sheet Metal
Weldments
Point Cloud
Data Exchange
Heal
PMI
Tools
Visualize
Inquire
Electrode
Mold
Piping
Tubing
Structure
App
Find a command
Import
Quick Import
Multi-Import
External Geom Copy
External Sketch
External Part
Import STL
From File
Reference
Simplified Wrap
Export
Multi-Export
Geom to Part
Publish Set
Flag Holes
Flag Ext Thread
Flag Fillet Face
Recover Faces
Reparameterize Faces
Split STL File
Data Operations
Entire Assembly
Single Pick
Layer0000
Manager
Show All
Assembly Node
Assembly001
* Part001.Z3PRT
* Assembly001.Z3ASM
200mm
Select command or entity.

- In the Insert command, the automatic filling function is removed, and users need to manually select the inserted file objects. At the same time, these changes are applied synchronously to other commands, such as layout/standard view of a drawing sheet projection, drawing sheet block, sketch block, copy external geometry, external sketch, external part, copy geometry to parts, etc.



File/Part

1-Valve.Z3
Assembly001.Z3ASM
Part001.Z3PRT

Preview: Off

【Where is it】：

Assembly Environment >> Component >> Insert/Muti Insert/Include Unplaced Component

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

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

Sketch Environment >> Sub-Sketch>>Insert Block

Data Exchange >>Copy External Geom/ External Sketch/External Part/ Copy geometry to a part

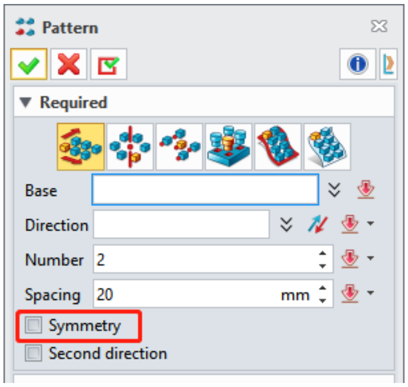
3.4.8 Assembly Pattern

The assembly pattern supports symmetry in the linear mode. In addition, the assembly pattern can reference the linear/circular pattern in the Part environment and link it correctly.

Variable spacing pattern is added in ZW3D 2025 SP, which allows you to fill in the spacing in a table to create variable spacing pattern.

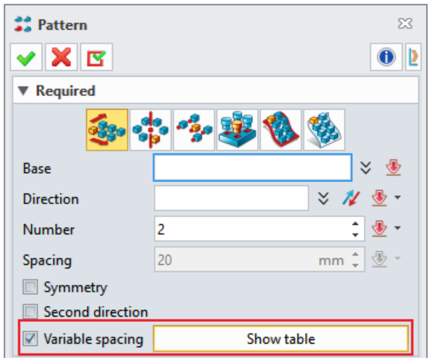
【What can be done】:

- ✓ Quickly generate the same pattern in the opposite direction by checking "Symmetry".

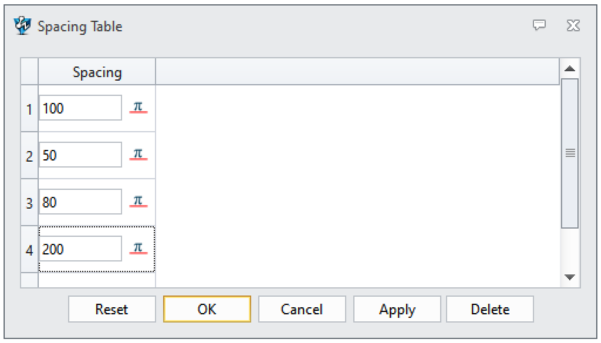


Base
Direction
Number 2
Spacing 20 mm
☐ Symmetry
☐ Second direction

- ✓ By checking the variable spacing box and filling in the spacing table, you can quickly generate an variable spacing pattern in the specified direction.

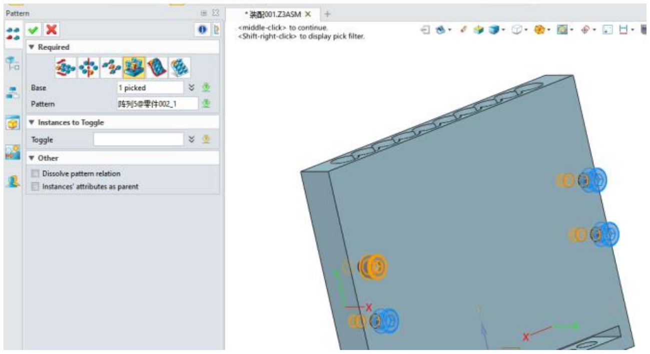


Base
Direction
Number 2
Spacing 20 mm
☐ Symmetry
☐ Second direction
☑ Variable spacing



	Spacing
1	100
2	50
3	80
4	200

- ✓ Users can create the assembly pattern quickly by referencing the linear/circular pattern in the Part environment. Dragging linkage is supported in the Assembly environment. After the parameters of the part pattern are modified, the parameters of the assembly pattern can be updated synchronously by Regeneration.



Pattern

▼ Required

Base: 1 picked

Pattern: 阵列5@零件002_1

▼ Instances to Toggle

Toggle:

▼ Other

☐ Dissolve pattern relation

☐ Instances' attributes as parent

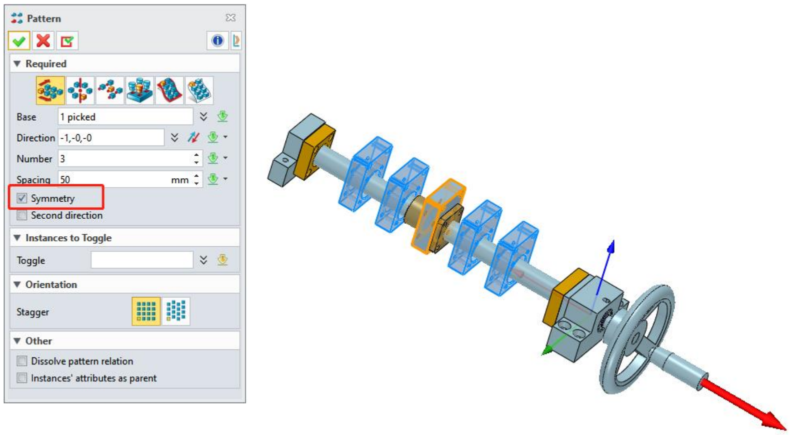
* 装配001.Z3ASM

<middle-click> to continue.

<Shift-right-click> to display pick filter.

【Example】:

Activate the Pattern command, select the component and direction of the pattern, set the distance, check the “Symmetry” option, and the preview will show that the components will be generated symmetrically.



Pattern

▼ Required

Base: 1 picked

Direction: -1,-0,-0

Number: 3

Spacing: 50 mm

☑ Symmetry

☐ Second direction

▼ Instances to Toggle

Toggle:

▼ Orientation

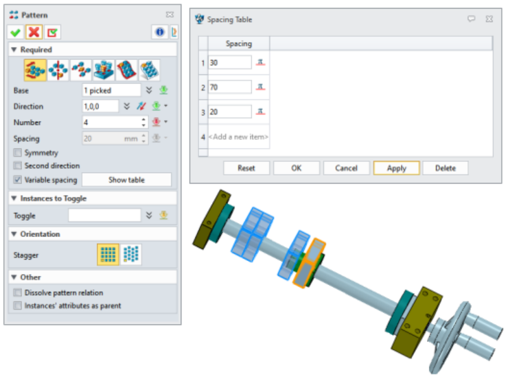
Stagger

▼ Other

☐ Dissolve pattern relation

☐ Instances' attributes as parent

After selecting the components and direction of the pattern, select variable spacing, set the spacing between the pattern components, and then preview the components to be generated according to the set spacing.


Pattern
Required
Show table
Instances to Toggle
Orientation
Stagger
Other
Spacing Table
Reset
OK
Cancel
Apply
Delete

Base: 1 picked
Direction: 1,0,0
Number: 4
Spacing: 20 mm
☐ Symmetry
☐ Second direction
☑ Variable spacing
Toggle:
☐ Dissolve pattern relation
☐ Instances' attributes as parent

	Spacing
1	30
2	70
3	20
4	<Add a new item>

【Where is it】:

Assembly Environment >> Pattern

3.4.9 Assembly Mirror

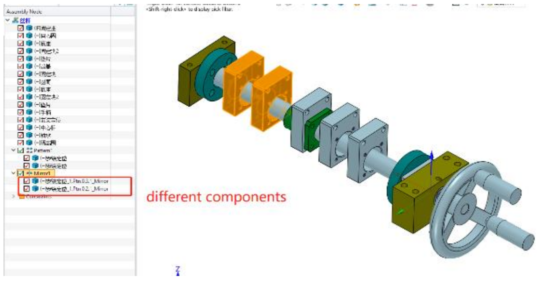
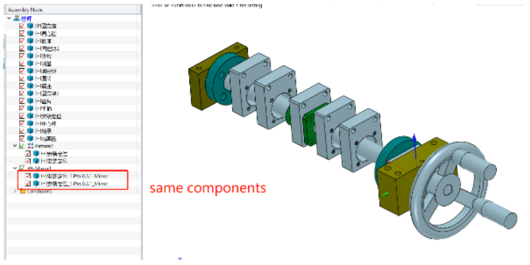
When creating new components by Copy, users can select multiple identical components with only one file generated, which reduces the number of files.

【What can be done】:

- ✓ When using the Mirror command, the existing mirror copy components can be reused to make the identical components generate a single object, which reduces the number of new files generated.

【Example】:

If you need to completely mirror multiple identical components without generating multiple parts, check two identical components and select mirror plane to create.

ZW3D 2024	ZW3D 2025
 different components	 same components

【Where is it】:

Assembly Environment >> Mirror

3.4.10 Assembly Tree Sorting

The assembly tree supports sorting, which quickly sorts the assembly tree according to the specified rules, helping users find the required component objects quickly.

【What can be done】:

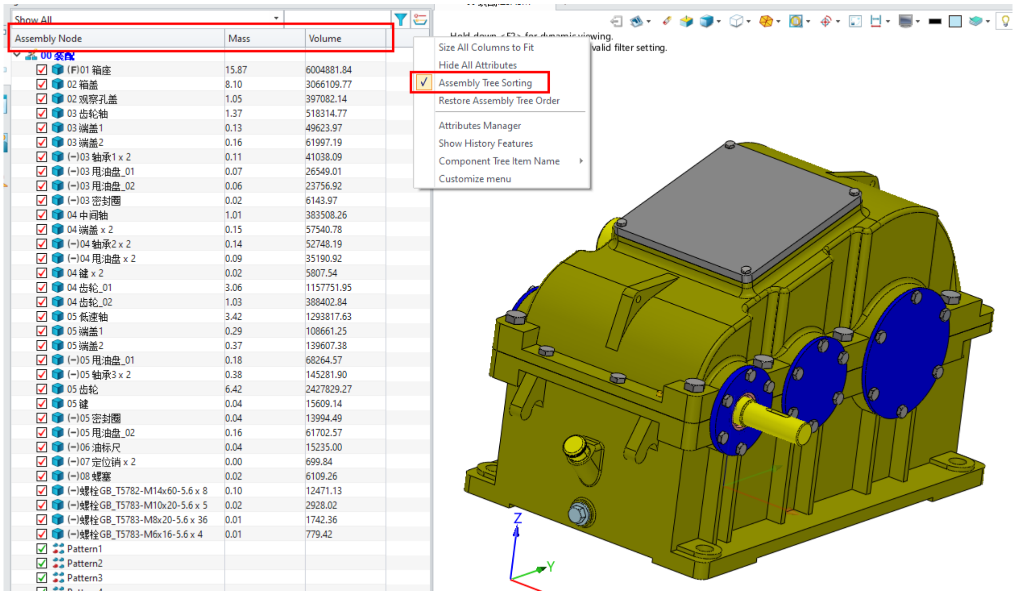
- ✓ Users can sort the top-level component in the assembly tree according to the specified rules, such as name, quality, volume, density, length, width, height, etc.

【Note】:

- It can only sort the top-level components, not the cross-level components.

【Example】:

Open an assembly file, display the attributes that need to be sorted in the assembly tree, and right-click the top of the assembly tree to start the assembly tree sorting. Click different table headers to sort according to the corresponding header rules, and click again to toggle between ascending sorting and descending sorting. The first column headers are sorted by the component name order.



Show All

Assembly Node	Mass	Volume
00 装配		
☑ (F)01 箱座	15.87	6004881.84
☑ 02 箱盖	8.10	3066109.77
☑ 02 观察孔盖	1.05	397082.14
☑ 03 齿轮轴	1.37	518314.77
☑ 03 端盖1	0.13	49623.97
☑ 03 端盖2	0.16	61997.19
☑ (−)03 轴承1 x 2	0.11	41038.09
☑ (−)03 甩油盘_01	0.07	26549.01
☑ (−)03 甩油盘_02	0.06	23756.92
☑ (−)03 密封圈	0.02	6143.97
☑ 04 中间轴	1.01	383508.26
☑ 04 端盖 x 2	0.15	57540.78
☑ (−)04 轴承2 x 2	0.14	52748.19
☑ (−)04 甩油盘 x 2	0.09	35190.92
☑ 04 键 x 2	0.02	5807.54
☑ 04 齿轮_01	3.06	1157751.95
☑ 04 齿轮_02	1.03	388402.84
☑ 05 低速轴	3.42	1293817.63
☑ 05 端盖1	0.29	108661.25
☑ 05 端盖2	0.37	139607.38
☑ (−)05 甩油盘_01	0.18	68264.57
☑ (−)05 轴承3 x 2	0.38	145281.90
☑ 05 齿轮	6.42	2427829.27
☑ 05 键	0.04	15609.14
☑ (−)05 密封圈	0.04	13994.49
☑ (−)05 甩油盘_02	0.16	61702.57
☑ (−)06 油标尺	0.04	15235.00
☑ (−)07 定位销 x 2	0.00	699.84
☑ (−)08 螺塞	0.02	6109.26
☑ (−)螺栓GB_T5782-M14x60-5.6 x 8	0.10	12471.13
☑ (−)螺栓GB_T5783-M10x20-5.6 x 5	0.02	2928.02
☑ (−)螺栓GB_T5783-M8x20-5.6 x 36	0.01	1742.36
☑ (−)螺栓GB_T5783-M6x16-5.6 x 4	0.01	779.42
☑ Pattern1		
☑ Pattern2		
☑ Pattern3		

Hold down <F2> for dynamic viewing.

valid filter setting.

【Where is it】：

Assembly Environment >> Assembly Manager

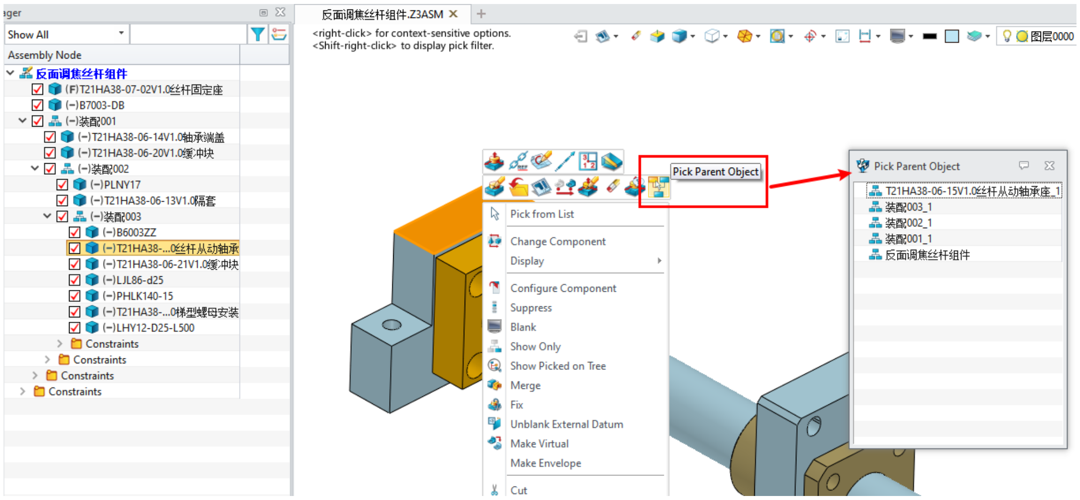
3.4.11 Assembly Drawing Area Interface

【What can be done】：

- ✓ Right-click the selected object in the drawing area to check the parent object of it, which can be traced back to the current general assembly.

【Example】：

Right-click the selected object in the drawing area, click "Pick Parent Object" in the right-click menu, and you can see the different-level parent object of the selected object in the pop-up window.


ager
Show All
Assembly Node
反面调焦丝杆组件
(F)T21HA38-07-02V1.0丝杆固定座
(−)B7003-DB
(−)装配001
(−)T21HA38-06-14V1.0轴承端盖
(−)T21HA38-06-20V1.0缓冲块
(−)装配002
(−)PLNY17
(−)T21HA38-06-13V1.0隔套
(−)装配003
(−)B6003ZZ
(−)T21HA38-...0丝杆从动轴承
(−)T21HA38-06-21V1.0缓冲块
(−)LJL86-d25
(−)PHLK140-15
(−)T21HA38-...0梯型螺母安装
(−)LHY12-D25-L500
Constraints
Constraints
Constraints
Constraints
反面调焦丝杆组件.Z3ASM
<right-click> for context-sensitive options.
<Shift-right-click> to display pick filter.
图层0000
Pick Parent Object
Pick from List
Change Component
Display
Configure Component
Suppress
Blank
Show Only
Show Picked on Tree
Merge
Fix
Unblank External Datum
Make Virtual
Make Envelope
Cut
Pick Parent Object
T21HA38-06-15V1.0丝杆从动轴承座_1
装配003_1
装配002_1
装配001_1
反面调焦丝杆组件

【Where is it】：

Assembly Environment >> Drawing Area

3.4.12 Assembly Tree Optimization

【What can be done】：

1. When switching settings for multi-configuration components, the original component configuration will be switched synchronously.

2. Set components to virtual components/flexible sub-assemblies/lightweight components at multi-

levels in batches.

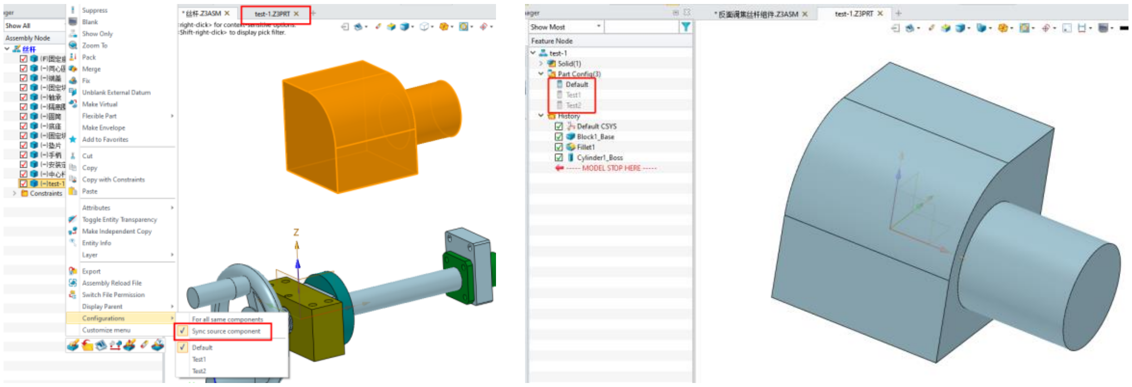
3. The multi-column tree displays values with two decimal places by default, and the displayed decimal places can be customized.

【Note】:

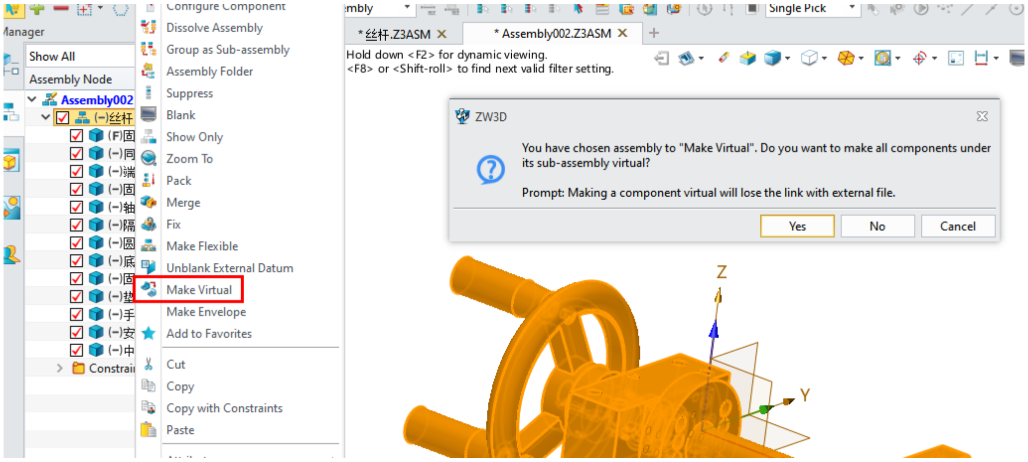
- The shape in assembly is not supported to be lightweight, and the setting behavior of the assembly is to set all the child components that don't include assembly to be lightweight.

【Example】:

1. When switching settings for multi-configuration components, the original component configuration will be switched synchronously. Open the assembly file and the part file at the same time, select components under Assembly, check the option "Sync source component" in right-click menu, then switch the configuration, and you can find that the opened part file is also switched to the corresponding configuration.


Show All
Assembly Node
Constraints
Suppress
Blank
Show Only
Zoom To
Pack
Merge
Fix
Unblank External Datum
Make Virtual
Flexible Part
Make Envelope
Add to Favorites
Cut
Copy
Copy with Constraints
Paste
Attributes
Toggle Entity Transparency
Make Independent Copy
Entity Info
Layer
Export
Assembly Reload File
Switch File Permission
Display Parent
Configurations
Customize menu
test-1.Z3PRT
For all same components
Sync source component
Default
Test1
Test2
Z
Show Most
Feature Node
test-1
Solid(1)
Part Configs(3)
Default
Test1
Test2
History
Default CSYS
Block1_Base
Fillet1
Cylinder1_Boss
----- MODEL STOP HERE -----
test-1.Z3PRT

2. Set components to virtual components at multi-levels in batches. When an assembly is set up as virtual component, it will pop up a dialog allowing the user to choose whether all components in this assembly need to be set up as virtual components at once. Select "Yes" to set the assembly and all components in this assembly to virtual components in batches. Select "No" to set the corresponding component to virtual separately.

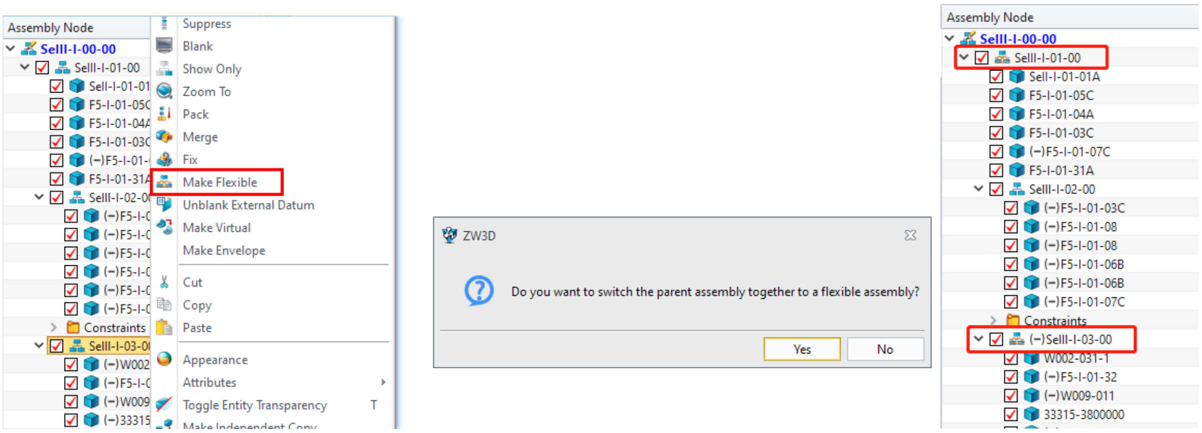

Manager
Show All
Assembly Node
Assembly002
Constrai
Configure Component
Dissolve Assembly
Group as Sub-assembly
Assembly Folder
Suppress
Blank
Show Only
Zoom To
Pack
Merge
Fix
Make Flexible
Unblank External Datum
Make Virtual
Make Envelope
Add to Favorites
Cut
Copy
Copy with Constraints
Paste
Single Pick
*丝杆.Z3ASM
* Assembly002.Z3ASM
ZW3D
Yes
No
Cancel
Z
Y

Hold down <F2> for dynamic viewing.
<F8> or <Shift-roll> to find next valid filter setting.

You have chosen assembly to "Make Virtual". Do you want to make all components under its sub-assembly virtual?

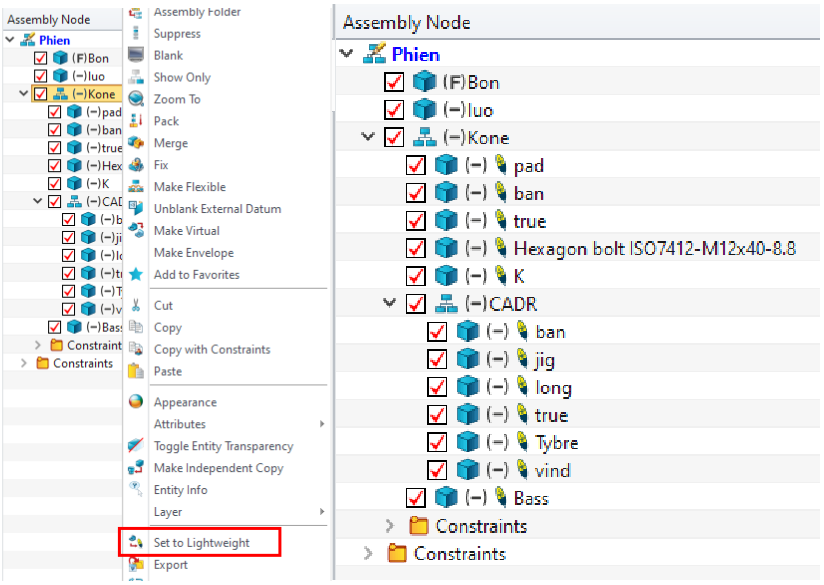
Prompt: Making a component virtual will lose the link with external file.

3. Flexible sub-assemblies can be set across levels. You can directly right-click the sub-assembly that you want to set to flexible (the lowest sub-assembly in the multi-layer assembly), and the pop-up dialog will prompt whether to set the parent assembly to a flexible assembly at the same time. Select “Yes” to set both the selected assembly and the parent assembly of it to flexible. Select “No” to only switch the selected component to flexible.

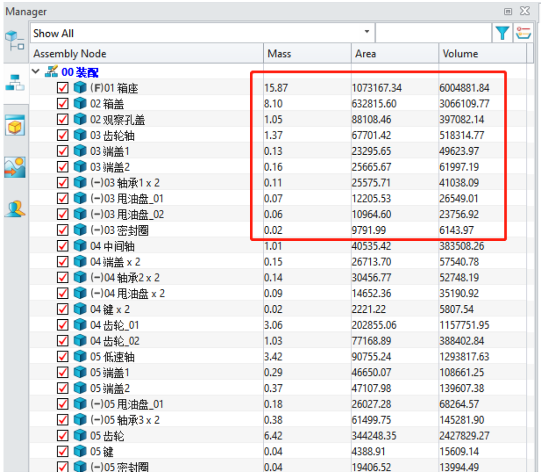

Assembly Node
SellI-I-00-00
SellI-I-01-00
Sell-I-01-01A
F5-I-01-05C
F5-I-01-04A
F5-I-01-03C
(−)F5-I-01-07C
F5-I-01-31A
SellI-I-02-00
(−)F5-I-01-03C
(−)F5-I-01-08
(−)F5-I-01-08
(−)F5-I-01-06B
(−)F5-I-01-06B
(−)F5-I-01-07C
Constraints
(−)SellI-I-03-00
W002-031-1
(−)F5-I-01-32
(−)W009-011
33315-3800000
Suppress
Blank
Show Only
Zoom To
Pack
Merge
Fix
Make Flexible
Unblank External Datum
Make Virtual
Make Envelope
Cut
Copy
Paste
Appearance
Attributes
Toggle Entity Transparency
T
Make Independent Copy
ZW3D
Yes
No

Do you want to switch the parent assembly together to a flexible assembly?

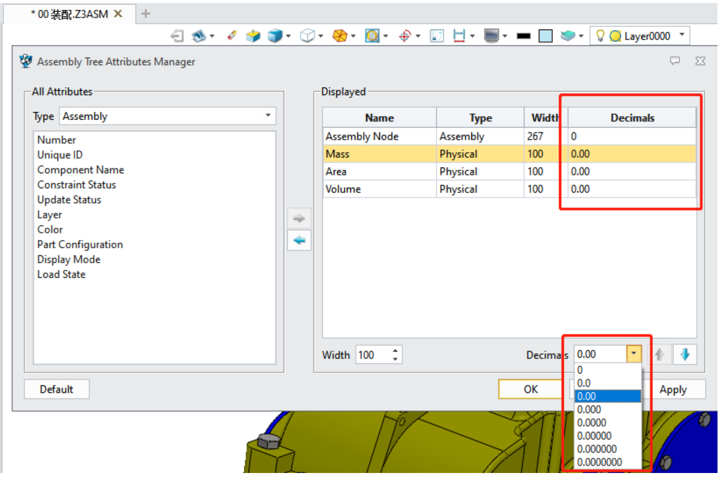
4. Set components to lightweight at multi-levels in batches. After “Enable lightweight cache” is enabled in Configuration, right-click the target assembly and choose “Set to Lightweight” from the right-click menu to convert all components at the lower level of the target assembly to lightweight components in batches.


Assembly Node
Phien
(F)Bon
(−)luo
(−)Kone
(−)pad
(−)ban
(−)true
(−)Hex
(−)K
(−)CAD
Constraint
Constraints
Assembly Folder
Suppress
Blank
Show Only
Zoom To
Pack
Merge
Fix
Make Flexible
Unblank External Datum
Make Virtual
Make Envelope
Add to Favorites
Cut
Copy
Copy with Constraints
Paste
Appearance
Attributes
Toggle Entity Transparency
Make Independent Copy
Entity Info
Layer
Set to Lightweight
Export
Assembly Node
Phien
(F)Bon
(−)luo
(−)Kone
(−) pad
(−) ban
(−) true
(−) Hexagon bolt ISO7412-M12x40-8.8
(−) K
(−)CADR
(−) ban
(−) jig
(−) long
(−) true
(−) Tybre
(−) vind
(−) Bass
Constraints
Constraints

5. Set the multi-column tree content to be displayed by Attribute Manager at the top of the assembly tree when needing to view the quality, volume, and other information of components. You can check the decimal places of the corresponding displayed content and adjust the decimal places. A maximum of seven decimal places can be shown.


Manager
Show All

Assembly Node	Mass	Area	Volume
00 装配			
(F)01 箱座	15.87	1073167.34	6004881.84
02 箱盖	8.10	632815.60	3066109.77
02 观察孔盖	1.05	88108.46	397082.14
03 齿轮轴	1.37	67701.42	518314.77
03 端盖1	0.13	23295.65	49623.97
03 端盖2	0.16	25665.67	61997.19
(−)03 轴承1 x 2	0.11	25575.71	41038.09
(−)03 甩油盘_01	0.07	12205.53	26549.01
(−)03 甩油盘_02	0.06	10964.60	23756.92
(−)03 密封圈	0.02	9791.99	6143.97
04 中间轴	1.01	40535.42	383508.26
04 端盖 x 2	0.15	26713.70	57540.78
(−)04 轴承2 x 2	0.14	30456.77	52748.19
(−)04 甩油盘 x 2	0.09	14652.36	35190.92
04 键 x 2	0.02	2221.22	5807.54
04 齿轮_01	3.06	202855.06	1157751.95
04 齿轮_02	1.03	77168.89	388402.84
05 低速轴	3.42	90755.24	1293817.63
05 端盖1	0.29	46650.07	108661.25
05 端盖2	0.37	47107.98	139607.38
(−)05 甩油盘_01	0.18	26027.28	68264.57
(−)05 轴承3 x 2	0.38	61499.75	145281.90
05 齿轮	6.42	344248.35	2427829.27
05 键	0.04	4388.91	15609.14
(−)05 密封圈	0.04	19406.52	13994.49

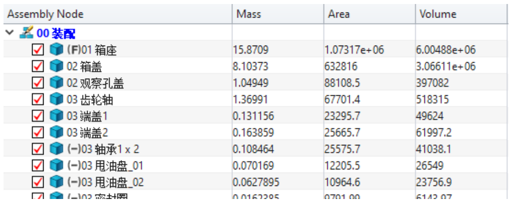

* 00 装配.Z3ASM
Layer0000
Assembly Tree Attributes Manager
All Attributes
Number
Unique ID
Component Name
Constraint Status
Update Status
Layer
Color
Part Configuration
Display Mode
Load State
Displayed
Default
OK
Apply
0
0.0
0.00
0.000
0.0000
0.00000
0.000000
0.0000000

Type: Assembly

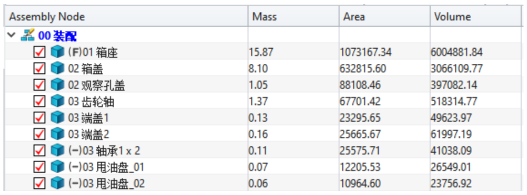
Name	Type	Width	Decimals
Assembly Node	Assembly	267	0
Mass	Physical	100	0.00
Area	Physical	100	0.00
Volume	Physical	100	0.00

Width: 100
Decimals: 0.00

ZW3D 2024	ZW3D 2025



Assembly Node	Mass	Area	Volume
00 装配			
☑ (F)01 箱座	15.8709	1.07317e+06	6.00488e+06
☑ 02 箱盖	8.10373	632816	3.06611e+06
☑ 02 观察孔盖	1.04949	88108.5	397082
☑ 03 齿轮轴	1.36991	67701.4	518315
☑ 03 端盖1	0.131156	23295.7	49624
☑ 03 端盖2	0.163859	25665.7	61997.2
☑ (−)03 轴承1 x 2	0.108464	25575.7	41038.1
☑ (−)03 甩油盘_01	0.070169	12205.5	26549
☑ (−)03 甩油盘_02	0.0627895	10964.6	23756.9



Assembly Node	Mass	Area	Volume
00 装配			
☑ (F)01 箱座	15.87	1073167.34	6004881.84
☑ 02 箱盖	8.10	632815.60	3066109.77
☑ 02 观察孔盖	1.05	88108.46	397082.14
☑ 03 齿轮轴	1.37	67701.42	518314.77
☑ 03 端盖1	0.13	23295.65	49623.97
☑ 03 端盖2	0.16	25665.67	61997.19
☑ (−)03 轴承1 x 2	0.11	25575.71	41038.09
☑ (−)03 甩油盘_01	0.07	12205.53	26549.01
☑ (−)03 甩油盘_02	0.06	10964.60	23756.92

【Where is it】:

Assembly Environment >> Assembly Manager

3.4.13 BOM Function Optimization

The BOM attribute setting “Indivisible part” has been added to Assembly Properties. For some overall procurement models, it is only necessary to show the corresponding assembly in the BOM.

3D BOM Added the legend property in ZW3D 2025 SP, enabling you to view thumbnails of assembly components in 3D BOM.

【What can be done】:

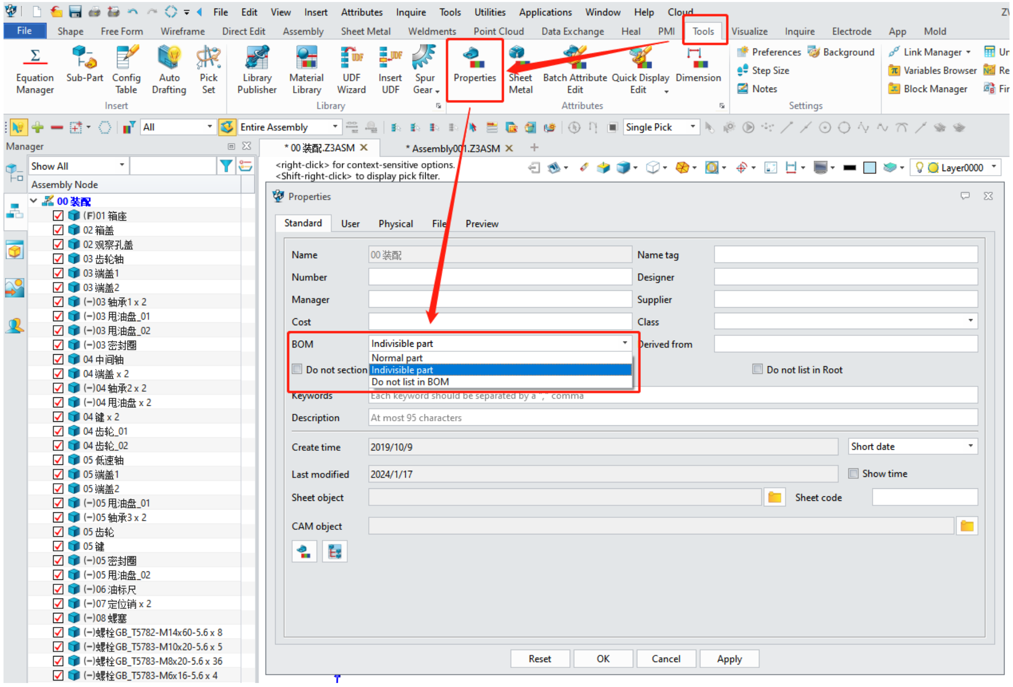
- ✓ Support only showing the assembly itself, not the components within in the 3D BOM.
- ✓ View thumbnails of assembly components in 3D BOM.

【Note】:

- “Indivisible part” is valid only for assemblies, parts can’t be set to Indivisible.

【Example】:

Open the assembly file which needs to be non-detachable and set the assembly attribute to “Indivisible part” in Properties. Opening the parent assembly file, you will find that only the corresponding assembly is shown in 3D BOM without the interior components in the assembly.


Tools
Properties
Standard
User
Physical
File
Preview
Reset
OK
Cancel
Apply

Name: 00 装配

BOM: Indivisible part

Normal part
Indivisible part
Do not list in BOM

☐ Do not section

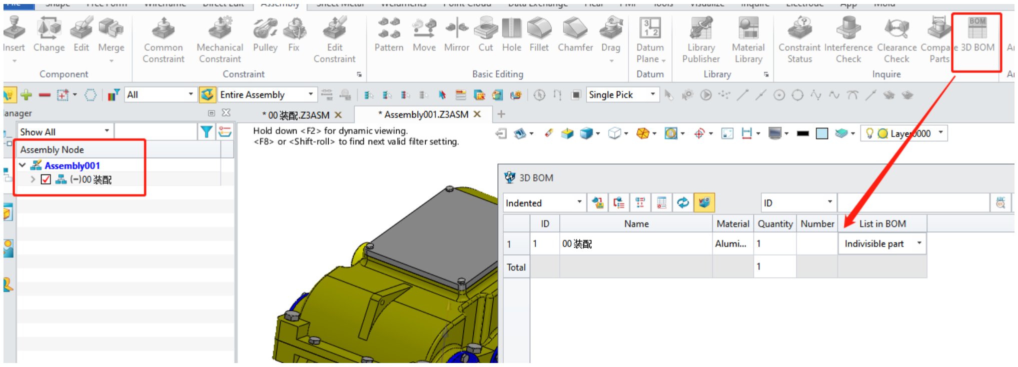
☐ Do not list in Root

Keywords: Each keyword should be separated by a ',' comma

Description: At most 95 characters

Create time: 2019/10/9 Short date

Last modified: 2024/1/17 ☐ Show time


3D BOM
Assembly Node
Assembly001
(−)00 装配
Indented
ID

	ID	Name	Material	Quantity	Number	List in BOM
1	1	00 装配	Alumi...	1		Indivisible part
Total				1		

【Where is it】:

Tools >> Properties

Assembly Environment >> 3D BOM

3.4.14 Smart Fastener optimization（ZW3D 2025 SP）

【What can be done】:

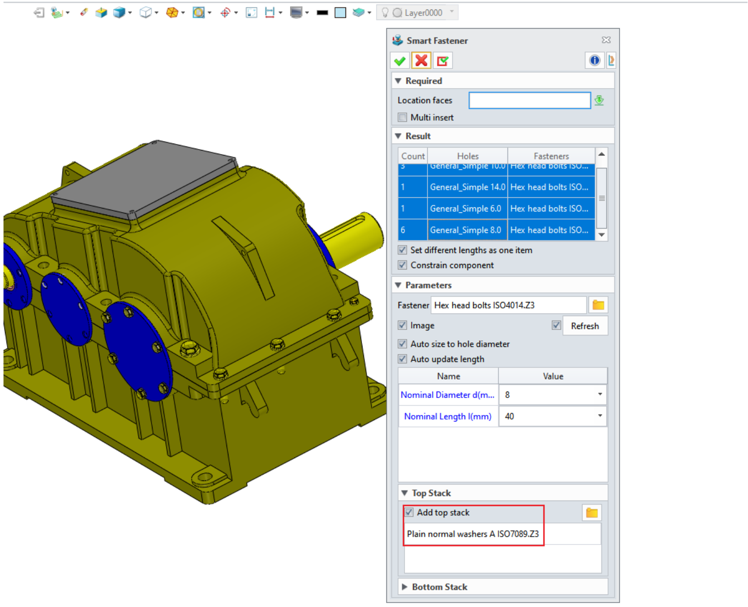
- ✓ Supports adding nuts and gaskets in batches for fasteners of different sizes, and modifying the length and nominal diameter parameters of fasteners in batches.

【Note】:

- If the selected objects contain fasteners that do not support adding stack or fasteners of different types, the selected items will be disabled. Batch change is not allowed.

【Example】:

Hold down Ctrl to select the fasteners that need to be gasketed and select "Add to Top Stack" to add gaskets for selected fasteners of different sizes at one time.



Smart Fastener

Required

Location faces

☐ Multi insert

Result

Count	Holes	Fasteners
3	General_Simple 10.0	Hex head bolts ISO...
1	General_Simple 14.0	Hex head bolts ISO...
1	General_Simple 6.0	Hex head bolts ISO...
6	General_Simple 8.0	Hex head bolts ISO...

☑ 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...	8
Nominal Length l(mm)	40

Top Stack

☑ Add top stack

Plain normal washers A ISO7089.Z3

Bottom Stack

【Where is it】:

Assembly Environment >> Smart Fastener

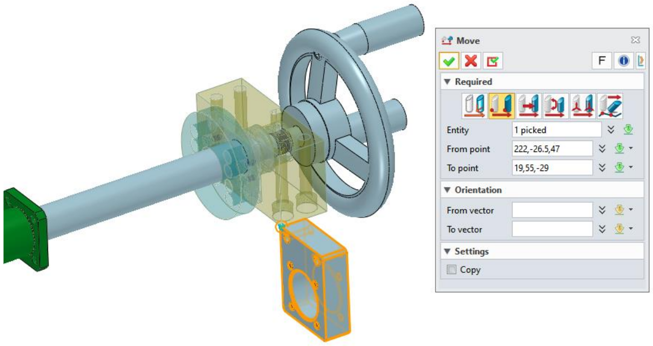
3.4.15 Assembly Movement Optimization（ZW3D 2025 SP）

【What can be done】:

- ✓ The move command supports cross-level selection of target locations.

【Example】:

When you select a component to move, you can move it point-to-point. You can directly select objects on other level components as the reference for moving, and quickly move the component.



Entity: 1 picked
From point: 222,-26.5,47
To point: 19,55,-29
From vector:
To vector:
☐ Copy

【Where is it】:

Assembly Environment >> Move

3.4.16 Component Display Optimization（ZW3D 2025 SP）

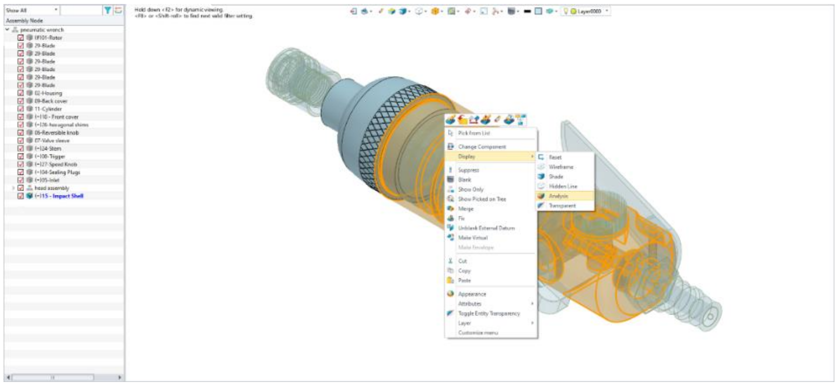
When editing components in place, other components other than this environment can change the display mode and record it in the context of the current final assembly file. After the final installation environment changes the display mode of the component, when double-clicking other components for on-the-site editing, the off-site editing component still records the changed display mode.

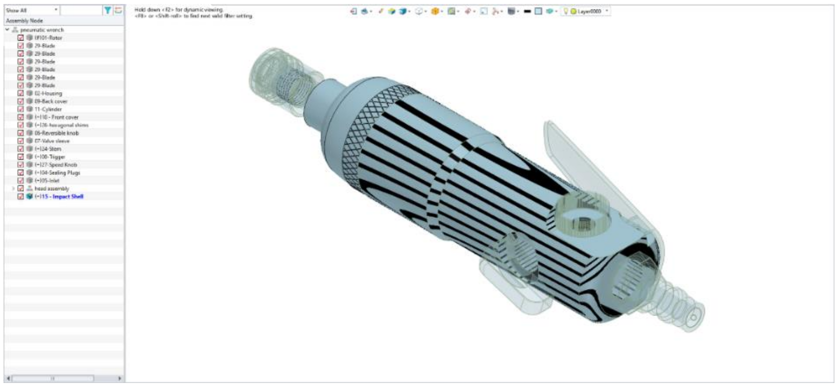
【What can be done】:

- ✓ When editing components in place, other components that are not in this environment can change the display mode.
- ✓ The display mode of non-in-place editing components is still recorded in the assembly environment.

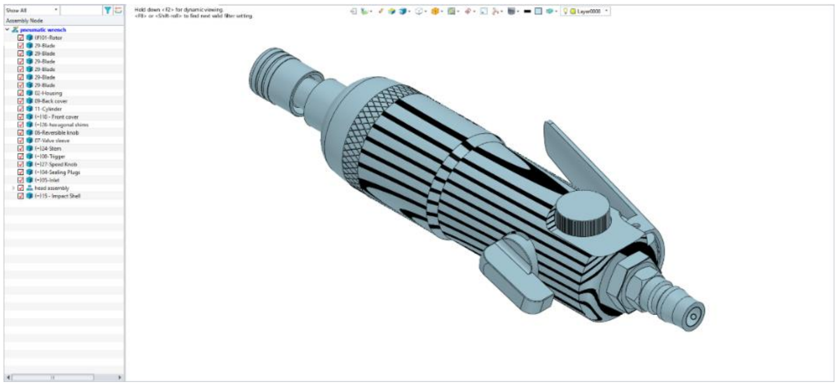
【Example】:

1. Double-clicking any component to enter the in-place editing environment can change the display mode of non-native components.

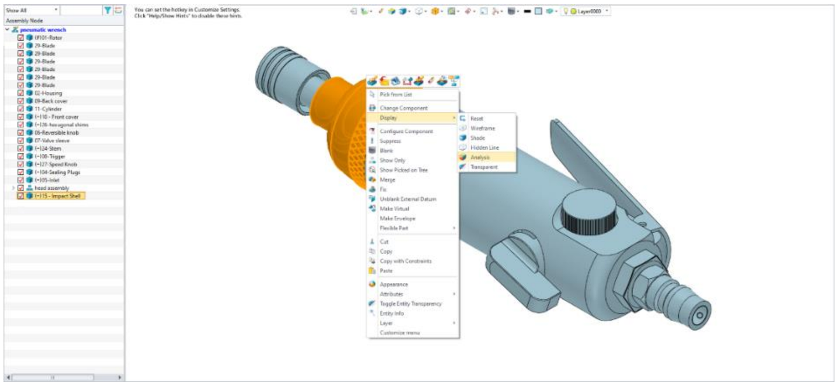

Pick From List
Change Component
Display
Suppress
Blank
Show Only
Show Picked on Tree
Merge
Fix
Unblank External Datum
Make Virtual
Make Envelope
Cut
Copy
Paste
Appearance
Attributes
Toggle Entity Transparency
Layer
Customize menu
Reset
Wireframe
Shade
Hidden Line
Analysis
Transparent



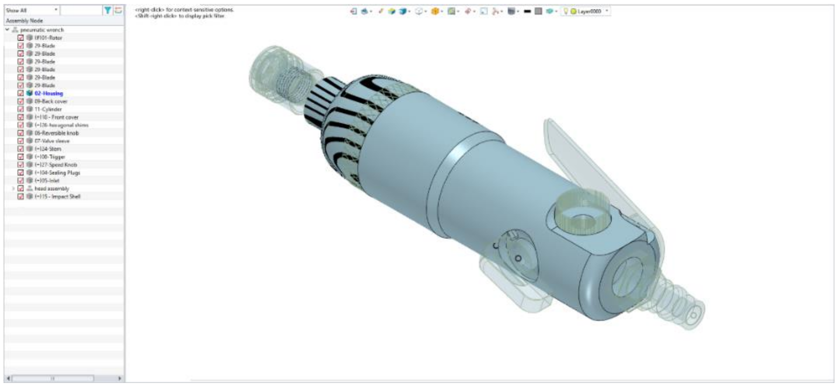
2. Return to the assembly environment and still record the display mode changed by the non-in-site editing component.



3. Change the display mode of the component in the assembly environment.


You can set the hotkey in Customize Settings.
Click "Help/Show Hints" to disable these hints.
Show All
Assembly Node
pneumatic wrench
UF101-Rotor
29-Blade
29-Blade
29-Blade
29-Blade
29-Blade
29-Blade
02-Housing
09-Back cover
11-Cylinder
(~)110 - Front cover
(~)126-hexagonal shims
06-Reversible knob
07-Valve sleeve
(~)124-Stem
(~)108-Trigger
(~)127-Speed Knob
(~)104-Sealing Plugs
(~)105-Inlet
head assembly
(~)115 - Impact Shell
Pick from List
Change Component
Display
Configure Component
Suppress
Blank
Show Only
Show Picked on Tree
Merge
Fix
Unblank External Datum
Make Virtual
Make Envelope
Flexible Part
Cut
Copy
Copy with Constraints
Paste
Appearance
Attributes
Toggle Entity Transparency
Entity Info
Layer
Customize menu
Reset
Wireframe
Shade
Hidden Line
Analysis
Transparent

4. When double-clicking other components to enter in-place editing, non-in-place editing components still record the changed display mode.


<right-click> for context sensitive options.
<Shift-right-click> to display pick filter.
Show All
Assembly Node
pneumatic wrench
UF101-Rotor
29-Blade
29-Blade
29-Blade
29-Blade
29-Blade
29-Blade
02-Housing
09-Back cover
11-Cylinder
(~)110 - Front cover
(~)126-hexagonal shims
06-Reversible knob
07-Valve sleeve
(~)124-Stem
(~)108-Trigger
(~)127-Speed Knob
(~)104-Sealing Plugs
(~)105-Inlet
head assembly
(~)115 - Impact Shell

【Where is it】

Assembly Environment >> Right-click menu >> Show

3.5 Drawing Sheet Design

3.5.1 ★New Discrete Projection

ZW3D 2025 has added a new discrete projection function, which can greatly improve the view projection and view section efficiency of the engineering drawing, especially for large assembly model efficiency, and can effectively enhance the output efficiency of engineering drawings for customers.

【What can be done】：

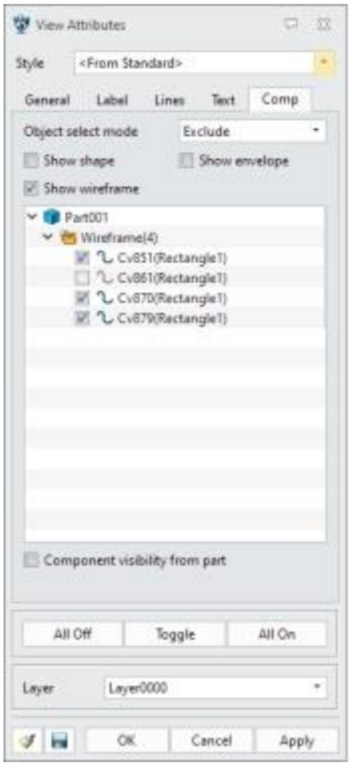
- The view projection efficiency of discrete projection

Case	Component Size	3D View Projection Time
Case1	1000+	4s
Case2	7300+	15s
Case3	14000+	36s

- The view section efficiency of the discrete projection

Case	Component Size	Full Section Time	Partial Section Time
Case1	1000+	3s	7s
Case2	7300+	9s	1s
Case3	14000+	28s	69s

- Discrete projection view supports independent control of the visibility of each 3D wire frame data in the view. The control entry is in View Attribute >> Components >> Show wireframe.


View Attributes

Style: <From Standard>

General | Label | Lines | Text | Comp

Object select mode: Exclude

☐ Show shape ☐ Show envelope

☑ Show wireframe

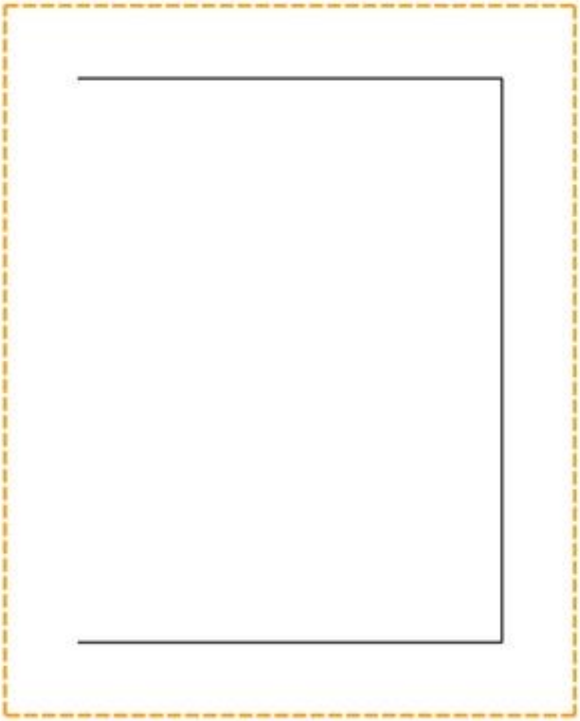
- Part001
  - Wireframe(4)
    - ☑ Cv851(Rectangle1)
    - ☐ Cv861(Rectangle1)
    - ☑ Cv870(Rectangle1)
    - ☑ Cv879(Rectangle1)

☐ Component visibility from part

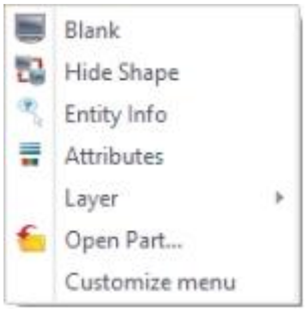
All Off | Toggle | All On

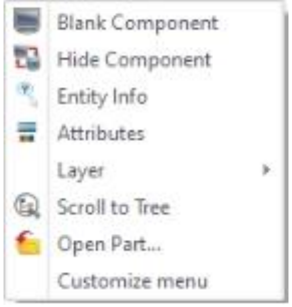
Layer: Layer0000

OK | Cancel | Apply



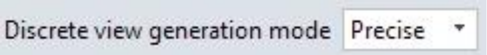
- Add Hide component and Hide shape options in the view area.


Blank
Hide Shape
Entity Info
Attributes
Layer
Open Part...
Customize menu

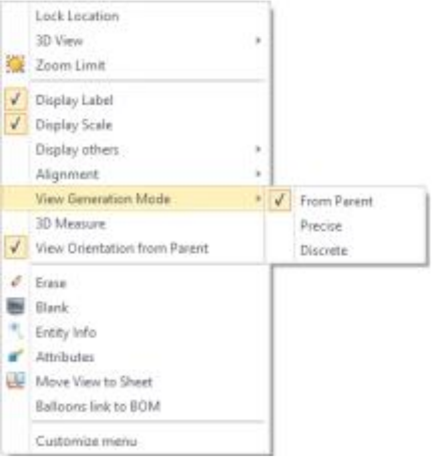

Blank Component
Hide Component
Entity Info
Attributes
Layer
Scroll to Tree
Open Part...
Customize menu

【Note】:

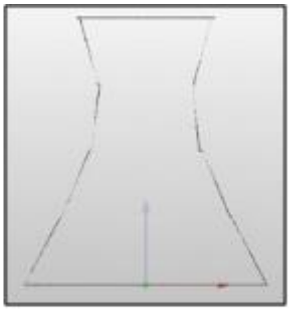
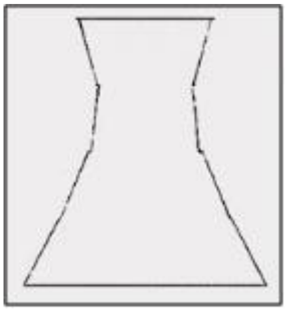
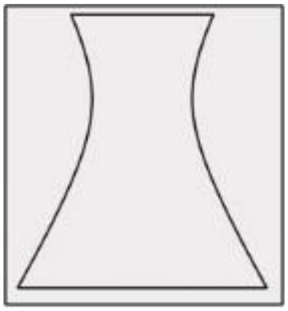
- The discrete projection view area distinguishes high and low quality. By the generation mode option in Configuration >> 2D >> Drawing sheet >> Discrete, you can control the default quality of view projection. After projecting the view, you can change the view quality in the right menu of the view. The right-click menu supports “From Parent”, “Precise”, and “Discrete”.



Discrete view generation mode: Precise


Lock Location
3D View
Zoom Limit
Display Label
Display Scale
Display others
Alignment
View Generation Mode
From Parent
Precise
Discrete
3D Measure
View Orientation from Parent
Erase
Blank
Entity Info
Attributes
Move View to Sheet
Balloons link to BOM
Customize menu

- The high (Precise) and low (Discrete) quality of the view have certain differences in display, and the low-quality view will have poor display quality in the case of complex surfaces.

3D Shade	3D Wireframe	Discrete projection in low-quality view	Discrete projection in high-quality view
			

- The edge property settings take effect only with high quality, not with low quality.
- Auto-dimension/auto-drafting are only available for high quality view.
- Select (default) “Auto load geometric data while saving drawings or exporting PDF”. The geometry data for all views must be loaded before you save the file. If you do not select this option, you need to check whether the geometry data of the view is loaded when saving. The loaded view will save the geometry data, while the incomplete view will only save the display data.
- Select the (default) " Auto load geometric data while saving drawings or exporting PDF" option. After loading the geometry data of all views before exporting PDF, ensure that the

output results are of high quality. If it is not checked, the output PDF determines whether the geometric data of the view is loaded. The loaded view outputs high-quality PDF, and the unloaded view displays data output (low quality). In this way, there is no need to wait for output.

- You need to wait for the geometric data to be loaded before printing/outputting DWG.
- Blank function is adjusted. High quality: support a single Blank sideline; Low quality: support the entire assembly/shape of Blank.

【Example】:

1. Use large assembly files and enter the three-view diagram of the drawing sheet projection to improve the projection efficiency.
2. For large assembly views, use full section operation to improve the view section efficiency.

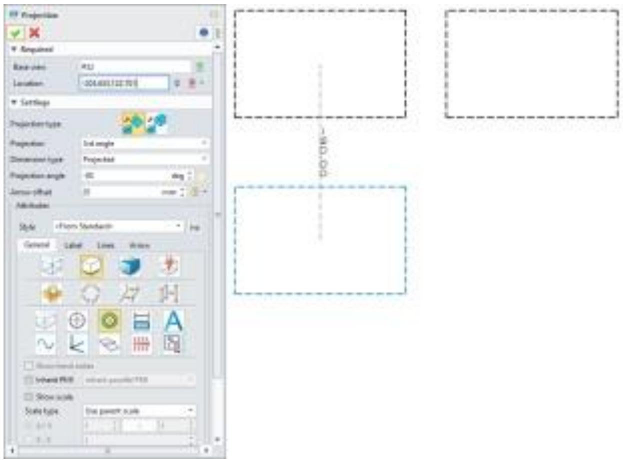
【Where is it】:

Configuration >> 2D >> Drawing sheet >> Discrete projection/ Discrete view generation mode

Configuration >> 2D >> Drawing sheet >> Auto load geometric data while saving drawings or exporting PDF

3.5.2 New Empty View

ZW3D 2025 has added Empty View. When defining an engineering drawing template, you can add an empty view to define the position of the view layout and view attributes in advance. In the future, the engineering drawing template can inherit this view information to help users define the template view more accurately.

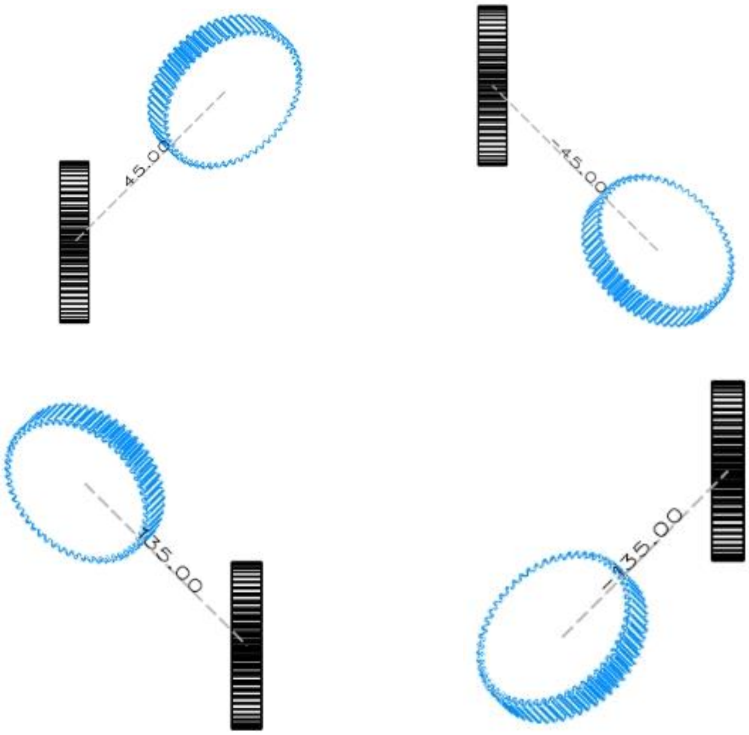


【Where is it】:

Drawing Sheet Environment >> Layout >> View >> Empty

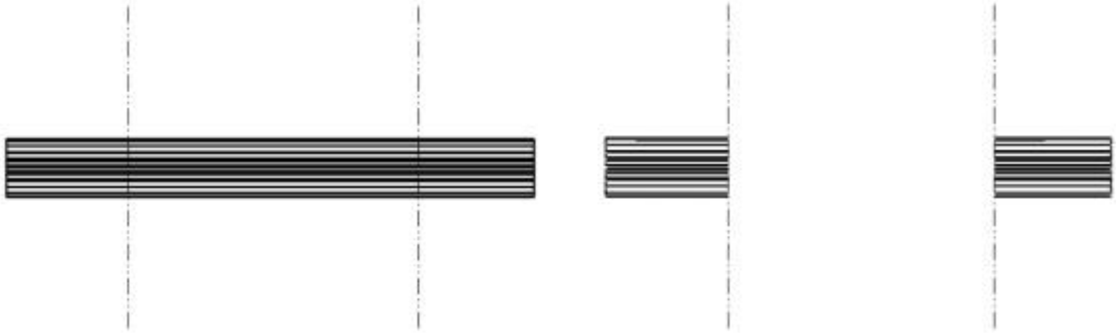
3.5.3 Isometric View Optimization

ZW3D 2025 has optimized the creation of axonometric views of engineering drawings. After the main view is projected, the system can automatically switch to the corresponding axonometric view when projecting views at 45°in four directions.



3.5.4 Un-break View

ZW3D 2025 has added the Un-break View function to switch the fracture state of the created fracture view, which can un-break the view state to observe the whole picture of the part or restore the broken view state.



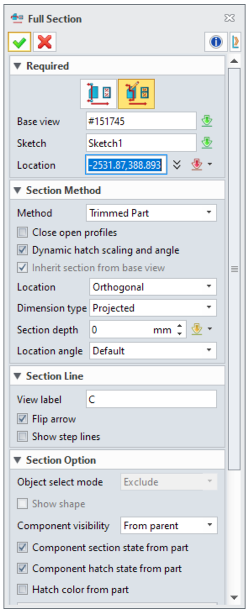
【Where】：

Drawing Sheet Environment >> View >> Break Line View >> Context Menu >> Un-Break View/Restore Break View

3.5.5 Section View Optimization

3.5.5.1 Support Section View Created by Sketch

It is a common method in engineering drawing design to define the cut line of a section view through a sketch because the sketch can accurately locate the section area by means of constraints and dimensions. ZW3D 2025 has added the function of defining section lines with sketches, through which users can determine the position of section lines based on drawing sketches.



Full Section

Required

Base view: #151745
Sketch: Sketch1
Location: -2531.87,388.893

Section Method

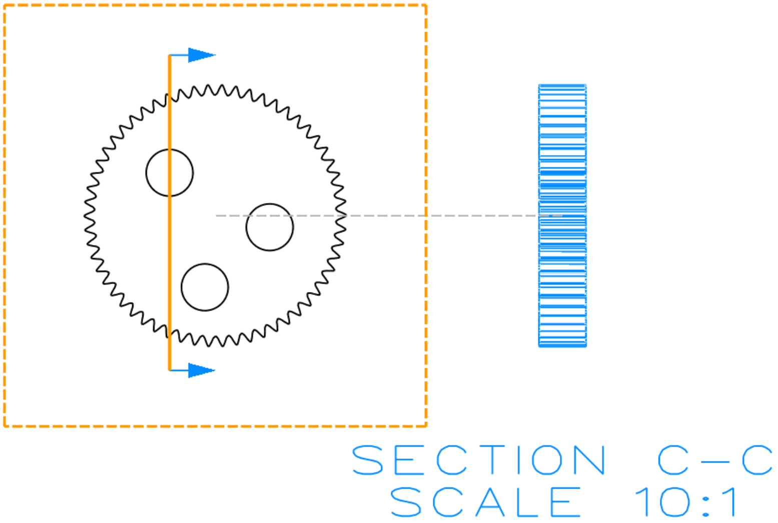
Method: Trimmed Part
☐ Close open profiles
☑ Dynamic hatch scaling and angle
☑ Inherit section from base view
Location: Orthogonal
Dimension type: Projected
Section depth: 0 mm
Location angle: Default

Section Line

View label: C
☑ Flip arrow
☐ Show step lines

Section Option

Object select mode: Exclude
☐ Show shape
Component visibility: From parent
☑ Component section state from part
☑ Component hatch state from part
☐ Hatch color from part

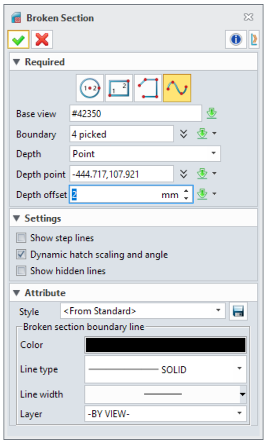


【Where is it】:

Drawing Sheet Environment >> Layout >> View >> Full Section

3.5.5.2 Broken Section Supports Spline

To support the creation of more and more complex local profile areas of engineering drawings, ZW3D 2025 has added a local profile creation type: spline curve. With the spline creation mode, users can create spline curve boundaries to generate broken section views.



Broken Section

Required

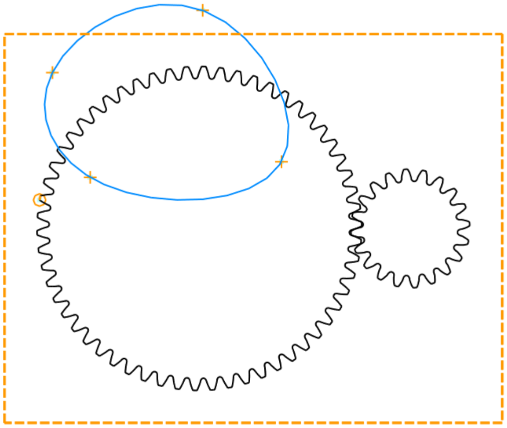
Base view: #42350
Boundary: 4 picked
Depth: Point
Depth point: -444.717,107.921
Depth offset: 2 mm

Settings

☐ Show step lines
☑ Dynamic hatch scaling and angle
☐ Show hidden lines

Attribute

Style: <From Standard>
Broken section boundary line
Color: ■
Line type: SOLID
Layer: -BY VIEW-

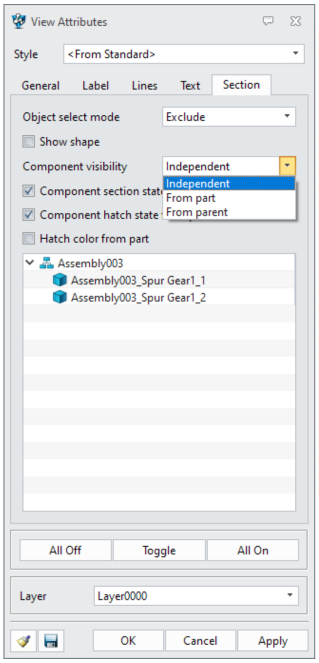


【Where is it】:

Drawing Sheet Environment >> Layout >> View >> Broken Section

3.5.5.3 Section View Supports Independently Controlling Object Visibility

ZW3D 2025 section view supports three kinds of object visibility control methods, namely component visibility from part, from parent, and independent. Users can flexibly set it according to actual production needs.



Style <From Standard>

Object select mode Exclude

☐ Show shape

Component visibility Independent

Independent
From part
From parent

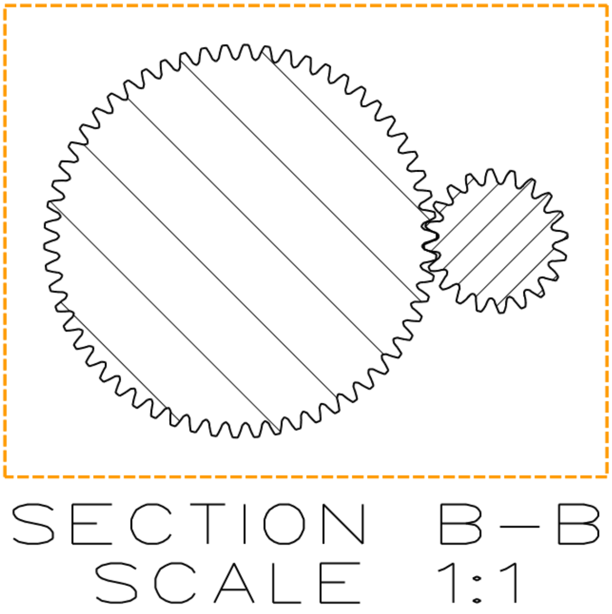
☑ Component section stat

☑ Component hatch state

☐ Hatch color from part

Assembly003
Assembly003_Spur Gear1_1
Assembly003_Spur Gear1_2

Layer Layer0000



【Where is it】:

Drawing Sheet Environment >> View Area >> Section View >> View Attributes >> Section

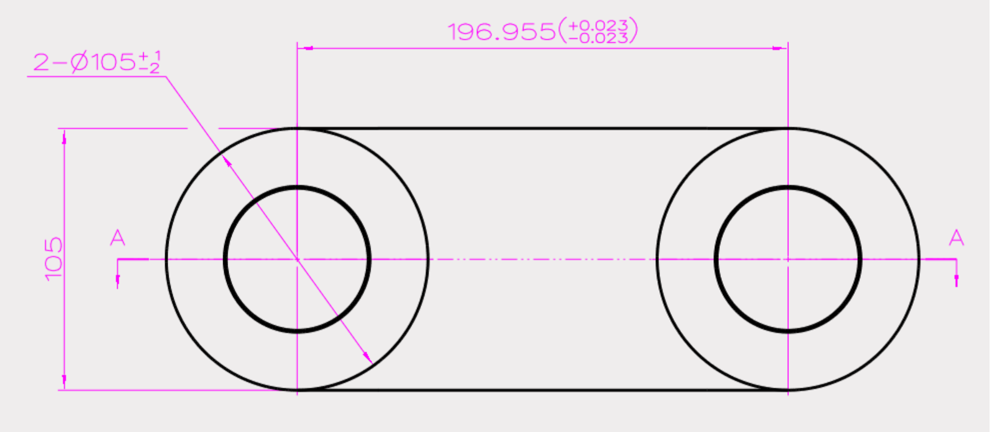
3.5.6 ★Dimension Attribute Enhancement

When adding dimensions for drawing sheets, ZW3D 2025 supports the setting of "break dimension lines", "8 text boxes", "GB/T 1800-2020 Tolerance Bands" for annotation attributes so that the dimension results are clearer and meet the standard requirements.

3.5.6.1 Break Dimension Lines

【What can be done】:

- ✓ Support setting the dimension attribute to "break dimension lines" to eliminate interference and avoid unclear drawings.

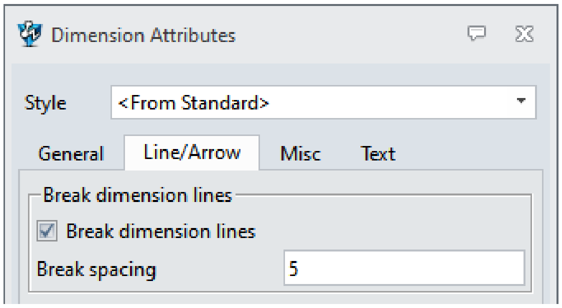


【Note】:

- Only dimensions created by commands such as Linear, Baseline, Continuous, Coordinate, Linear Offset, Symmetry, Chamfer Dimension, Angle, Angular Ordinate, Radial/Diametric, Arc Length, and Hole Callout are supported to set the breaking attribute.
- Among dimension lines, dimension boundary lines will be broken preferentially.
- If there is an intersection between the dimension boundary line and the dimension line, it has a higher priority for breaking.
- Only dimension boundary lines or dimension lines will be broken if there is an intersection between the lead line of annotations or symbols and dimension boundary lines or dimension lines.
- When there is an intersection between angle dimensions, the line with a larger radius will be broken preferentially. If the dimension line radiuses are equal, the previously created dimensions will be broken.
- Angle dimensions will be broken preferentially if there is an intersection between angle dimensions and linear dimensions.
- When dimension lines and dimension boundary lines overlap each other, dimension boundary lines will be broken preferentially.

【Example】:

1. Users can set "Break dimension lines" for intersecting dimensions by right-click menu.



Dimension Attributes

Style <From Standard>

General | Line/Arrow | Misc | Text

Break dimension lines

☑ Break dimension lines

Break spacing 5

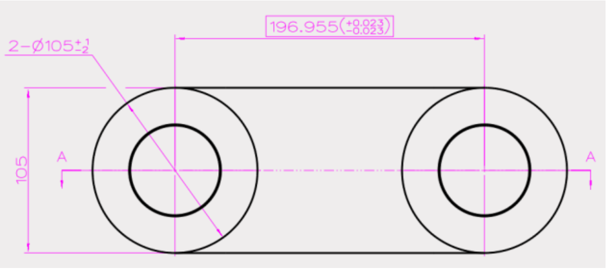
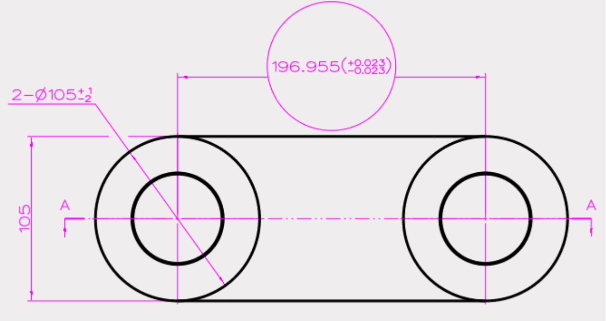
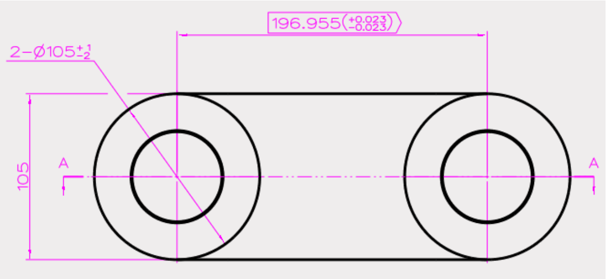
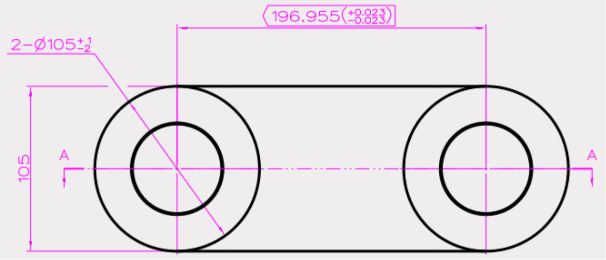
【Where】：

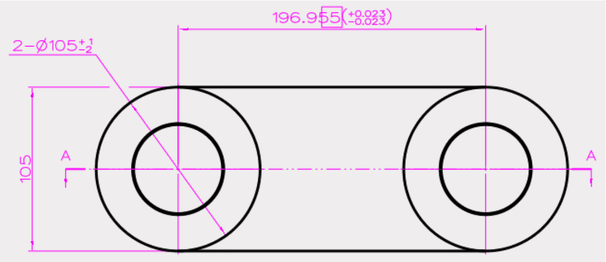
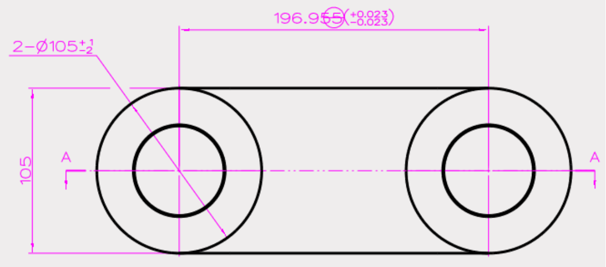
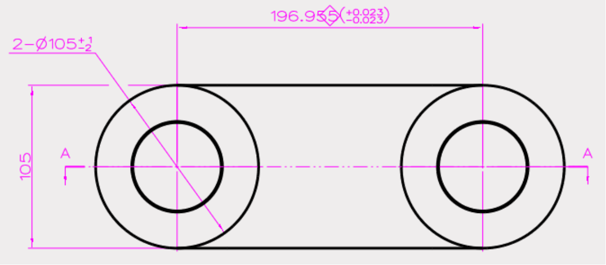
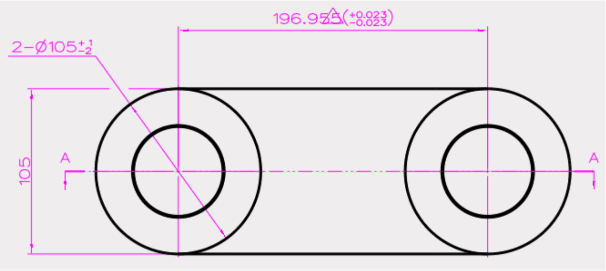
Drawing Sheet >> Dimension >> Attributes >> Line/Arrow >> Break dimension lines

3.5.6.2 Text box

【What can be done】：

- ✓ ZW3D 2025 provides eight types of text boxes: rectangle, circle, right mark, left mark, square, scored circle, diamond, and triangle.
- ✓

Text Box Types	
 196.955 2-Ø105 105 A A Rectangle	 196.955 2-Ø105 105 A A Circle
 196.955 2-Ø105 105 A A	 196.955 2-Ø105 105 A A

Right mark	Left mark
 Square	 Scored circle
 Diamond	 Triangle

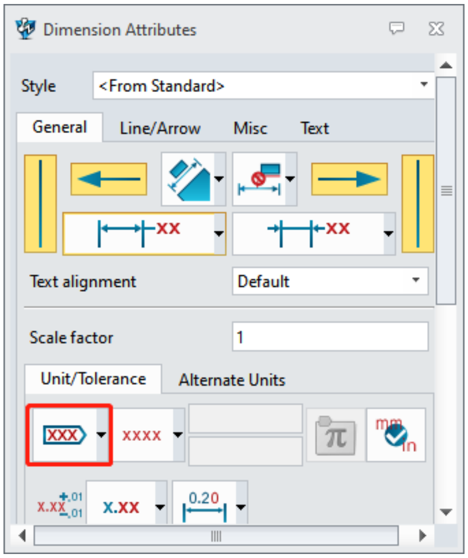
【Note】:

- Only dimensions created by commands such as Linear, Baseline, Continuous, Coordinate, Linear Offset, Symmetry, Chamfer Dimension, Angle, Angular Ordinate, Radial/Diametric, Arc Length, and Hole Callout are supported to set attributes.
- Rectangle, circle, right mark, left mark, and long circle can calculate the text box size based on the text size automatically.
- Square, scored circle, diamond, and triangle take the minimum values of text length and width

and generate a text box in the center of the text.

【Example】：

1. The box type can be set in Dimension Attributes. For example, the right mark shown in the picture below:


Dimension Attributes
Style
<From Standard>
General
Line/Arrow
Misc
Text
XX
XX
Text alignment
Default
Scale factor
1
Unit/Tolerance
Alternate Units
XXX
XXXX
x.xx
0.20

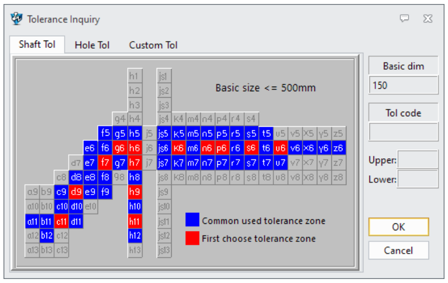
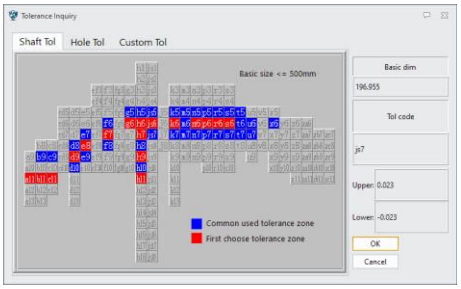
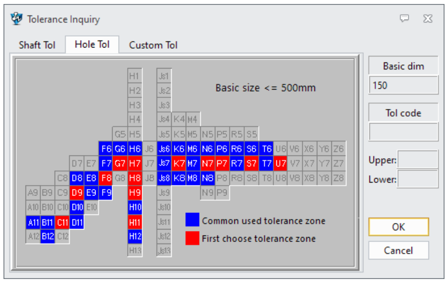
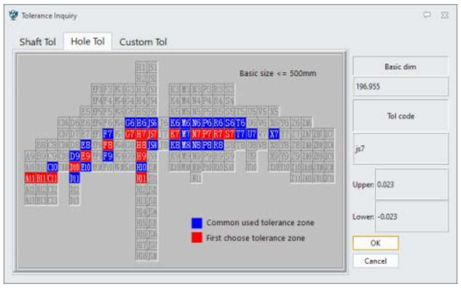
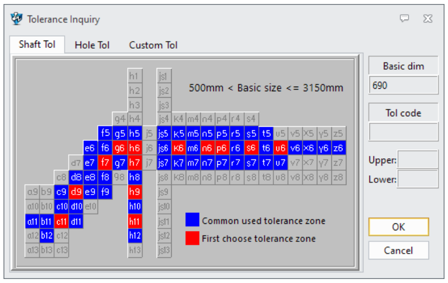
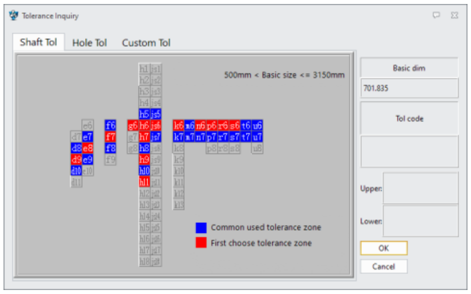
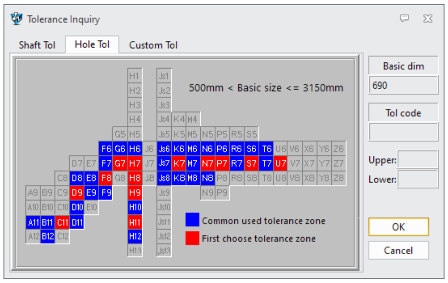
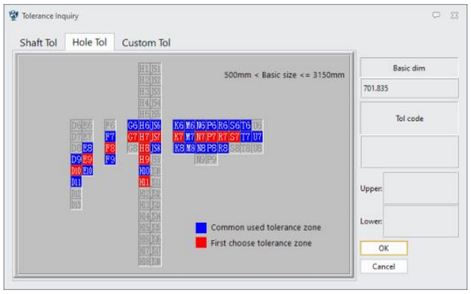
【Where】：

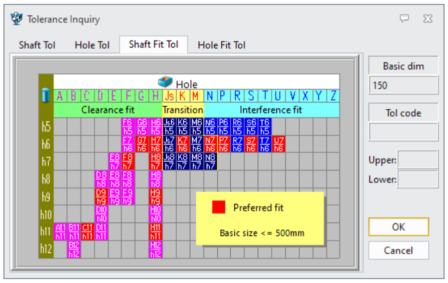
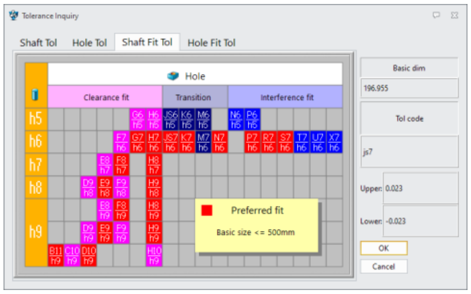
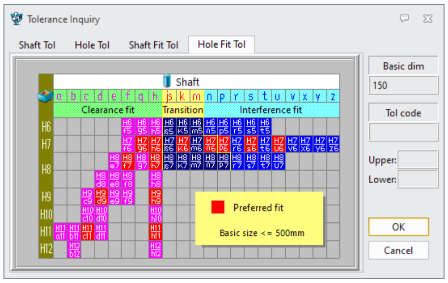
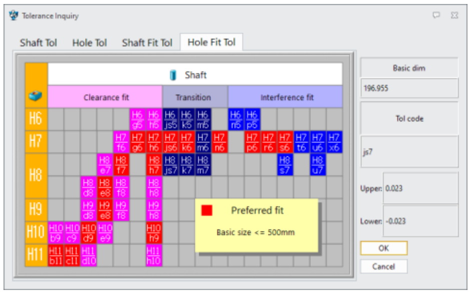
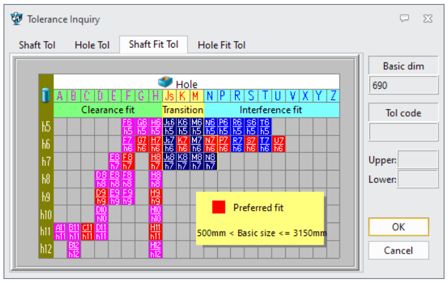
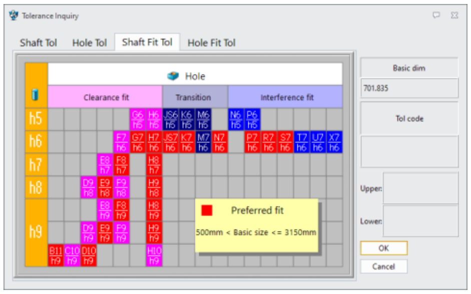
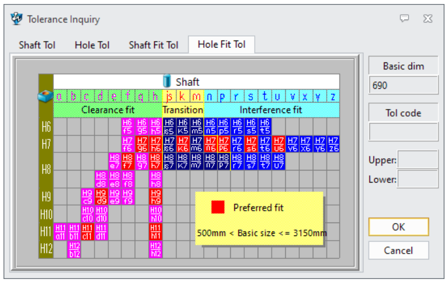
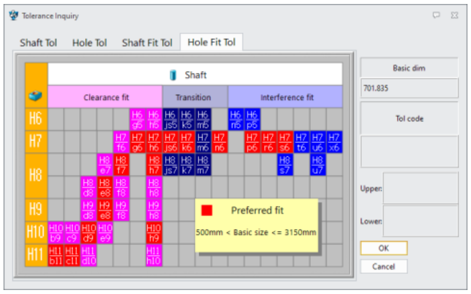
Drawing Sheet >> Dimension Attributes >> General >> Unit/Tolerance

3.5.6.3 Tolerance standard

【What can be done】：

- ✓ The tolerance standard is updated from GB/T 1800-1997 to GB/T 1800-2020, and more tolerance codes are supported, such as ef, fg, za, zb, zc, etc.

Fit type	ZW3D 2024	ZW3D 2025
Shaft Tolerance (Basic size≤500mm)	 Tolerance Inquiry Shaft Tol Basic size <= 500mm Basic dim: 150 Tol code: Upper: Lower: Common used tolerance zone First choose tolerance zone	 Tolerance Inquiry Shaft Tol Basic size <= 500mm Basic dim: 196.955 Tol code: js7 Upper: 0.023 Lower: -0.023 Common used tolerance zone First choose tolerance zone
Hole Tolerance (Basic size ≤500mm)	 Tolerance Inquiry Hole Tol Basic size <= 500mm Basic dim: 150 Tol code: Upper: Lower: Common used tolerance zone First choose tolerance zone	 Tolerance Inquiry Hole Tol Basic size <= 500mm Basic dim: 196.955 Tol code: js7 Upper: 0.023 Lower: -0.023 Common used tolerance zone First choose tolerance zone
Shaft Tolerance (500mm< Basic size ≤ 3150mm)	 Tolerance Inquiry Shaft Tol 500mm < Basic size <= 3150mm Basic dim: 690 Tol code: Upper: Lower: Common used tolerance zone First choose tolerance zone	 Tolerance Inquiry Shaft Tol 500mm < Basic size <= 3150mm Basic dim: 701.835 Tol code: Upper: Lower: Common used tolerance zone First choose tolerance zone
Hole Tolerance (500mm< Basic size ≤ 3150mm)	 Tolerance Inquiry Hole Tol 500mm < Basic size <= 3150mm Basic dim: 690 Tol code: Upper: Lower: Common used tolerance zone First choose tolerance zone	 Tolerance Inquiry Hole Tol 500mm < Basic size <= 3150mm Basic dim: 701.835 Tol code: Upper: Lower: Common used tolerance zone First choose tolerance zone

Shaft Fit Tolerance (Basic size ≤500mm)	 Shaft Fit Tol Basic dim: 150 Basic size <= 500mm	 Shaft Fit Tol Basic dim: 196.955 Tol code: js7 Upper: 0.023 Lower: -0.023 Basic size <= 500mm
Hole Fit Tolerance (Basic size ≤500mm)	 Hole Fit Tol Basic dim: 150 Basic size <= 500mm	 Hole Fit Tol Basic dim: 196.955 Tol code: js7 Upper: 0.023 Lower: -0.023 Basic size <= 500mm
Shaft Fit Tolerance (500mm < Basic size ≤ 3150mm)	 Shaft Fit Tol Basic dim: 690 500mm < Basic size <= 3150mm	 Shaft Fit Tol Basic dim: 701.835 500mm < Basic size <= 3150mm
Hole Fit Tolerance (500mm< Basic size ≤ 3150mm)	 Hole Fit Tol Basic dim: 690 500mm < Basic size <= 3150mm	 Hole Fit Tol Basic dim: 701.835 500mm < Basic size <= 3150mm

【Note】:

- The intermediate basic deviations CD, EF, FG, cd, ef, and fg are mainly used in the precision mechanism and watch manufacturing industries. If you need to include the basic deviation in other nominal dimensions of the tolerance zone code, you can calculate it according to GB/T 1800.1-2020.

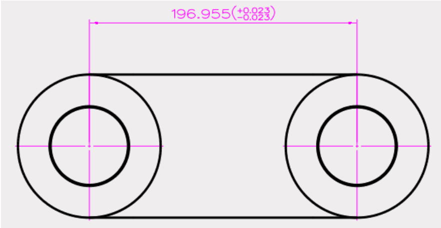
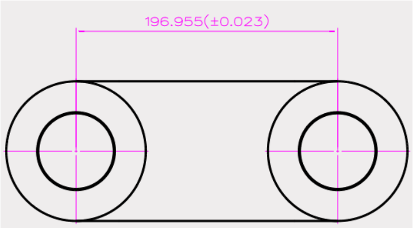
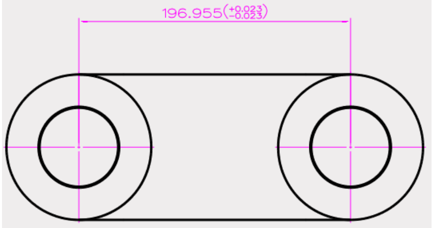
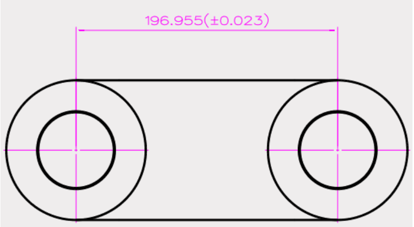
【Where】:

Drawing Sheet >> Dimension Attributes >> General >> Unit/Tolerance

3.5.6.4 Tolerance font size

【What can be done】:

- ✓ Set different font sizes and alignments for equal tolerance and unequal tolerance.

	Equal Tolerance	Unequal Tolerance
Half Align	 196.955(+0.023 -0.023)	 196.955(±0.023)
Bottom Align	 196.955(+0.023 -0.023)	 196.955(±0.023)

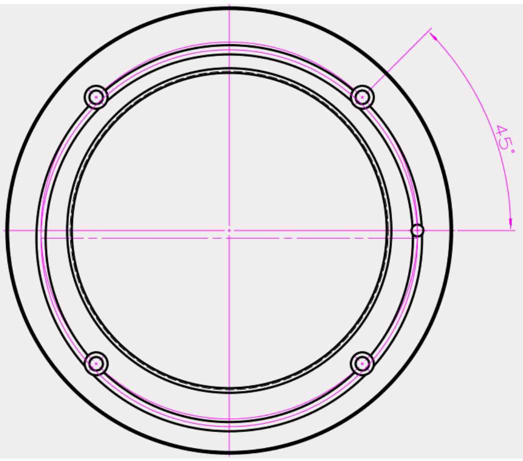
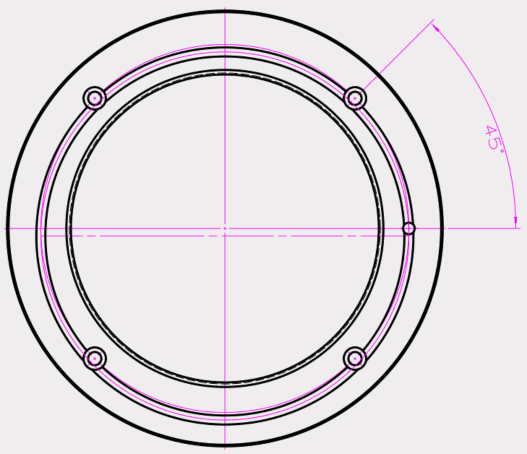
【Where】:

Drawing Sheet >> Dimension Attributes >> Text>>Text Shape

3.5.6.5 Angle Dimension Attributes

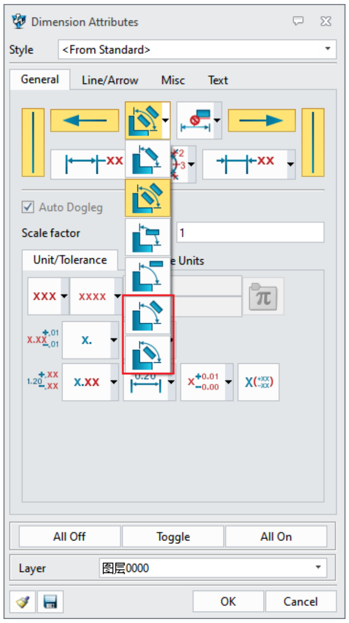
【What can be done】:

- ✓ Set text position for angle dimension attribute, allowing the text to be on the outside or inside of the arc.

Text is outside the arc	Text is inside the arc
	

【Example】:

1. Open the angle Dimension Attributes in the right-click menu and set the text outside the arc or inside the arc.



Dimension Attributes
General
Line/Arrow
Misc
Text
Unit/Tolerance
Units
All Off
Toggle
All On
OK
Cancel

Style <From Standard>

☑ Auto Dogleg

Scale factor 1

Layer 图层0000

【Where】:

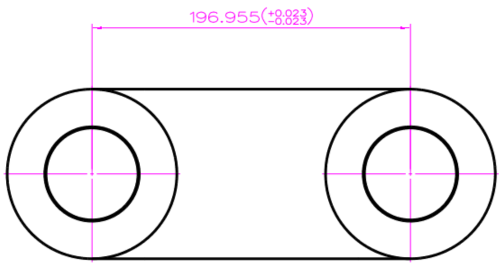
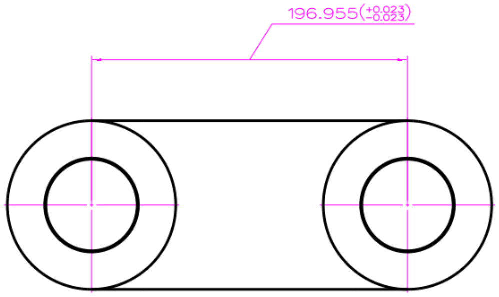
Drawing Sheet >> Dimension Attributes >> General

3.5.7 Fast Switching Style

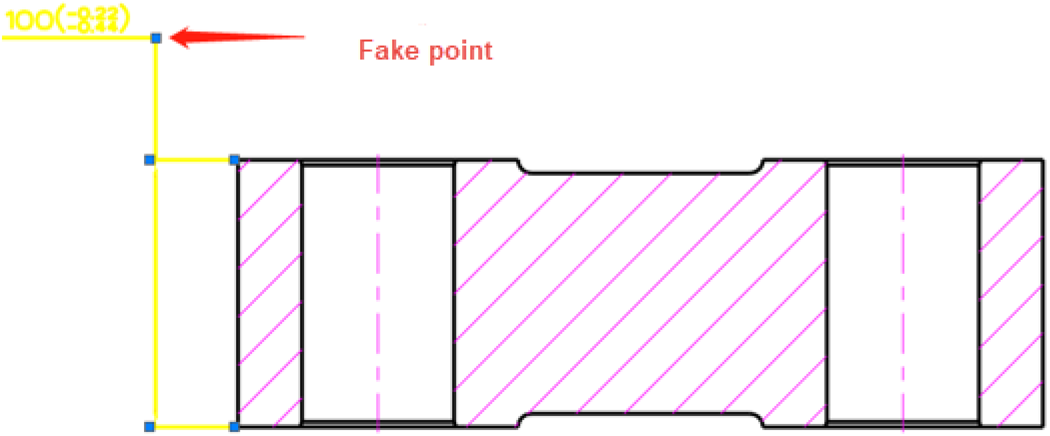
In the process of adding dimensions for drawing sheets, users can adjust the dimension style to make the dimension result clearer and more concise and take up less space.

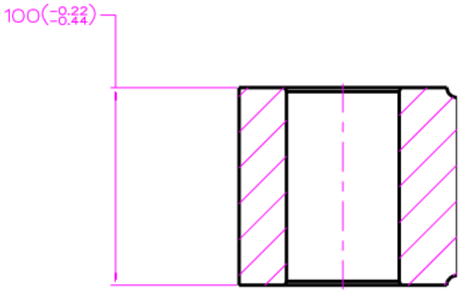
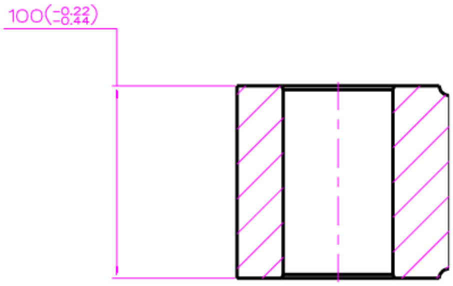
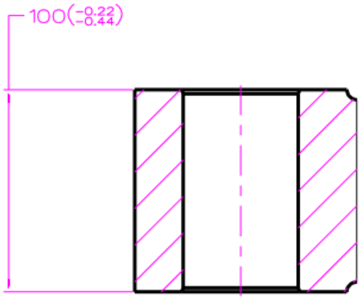
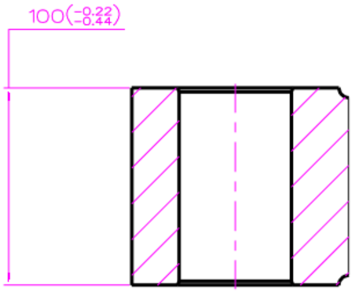
【What can be done】:

- ✓ Users can hold down Shift and left-click to drag the dimension to quickly change the style from "None" to "Above bent leader". Besides, users can hold down Shift and left-click to drag the dimension to quickly change the style from " Above bent leader" to " None ".

Dimension style “None”	Dimension style “Above bent leader”
	

- ✓ Users can click the "fake point" on the dimension with the "Horizontal text and bent leader" style or the "Horizontal text above line and bent leader" style to switch the bending direction.



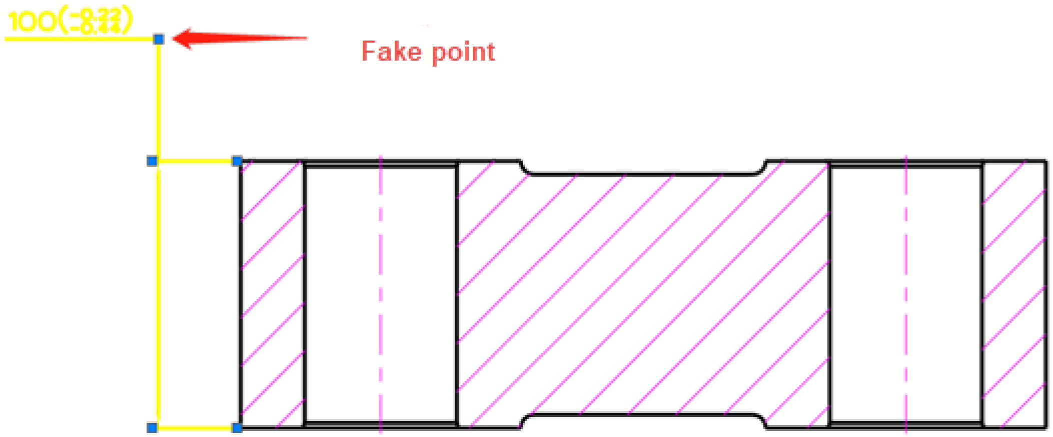
	Horizontal text and bent leader	**Horizontal text above line and bent leader**
Before switch direction		
After switch direction		

【Note】:

- Only dimensions created by the Linear, Baseline, Continuous, Linear Offset, Angle, and Arc Length commands can be quickly changed to “Above bent leader”.
- Only dimensions created by the Linear, Linear Offset, and Symmetrical Annotation commands can quickly change the bending direction.
- Dimensions created by commands such as Baseline, Continuous, Coordinate, Linear Chamfer, Angle, Angular Ordinate, Radial/Diametric, Arc Length, and Hole Callout do not support changing the bending direction.

【Example】:

1. For dimensions created by the Linear Dimension command, click the "fake point" on the “Horizontal text and bent leader” style to change the bending directions.



【Where is it】：

Drawing Sheet >> Drawing Area

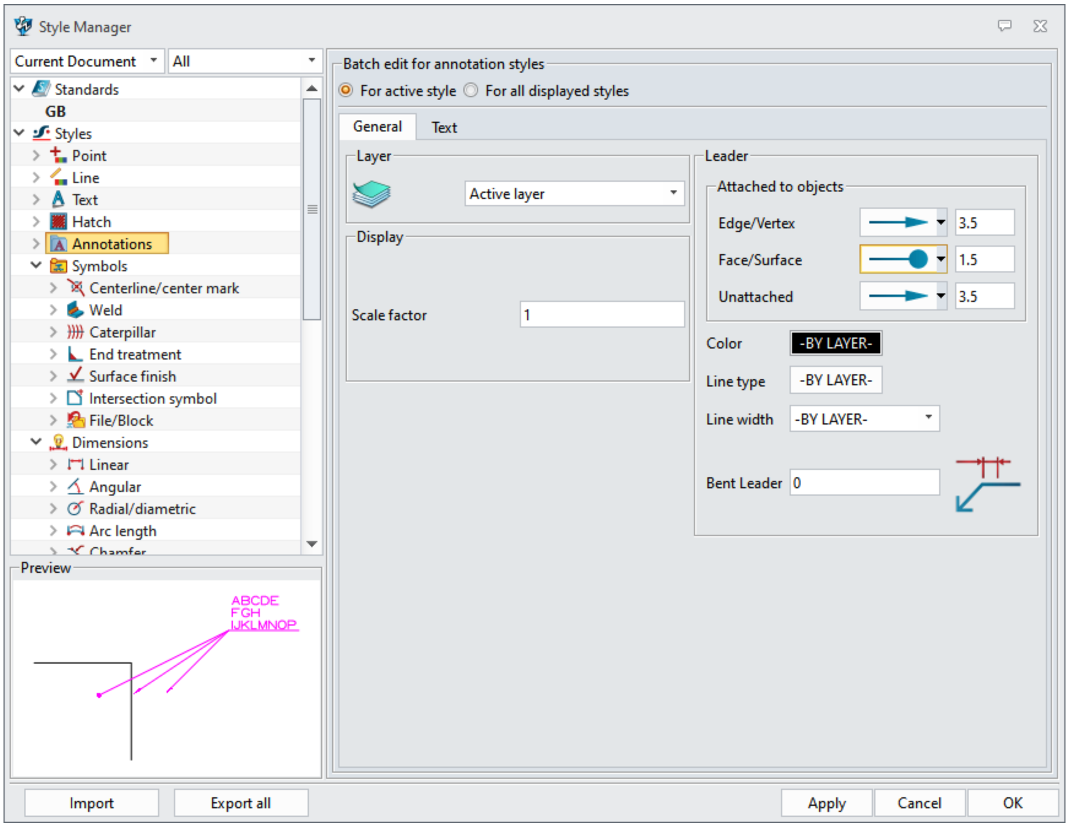
3.5.8 ★Easy-to-use Multiple Arrow (Copy and Attached Object Correctness)

Users can set different arrow types for the symbols based on different objects, such as Labels, Balloons, Feature Control symbols, Weld symbols, Surface Finish symbols.

In addition, users can also quickly add or delete arrows of dimensions and annotations, such as Radial, Diametric, and Chamfer dimensions, Hole Callout, Leader and Balloon annotations, Datum Feature, Weld symbols, and Surface Finish symbols, realizing the addition or deletion of multi-branch arrows without opening another interface.

【What can be done】：

- ✓ Set different types and sizes for arrows attached to edge/vertex, face/surface, or no attached object in Style Manager.


Style Manager
Current Document
All
Standards
GB
Styles
Point
Line
Text
Hatch
Annotations
Symbols
Centerline/center mark
Weld
Caterpillar
End treatment
Surface finish
Intersection symbol
File/Block
Dimensions
Linear
Angular
Radial/diametric
Arc length
Chamfer
Preview
ABCDE
FGH
IJKLMNOP
General
Text
Layer
Display
Leader
Attached to objects
Import
Export all
Apply
Cancel
OK

Batch edit for annotation styles: ◉ For active style ○ For all displayed styles

Layer: Active layer

Scale factor: 1

Edge/Vertex: 3.5

Face/Surface: 1.5

Unattached: 3.5

Color: -BY LAYER-

Line type: -BY LAYER-

Line width: -BY LAYER-

Bent Leader: 0

- ✓ Create different arrows according to the attached objects when creating the Labels, Balloons, Feature Control symbols, Weld symbols, and Surface finish symbols.
- ✓ Rapidly create arrow branches for Radial, Diametric, Linear Chamfer, Hole Callout, Label, Balloon, Datum Feature, Weld symbols, and Surface Finish symbols by holding down "Ctrl" and dragging the origin arrow.
- ✓ Delete arrow branches of Radial, Diametric, Linear Chamfer, Hole Callout, Label, Balloon, Datum Feature, Weld symbols, and Surface Finish symbols by selecting it and then pressing down "Delete" key.
- ✓ Create a branch for Label, Balloon, Datum Feature, Weld symbols and Surface Finish symbols without leaders by holding down "Shift" and dragging the objects.

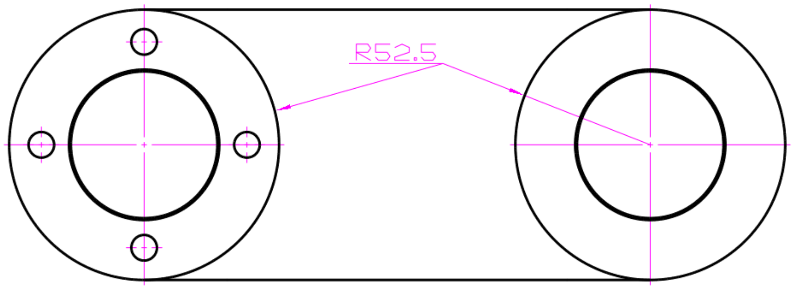
【Note】:

- For Balloon, except for the "User Text" type, other types do not support setting multiple leaders pointing to different components.

- The arrow type is automatic by default in the Attribute settings including Label, Balloon, Feature Control, and Surface Finish symbols.
- For dimension efficiency, the preview is inconsistent with the actual result because arrow type judgment happens after dragging.
- For the annotations with multiple arrow branches, the default arrow attribute is automatic. Each arrow needs to determine the attached object to generate an arrow.
- The quick-created branch insertion point is the turning point closest to the arrow, and the branch point is where the left mouse button is released.
- In the Radial, Diametric, Linear Chamfer dimension command, a branch can be created successfully if and only if the dimension of the pointing object is the same as that of the parent object.
- If the type of Radial dimension is bending type (Horizontal text above line and bent leader or Horizontal text and bent leader), the bending point is the insertion point of the branch. If the dimension type is non-bending, the branch insertion point is the midpoint of the text.
- If the type of Diametric dimension is bending type (Horizontal text above line and bent leader or Horizontal text and bent leader), the bending point is the insertion point of the branch. If the dimension type is non-bending, the branch insertion point is the midpoint of the text.
- In Linear Chamfer dimension, only the leader chamfer can lead out the branch.
- The Hole Callout can be created only when it points to an object with the same dimensions as the parent object (including the depth). When pointing to more than one object, quantity calculation is eliminated.
- The Balloon annotation and Datum Target can be created successfully only when the pointing object is the same as the parent object.

【Example】:

1. For the created radial dimension, hold down Ctrl and the left mouse button to drag the arrow to quickly create an arrow branch pointing to another object of the same size.



【**Where is it**】:

Drawing Sheet >> Drawing Area

3.5.9 Smart Dimension Ability Enhancement

In the drawing sheet design process, users can use smart dimension to annotate more scenes, such as all threaded holes mark, center mark circle, angle dimension.

【**What can be done**】**:**

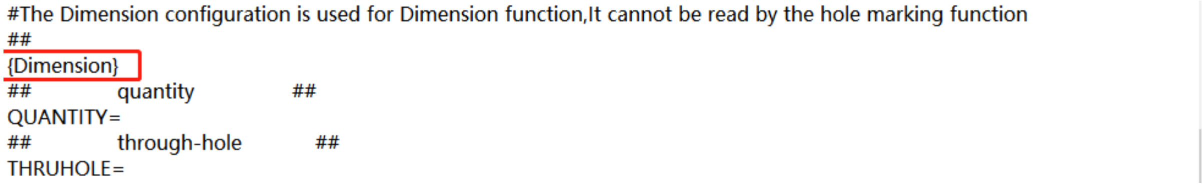
- ✓ Support marking tapered thread holes, simple thread holes, metric external threads, and inch external threads, and can customize annotation results through Hole Callout configuration files.
- ✓ Support marking the angle by two center marks in a center mark circle.
- ✓ Support marking the diameter of the center mark circle.
- ✓ Support marking the distance between the center of the center mark circle and the benchmark.
- ✓ Support marking the angles of each area of the two intersecting lines based on the mouse position.

【**Note**】**:**

- In the selection filter of smart dimension of ZW3D 2025, the midpoint filter option is off by default.

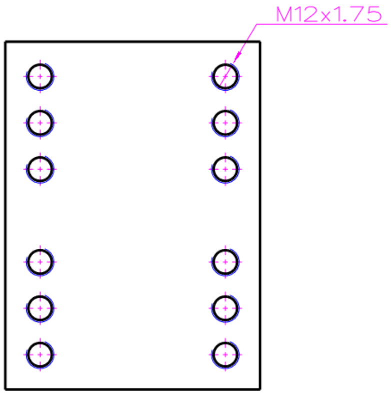
【**Example**】**:**

1. Open the file named "HoleCallFormat.txt" in the SUPP resource folder in the software installation directory. Users can customize the dimension result by modifying the dimension format in the Dimension configuration.

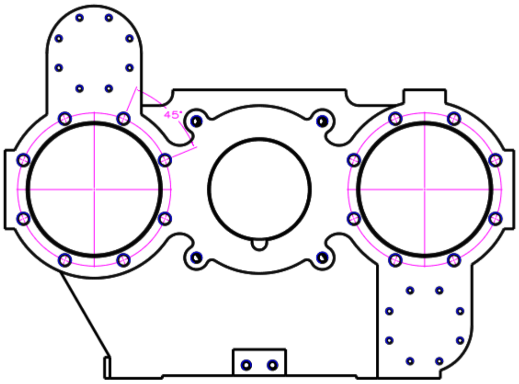


```
#The Dimension configuration is used for Dimension function,It cannot be read by the hole marking function
##
{Dimension}
##            quantity              ##
QUANTITY=
##            through-hole           ##
THRUHOLE=
```

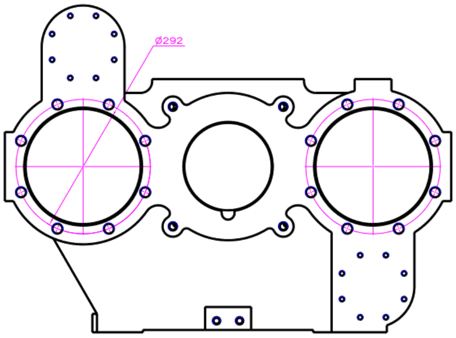
2. In the Drawing Sheet environment, use the “Dimension” command to select thread lines in the view to create a Hole Callout.



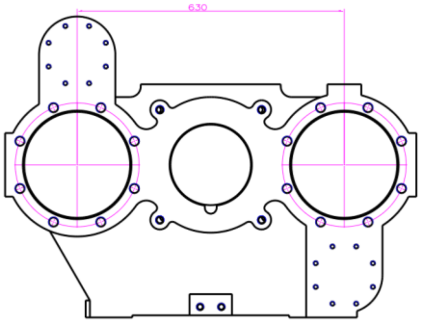
3. In the Drawing Sheet environment, use the “Dimension” command to select the center mark object in the center mark circle to mark the angle between the two center marks.



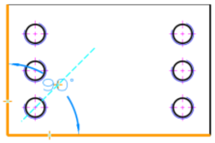
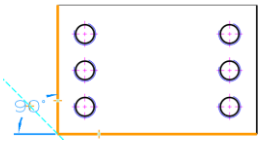
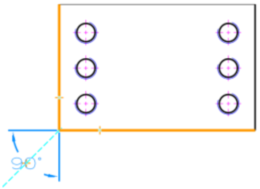
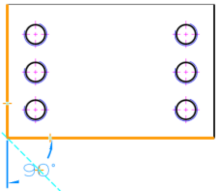
4. In the Drawing Sheet environment, use the “Dimension” command to select the center marking circle object to mark the size of the center marking circle.



5. In the Drawing Sheet environment, use the “Dimension” command to select the center of the center marking circle to mark the dimension of the center mark circle and the benchmark.



6. In the Drawing Sheet environment, use the “Dimension” command to select two lines and mark the angles of each area of the two lines based on the mouse position.

First quadrant	Second quadrant	Third quadrant	Forth quadrant
			

【Where is it】：

Drawing Sheet >> Dimension >> Dimension

3.5.10 Dimension Align

After the Linear dimension is complete, the Linear size can be set to align with the benchmark of the model according to the customer's machining intent.

【What can be done】:

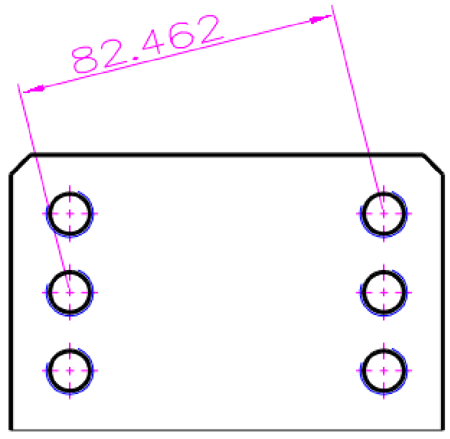
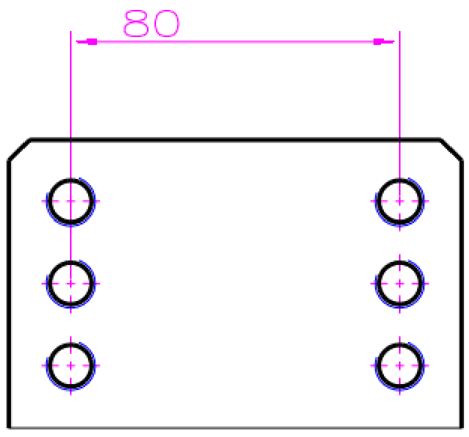
- ✓ Set horizontal alignment, vertical alignment, and line alignment for the created Linear dimensions.

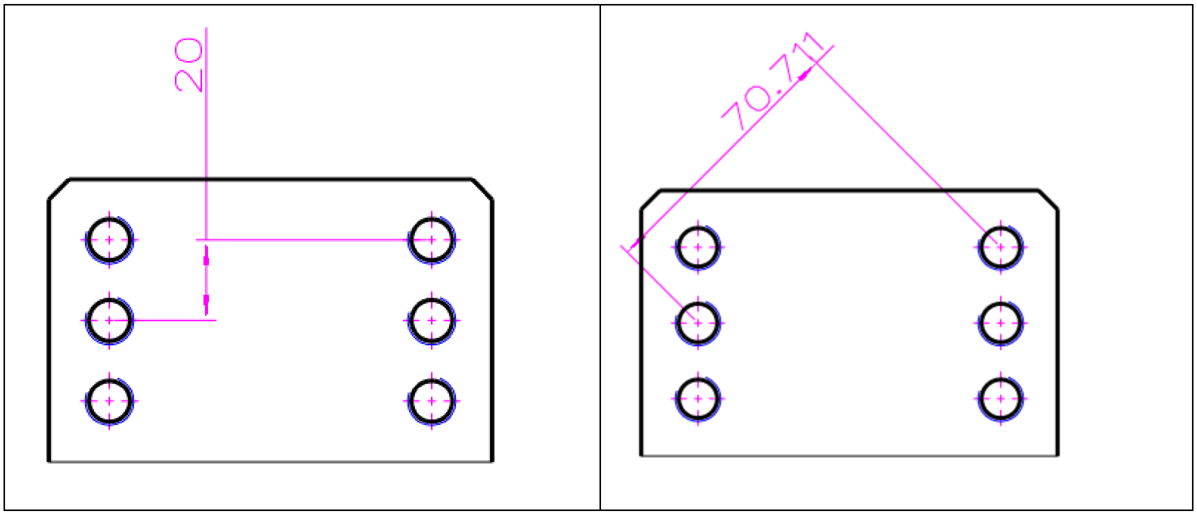
【Note】:

- The center of the annotated text after alignment overlaps with the center of the annotated text before alignment.
- Users can only select a straight line for alignment when creating line alignment.
- Cross-view selection of lines is not supported when creating the line alignment.

【Example】:

1. In the Drawing Sheet environment, right-click the Linear dimension to set the horizontal alignment, vertical alignment, and line alignment.

Before Alignment	Horizontal Alignment
	
Vertical Alignment	Line Alignment



【Where is it】:

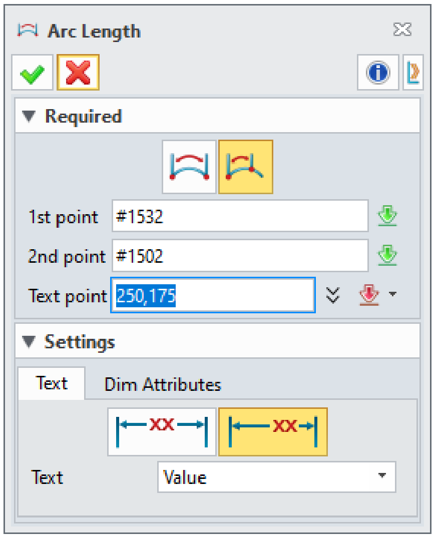
Drawing Sheet >> Drawing Area >> Linear Dimension

3.5.11 Arc Length（ZW3D 2025 SP）

ZW3D 2025 SP enriches the way of arc length labeling by selecting two points on the arc length to label the arc length, it improves the flexibility of arc length labeling.

【What can be done】:

✓ Mark the arc length by selecting two points on the arc.

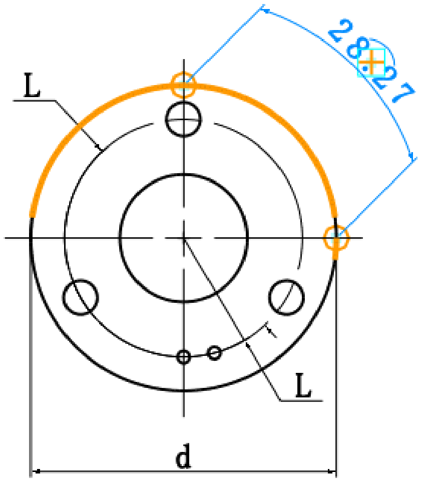

Arc Length
Required
Settings
Text
Dim Attributes

1st point #1532

2nd point #1502

Text point 250,175

Text Value



【Where is it】:

Drafting >> Dimension >> Arc Length

3.5.12 Hole Callout

In the Hole Callout scenario, users can customize the annotation style and symbol style of through-holes. For example, the word "through" is used to indicate that a hole is a through-hole, and the word "depth" replaces the depth symbol, which realizes the personalized configuration of the annotation results.

【What can be done】：

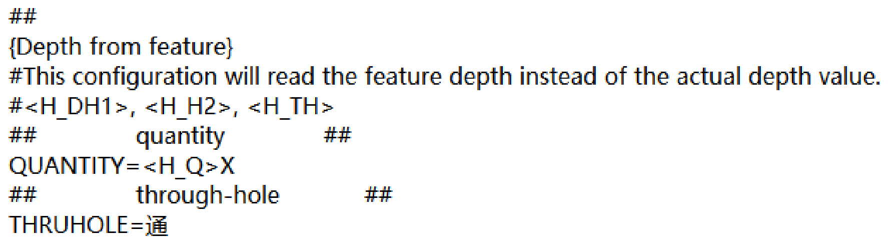
- ✓ Users can customize the expression form of the through-hole feature in multi-languages, such as Simplified Chinese, Traditional Chinese, English, Czech, German, Spanish, Italian, Polish, Japanese, Korean, Russian, Brazilian Portuguese, Turkish, etc.
- ✓ Users can customize the hole depth, diameter, and connection symbol in multi-languages, such as Simplified Chinese, Traditional Chinese, English, Czech, German, Spanish, Italian, Polish, Japanese, Korean, Russian, Brazilian Portuguese, Turkish, etc.
- ✓ Support dimensioning hole feature chamfer modeling scenarios.

【Note】：

- Only support the scenario in which a chamfer has been added to a hole feature inside. Do not support the modeling scenario in which a hole feature and a chamfer feature are added.

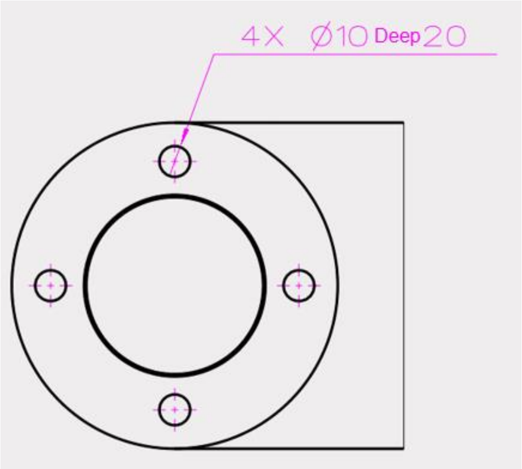
【Example】：

1. Open the Hole Callout resource files named “HoleCallFormat.txt” in the SUPP resource folder in the software installation directory, and customize the dimension result by modifying the dimension format in the “Default” and “Depth from feature” configurations.



```
##
{Depth from feature}
#This configuration will read the feature depth instead of the actual depth value.
#<H_DH1>, <H_H2>, <H_TH>
##          quantity          ##
QUANTITY=<H_Q>X
##          through-hole          ##
THRUHOLE=通
```

2. After modifying the configuration file, users can use the Hole Callout command with the “By callout format text” option in the Drawing Sheet environment to annotate customized results.



【Where is it】:

Configuration file: Installation Directory >> SUPP Folder >> HoleCallFormat.txt

Hole Callout: Drawing Sheet >> Dimension >> Hole Callout >> Settings >> By callout format text

3.5.13 Group Dimension

In the continuous dimension scenario, users can separate group dimension objects as a new group or change the group dimension type, for example, converting continuous dimensions to baseline dimensions or converting baseline dimensions to continuous dimensions.

【What can be done】:

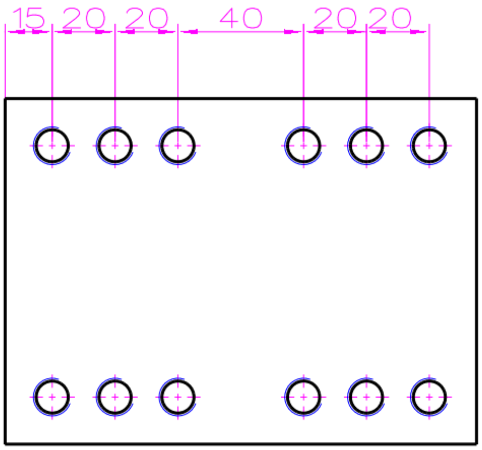
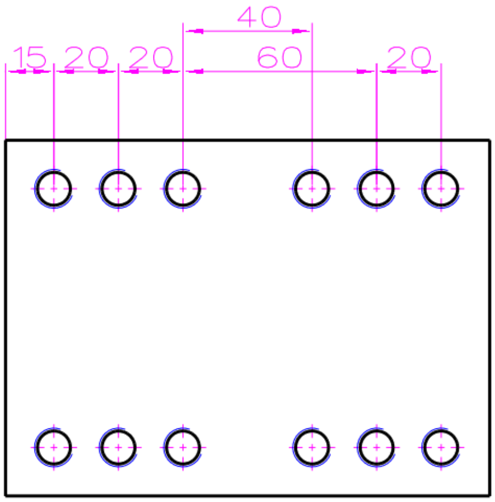
- ✓ Group dimensions for continuous dimensions allow a single dimension object to be separated from a group. After the dimension is separated from a group, the start dimension base of the next group dimension object is changed to the start dimension base of the removed dimension object.
- ✓ Continuous dimensions are allowed to separate the entire set of dimension objects selected by the Shift key from the group.
- ✓ Group dimensions of continuous dimensions support converting all dimension objects to baseline dimensions, and the converted baseline dimensions are also group dimensions.
- ✓ Group dimensions of baseline dimensions allow a single dimension object to be separated from

a group. When a dimension is separated from a group, all subsequent group dimension objects of the dimension object are moved forward by one text position, and the removed dimension is moved to the last text position.

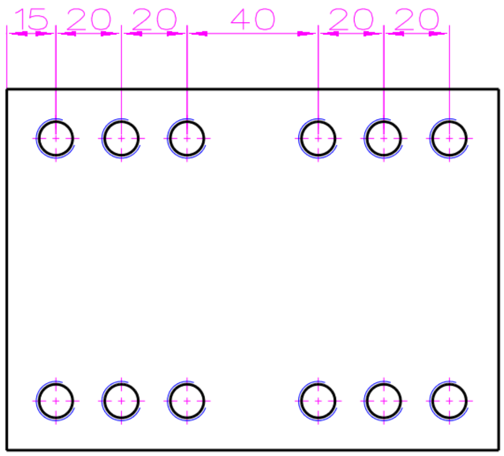
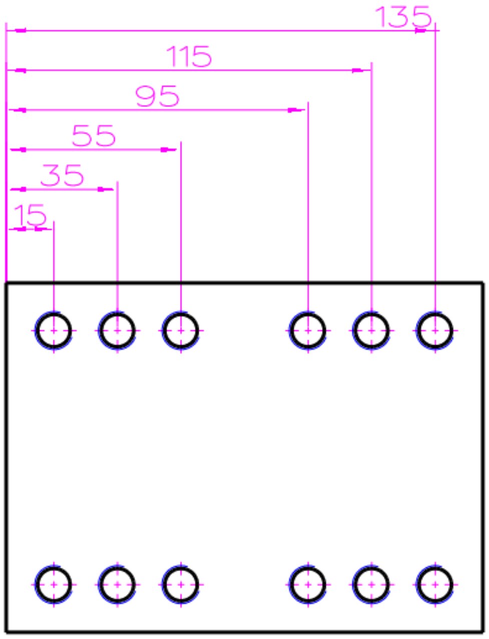
- ✓ Baseline dimensions allow users to separate the entire set of dimension objects selected by the Shift key from the group.
- ✓ Group dimensions of baseline dimensions can be converted to continuous dimensions, and the converted continuous dimensions are also group dimensions.
- ✓ Group dimensions of coordinates support separating the individual dimension objects from a group.
- ✓ Group dimensions of coordinates support separating the entire group of dimension objects selected by the Shift key from the group.

【Example】：

1. For the objects that have been created continuous dimensions, users can right-click the object to make it independent from the group.

Before dimension independence	After dimension independence
	

2. For the objects that have been created continuous dimension, right-click the object to convert the entire group to baseline dimensions.

Before converting to baseline dimension	After converting to baseline dimension
	

【Where is it】：

Drawing Sheet >> Drawing Area >> Group Dimension Object >> Right Click Menu

3.5.14 Annotation Improvement

When making annotations with commands such as Hole Table and Annotation, users can smoothly adjust the position of the text, and the annotation text does not change abruptly, which saves drawing space.

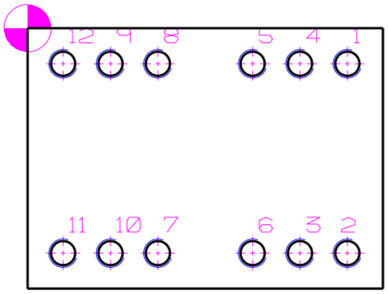
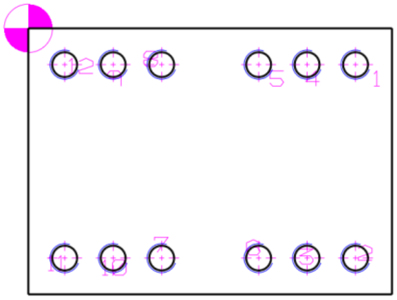
【What can be done】：

- ✓ The text control is activated automatically after the mouse determines the position point and text point of the annotation.
- ✓ When dragging annotations with leaders, they can smoothly transition without an abrupt change in text position.

【Example】：

1. After creating an annotation label of a hole table, users can adjust the annotation text position

at will.

Before position adjustment	After position adjustment
	

【Where is it】：

Drawing Sheet >> Dimension >> Annotation

3.5.15 ★BOM & Balloon Improvement

When making balloon annotations, users can associate an existing BOM table in the drawing sheet with balloons to obtain different BOM information. Users can also attach balloons to the existing annotation objects so that different BOM information can be annotated quickly in the same view.

【What can be done】：

- ✓ The text control is activated automatically after the mouse determines the position point and text point of the annotation.
- ✓ When creating balloons and automatic balloons, users can select “Assembly Tree” and “Any BOM associated with this model” as the source of balloons so that different BOM information can be annotated in the same view.
- ✓ When users drag balloons to dimensions, balloons can be attached to and moved with the dimensions.
- ✓ The balloon style and attributes can be set to lead the branch on the nearest side, and the branch will be led from the bend point closest to the balloon symbol.

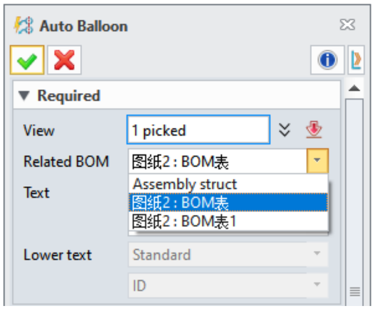
- ✓ Users can set the default spacing for balloons after attachment, called “combination spacing”.
- ✓ “Auto update IDs after sort” is added as a default option in BOM order rules to ensure that the BOM ID maintains ascending or descending order.
- ✓ Quickly find the components without being marked with balloons in the BOM table and highlight them in the view.

【Note】:

- Balloons can only be attached to the top, bottom, left, and right of a dimension text.
- The balloons after attachment can only be adjusted at the established position of the attached dimension. Adjustment can be made at the top, bottom, left, and right of a dimension text.
- Select the blue dot at the bottom of the attached balloon and drag it to disconnect the attachment between the balloon and the attached dimension.
- Multiple balloons can be attached in each direction of a dimension, but stacked balloons cannot be attached.
- When the balloon is set to “Aligned text”, “Horizontal text”, and the “Nearest side” property at the same time, the arrow branch extension line will pass through the center of the text.
- When a balloon is set to “Horizontal bend” and the " Nearest side " property is set at the same time, if the branch point is to the left of the center of the balloon text, the branch leads from the left side; If the branch point is to the right of the center of the balloon text, the branch leads from the right side.

【Example】:

1. When creating balloons and automatic balloons, after selecting the view, users can select "Assembly struct" or "Any BOM associated with this model" as the source of balloon information.



Auto Balloon

Required

View: 1 picked

Related BOM: 图纸2 : BOM表

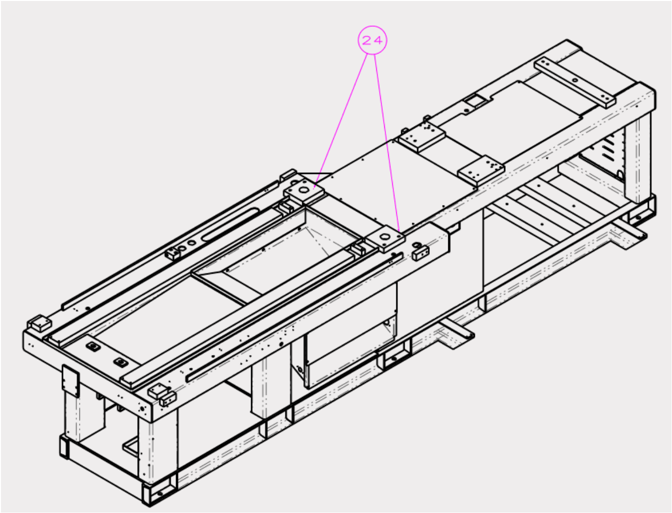
Assembly struct
图纸2 : BOM表
图纸2 : BOM表1

Text

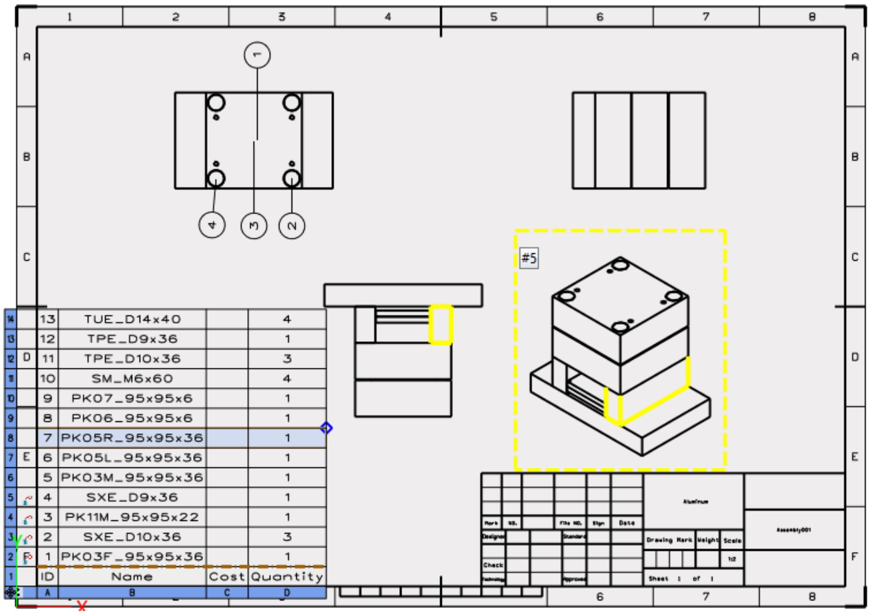
Lower text: Standard

ID

2. After a balloon is set to “Circular’, “Horizontal text”, and “Nearest side”, the branch leaders of balloons can be created without interference.



3. Select a component row in the table to highlight the component.



ID	Name	Cost	Quantity
13	TUE_D14x40		4
12	TPE_D9x36		1
11	TPE_D10x36		3
10	SM_M6x60		4
9	PK07_95x95x6		1
8	PK06_95x95x6		1
7	PK05R_95x95x36		1
6	PK05L_95x95x36		1
5	PK03M_95x95x36		1
4	SXE_D9x36		1
3	PK11M_95x95x22		1
2	SXE_D10x36		3
1	PK03F_95x95x36		1

【Where is it】：

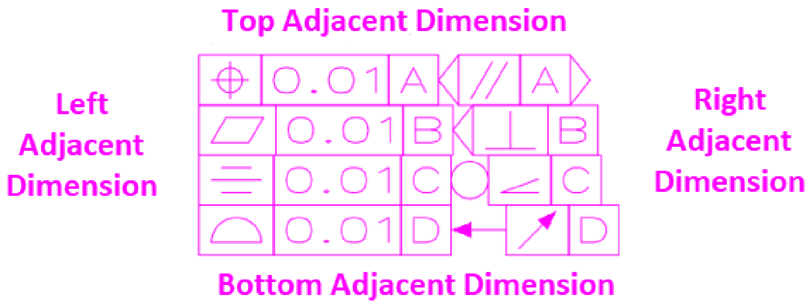
Drawing Sheet >> Dimension >> Balloon >> Dimension Attributes

3.5.16 ★Feature Control

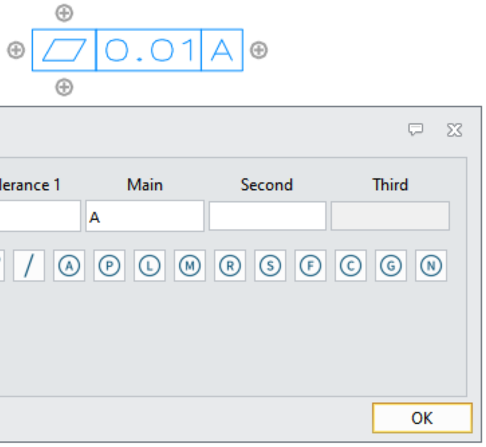
The feature control annotation covers GB/T 1182-2018 and ISO 1101:2017 standards, which achieves a quick adjustment of the style of feature control symbols. For example, support creating more than four lines of feature control annotations, auxiliary planes, and element frames, and drag features outside extension lines with their association maintained.

【What can be done】：

- ✓ Support creating more than four lines of feature control symbols.
- ✓ Support creating auxiliary planes and element frames.
- ✓ Support creating adjacent dimensions in the up, down, left, and right directions.

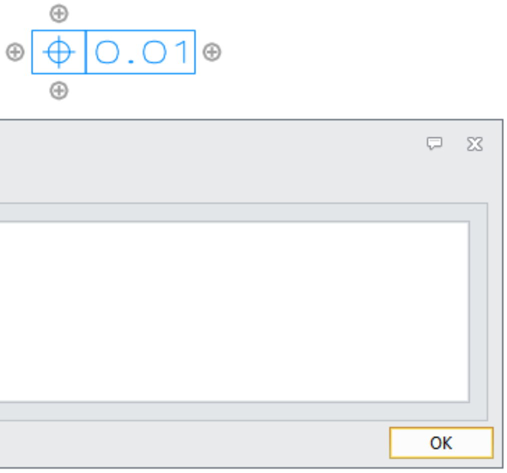

Top Adjacent Dimension
Left Adjacent Dimension
Right Adjacent Dimension
Bottom Adjacent Dimension
0.01 A
A
0.01 B
B
0.01 C
C
0.01 D
D

- ✓ Support building and editing feature control frames one by one.


0.01 A
Tolerance Indicator
OK

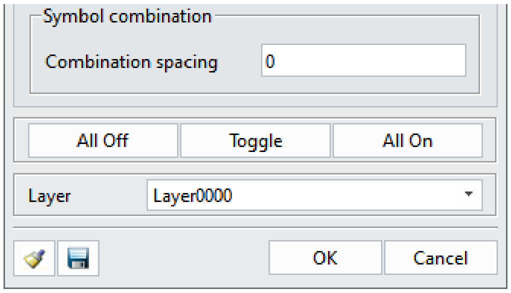
Frame	Symbol	Tolerance 1	Main	Second	Third
1	▱	0.01	A		

- ✓ Save, delete, and insert feature control symbols.


0.01
Tolerance Indicator
Edit Tol
Preset Tol
OK

example 平面度0.01

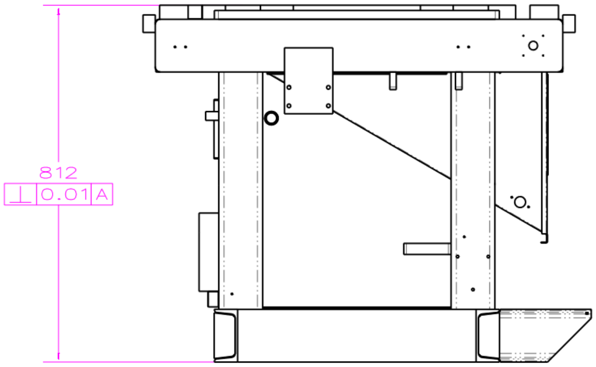
- ✓ Users can set the default spacing after attachment, called “combination spacing”.


Symbol combination

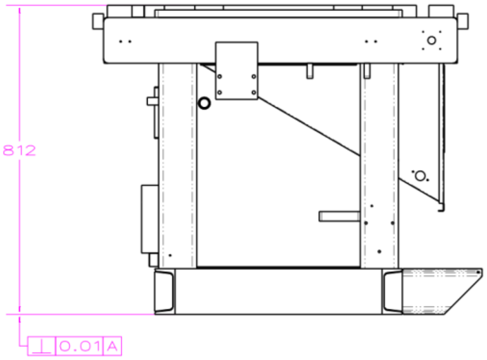
Combination spacing: 0

Layer: Layer0000

- ✓ Support setting leaders from “Nearest side”.
- ✓ After attaching to dimensions, the intersecting dimension lines can be automatically interrupted.



- ✓ Leaders can be automatically drawn when feature control symbols move outside extension lines.

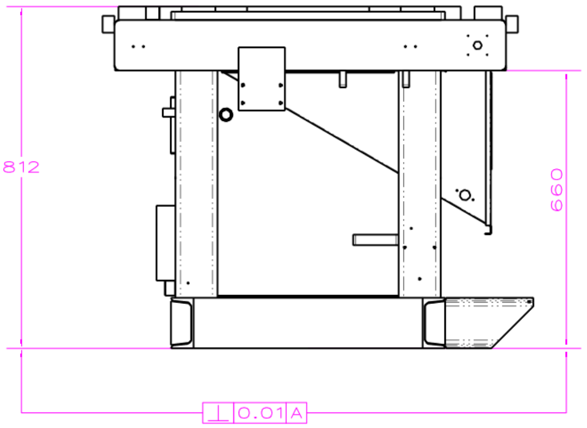


【Note】:

- When editing an old file in ZW3D 2025, its feature control symbols will be edited by Feature Control of the new version.

【Example】:

1. Leader can be automatically drawn when feature control symbols move outside extension lines, and the attached feature control symbols can create a branch pointing to another dimension by long pressing Ctrl and the left mouse button.



【Where is it】:

Drawing Sheet >> Dimension >> Feature Control

3.5.17 Center Mark

When marking holes, users can select circular arc objects in batches and create connection lines for centers to represent holes, which improves the efficiency and standardization of creating center marks and the correctness of the drawing sheet.

【What can be done】:

✓ Center Mark is separate from Center Line so that the two styles can be set separately.

✓ Center Mark can be created by box selecting circular arcs in batches. It also supports unselecting the objects by pressing Ctrl and the left mouse button.

✓ Center Mark can create connection lines for centers, and the pattern hole feature supports the automatic projection of connection lines for centers.

✓ Connection lines and center marks can be deleted separately from connection lines for centers.

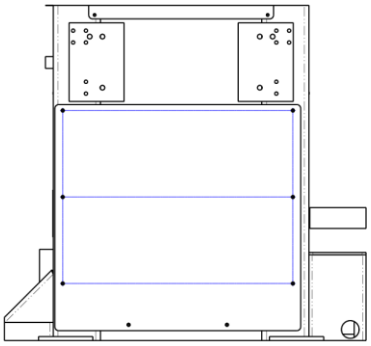
✓ Any arc connection line can be deleted separately from center mark circles.

【Note】:

- The pattern hole feature with more than two layers of nests does not support the automatic projection of center connection lines.
- Press Shift and the left mouse button to select center marks and center mark circle objects created at the same time and delete them in bathes.

【Example】:

1. When using the Center Mark command in the Drawing Sheet environment and checking the option "Create lines for center marks", lines for center marks will be created automatically during creating center marks.



【Where is it】:

Drawing Sheet >> Dimension >> Center Mark

3.5.18 Surface Finish Symbol

When creating surface finish symbols, users can select three dimension types: "No leader", "Leader" and "Unspecified", improving ease of use.

【What can be done】:

- ✓ Users can customize direction with the mouse when "No leader" surface finish symbols mark on dimension lines and extension lines.
- ✓ "Leader" surface finish symbols determine the position point first and then the text point, which is consistent with the workflow of symbol commands.
- ✓ "Unspecified" surface finish symbol can set the distance from the X and Y directions in the upper

right corner of a drawing sheet.

【Note】:

- Each drawing sheet can only create one " Unspecified " surface finish symbol.

【Where is it】:

Drawing Sheet>>Dimension>>Surface Finish

3.5.19 ★Drawing Sheet Block

When marking the drawing sheet blocks, users can create symbols with leaders and customized drop-down options, which helps users create more symbols in line with industry standards.

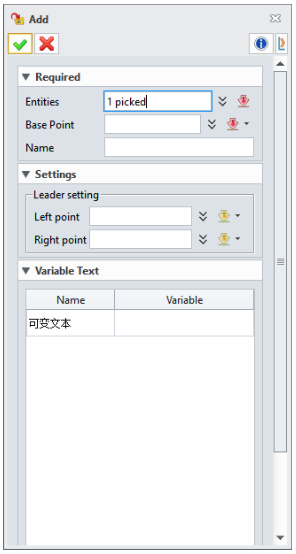
【What can be done】:

- ✓ Support setting styles and attributes for blocks.
- ✓ Blocks can be attached to a view and move with it.
- ✓ Support selecting the left and right leader points as the points that create leaders when the block is inserted.
- ✓ Branches of block leaders can be copied quickly.
- ✓ When creating blocks, support setting “variable” text as the drop-down option when inserting blocks.
- ✓ Support redefining the inputs of the block.

【Note】:

- Only one leader point can be selected.
- If the block is only set with the left insertion point, the insertion point can be entered into two points. The first point works as the base target point, while the second one works as the place point of the block base point. In addition, leaders can only be drawn from the left leader point.
- If the block is only set with the right insertion point, the insertion point can be entered into two points. The first point works as the base target point, while the second one works as the place point of the block base point. In addition, leaders can only be drawn from the right leader point.

- If the block is set with both the left and right leader points, the insertion point can be entered into two points. The first point works as the baseline target point, and the second one works as the block placement point. When the block base point is placed on the right side of the base point, the leader will be drawn from the left leader point. When the block base point is placed on the left side of the base point, the leader will be drawn from the right leader point.
- If a variable text format is “&text&”, all variable text options are listed when adding blocks, and multiple objects can be entered as drop-down options in the “variable” control. Different objects are separated by the symbol "\".



Entities: 1 picked

Name	Variable
可变文本	

- When inserting a block, users can select the required text based on the variable text set when the block was created.

【Example】:

1. When making a project block, use the text function definition format "&text&" to define a variable text object, and use the delimiter "\" to define a drop-down option value.
2. When creating a project block, you can set a branch creation point.

【Where is it】:

Drawing Sheet>>Dimension>> Add

3.5.20 Magnetic Line

When making dimensions in the Drawing Sheet environment, users can attach dimensions to magnetic lines during dimensioning.

【What can be done】：

✓ Attach dimensions to magnetic lines during dimensioning.

【Where is it】：

Drawing Sheet>>Drawing Area

3.5.21 Box Text

When creating a box text, users can make the box tilt along with the text after changing the inclination angle of the text, which facilitates users to adjust the position of the text.

【What can be done】：

✓ After changing the text’s inclination angle, the box tilts with the text.

✓ Boxes and position points are displayed when the mouse picks up and hovers over the text.

✓ Support dimensioning and adding constraints for location points and box auxiliary lines in the Sketch environment.

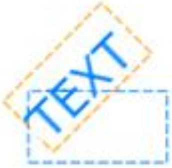
✓ Support selecting position points as the start point, target point, and base point in the Move, Copy, and Rotate commands.

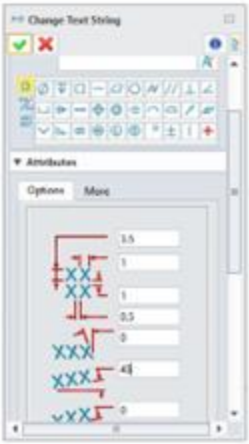
【Note】：

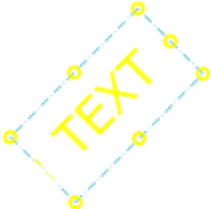
- An effect preview with an angle of 0 and a preview of the actual angle will be shown at the same time when adjusting the rotation angle.

【Example】

1. In the Drawing Sheet environment, setting the box text rotation angle will have the angle 0 and the actual angle preview effects, and the box will rotate with the text after the command is completed.







【Where is it】:

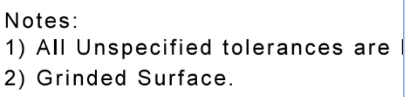
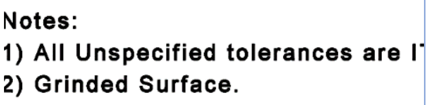
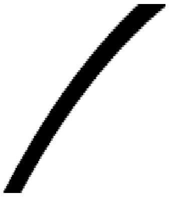
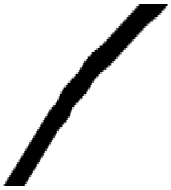
Drawing Sheet>Drawing Area>>Text>>Box Text

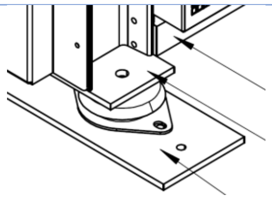
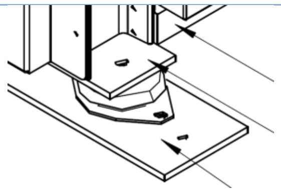
3.5.22 PDF Export & Print

Users need to export drawing sheets to PDF or print them after finish drawing to deliver and archive them. ZW3D 2025 reconstructs the codes of this part, greatly improving the accuracy and quality of printing.

【What can be done】:

- ✓ Printed and exported PDFs are smaller, with 68% of PDFs smaller than that of ZW3D 2024 in sizes and an average size reduction of 40%.
- ✓ Printed words are thinner and more beautiful, the picture clarity has been improved, and print arcs and axonometric lines are smoother.

Optimization	ZW3D 2025	ZW3D 2024
Prettier printed words	 Notes: 1) All Unspecified tolerances are 2) Grinded Surface.	 **Notes:** **1) All Unspecified tolerances are** **2) Grinded Surface.**
Smoother printed arcs and axonometric lines		

【Note】：

- A drawing sheet with more pictures will increase the exported PDF file size. The main increase is the picture size to ensure the quality of the exported picture.

【Where is it】：

Sheet Environment >> File >> Print/Plot

3.5.23 Isometric Section（ZW3D 2025 SP）

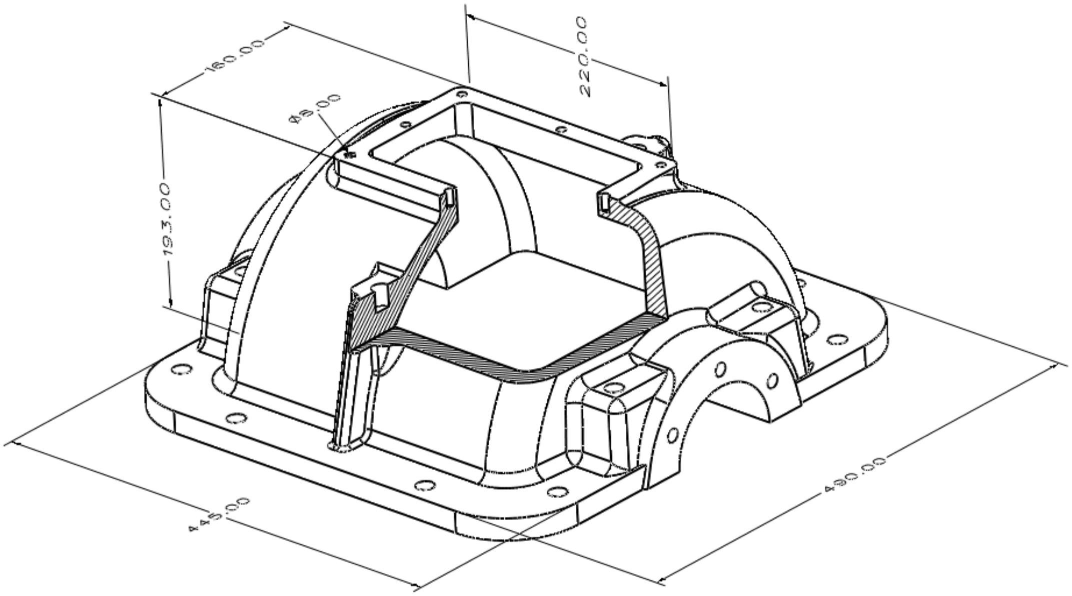
ZW3D 2025 SP optimizes the isometric section view of the engineering drawing.

Adds the inherit PMI function. The options include inherit parallel PMI, inherit all PMI, and inherit from view.

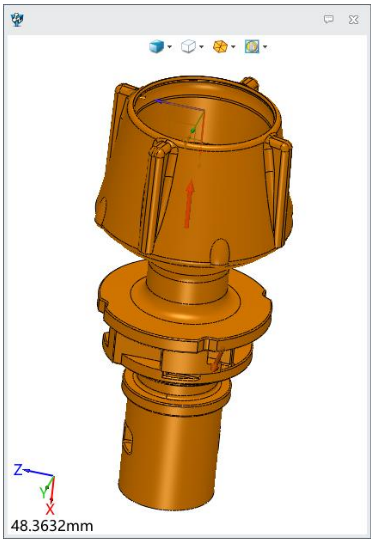
Adds orient view tool, supporting flexible positioning of sectional view projection direction.

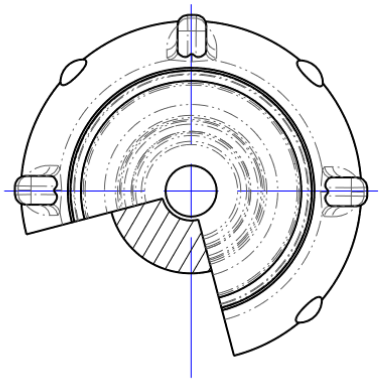
【What can be done】:

- Inherit the 3-dimensional PMI to the isometric section view.



- Use orient view tool to flexibly position the projection direction of section view.





【Where it is】:

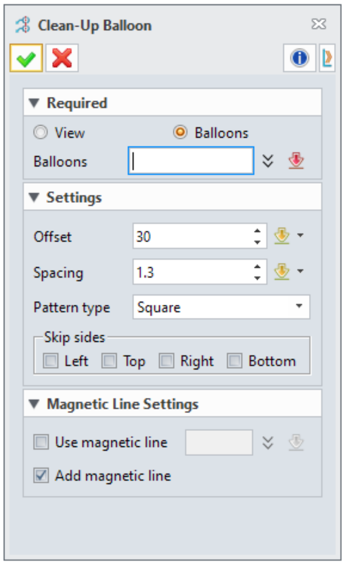
Drawing Sheet Environment >> Layout >> View >> Isometric Section

3.5.24 Clean-Up Balloon Optimization（ZW3D 2025 SP）

ZW3D 2025 SP has added a balloons mode to the clean-up balloon command, which can flexibly organize some balloon annotations in the view.

【What can be done】:

- ✓ Support batch picking of balloons through window pick and pick all;
- ✓ Support selecting existing magnetic lines and adsorbing balloons onto them;
- ✓ Add the option to create magnetic lines. When checked, the command will automatically create magnetic lines based on the offset, and balloons will automatically adhere to the magnetic lines.



Clean-Up Balloon

Required

○ View ● Balloons

Balloons

Settings

Offset 30

Spacing 1.3

Pattern type Square

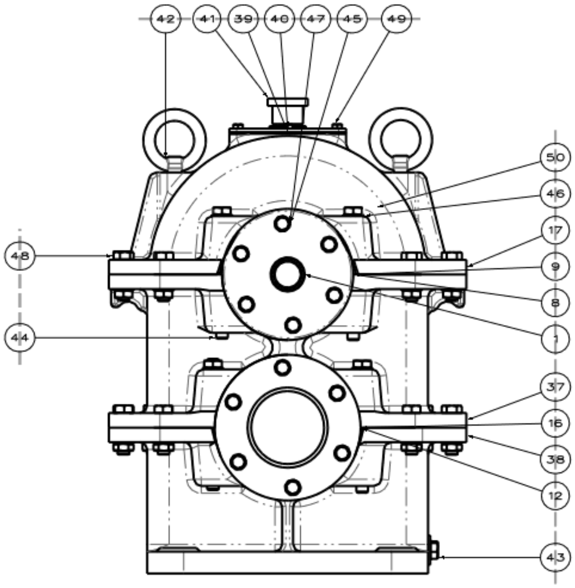
Skip sides

☐ Left ☐ Top ☐ Right ☐ Bottom

Magnetic Line Settings

☐ Use magnetic line

☑ Add magnetic line



【Where it is】:

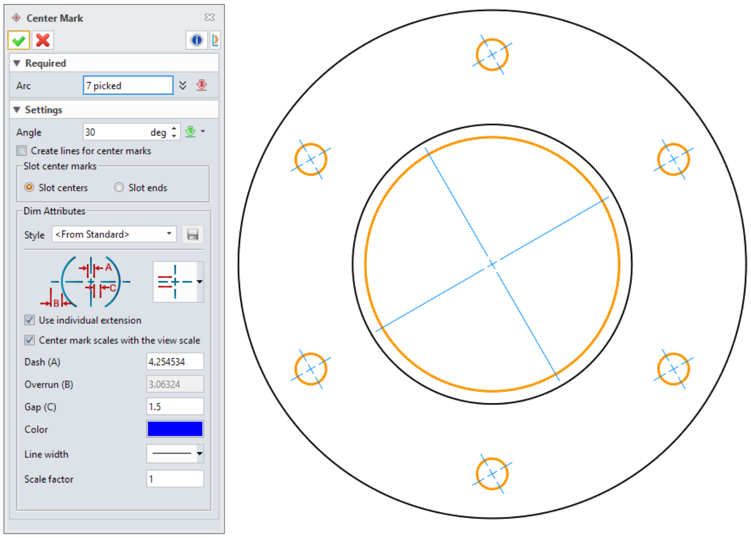
Drawing Sheet Environment >> Layout >> Edit Table >> Clean-Up Balloon

3.5.25 Center Mark Optimization（ZW3D 2025 SP）

ZW3D 2025 SP optimizes the center marking function of engineering drawings. When creating center marks, it supports rotating the center marks with accurate angle values to meet the scene of center mark rotation angles.

【What can be done】:

- ✓ The center marking command supports batch picking of arcs through window pick and pick all;
- ✓ The center mark command/rotate center mark command supports inputting accurate angle values to control the offset of the center mark.


Center Mark
Required
Settings
Slot center marks
Dim Attributes
A
B
C

Arc 7 picked
Angle 30 deg
☐ Create lines for center marks
● Slot centers ○ Slot ends
Style <From Standard>
☑ Use individual extension
☑ Center mark scales with the view scale
Dash (A) 4.254534
Overrun (B) 3.06324
Gap (C) 1.5
Color
Line width
Scale factor 1

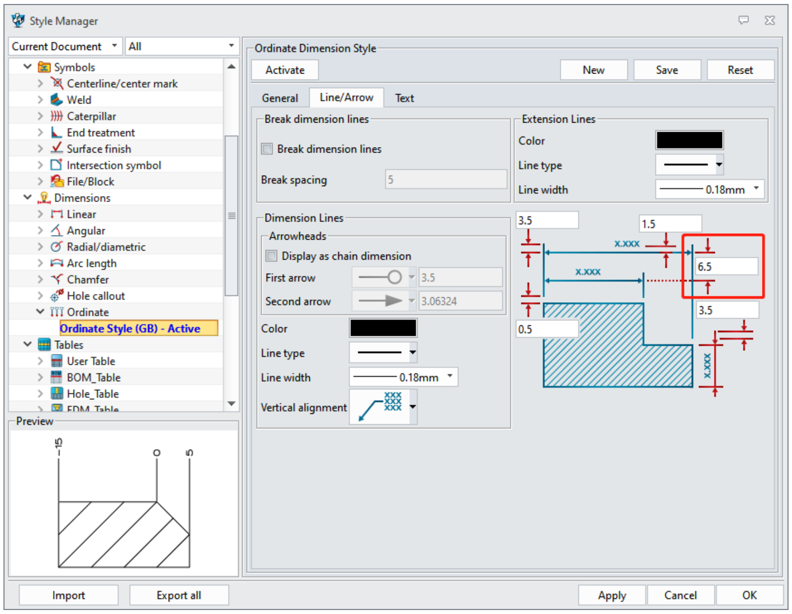
【Where it is】:

Drawing Sheet Environment >> Dimension >> Symbol >> Center Mark

Drawing Sheet Environment >> Dimension >> Edit Dimension >> Rotate Center Mark

3.5.26 Ordinate dimension optimization（ZW3D 2025 SP）

ZW3D 2025 SP support the setting of dimension line spacing for ordinate dimension.


Style Manager
Current Document
All
Symbols
Centerline/center mark
Weld
Caterpillar
End treatment
Surface finish
Intersection symbol
File/Block
Dimensions
Linear
Angular
Radial/diametric
Arc length
Chamfer
Hole callout
Ordinate
Ordinate Style (GB) - Active
Tables
User Table
BOM_Table
Hole_Table
EDM_Table
Preview
-15
0
5
Ordinate Dimension Style
Activate
New
Save
Reset
General
Line/Arrow
Text
Break dimension lines
Extension Lines
Dimension Lines
Arrowheads
x.xxx
x.xxx
x.xxx
Import
Export all
Apply
Cancel
OK

☐ Break dimension lines
Break spacing 5
Color
Line type
Line width 0.18mm
☐ Display as chain dimension
First arrow 3.5
Second arrow 3.06324
Color
Line type
Line width 0.18mm
Vertical alignment
3.5
1.5
6.5
3.5
0.5

【What can be done】:

- ✓ In the Style Manager, preset the dimension line spacing for ordinate dimension.
- ✓ Dimension object with radio ordinates can change the interval between the selected object and the previous labeled object.
- ✓ Multi-selection ordinate dimension objects are supported, and the interval between the selected object and the previous labeled object can be changed in batches.

【Notes】:

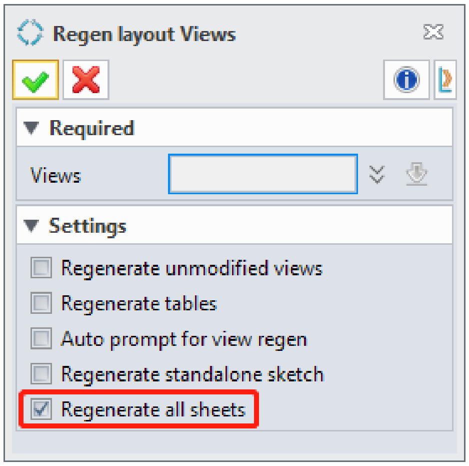
- This attribute only takes effect for the ordinate dimension of bending.

【Where it is】:

Drafting > > Ordinate dimension attributes > >Misc > > Dimension line spacing

3.5.27 Regenerate all sheets（ZW3D 2025 SP）

ZW3D 2025 SP support regenerate all sheets.



☐ Regenerate unmodified views
☐ Regenerate tables
☐ Auto prompt for view regen
☐ Regenerate standalone sketch
☑ Regenerate all sheets

【What can be done】:

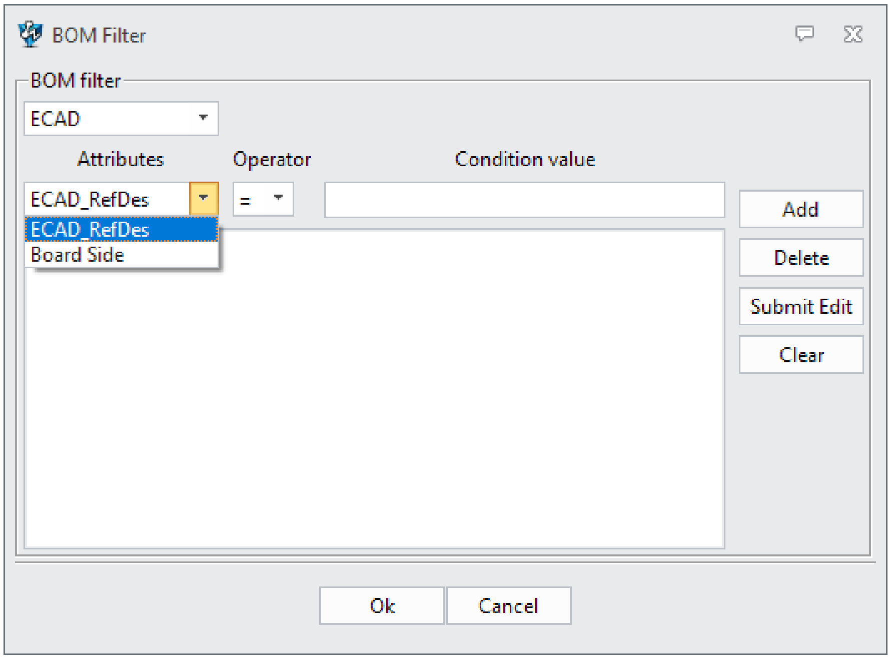
- ✓ Support regeneration of all sheets of the current drawing.
- ✓ Regenerate all sheets controls to remember the last selection.

【Where it is】:

Drafting>>Regen layout views>>Regenerate all sheets

3.5.28 BOM filter optimization（ZW3D 2025 SP）

ZW3D 2025 SP optimize the BOM filter of the drawing, add ECAD_RefDes and Board Side attributes, and support saving filter conditions in BOM template.


BOM Filter
BOM filter
Attributes
Operator
Condition value
ECAD_RefDes
Board Side
Add
Delete
Submit Edit
Clear
Ok
Cancel

BOM filter: ECAD

Attributes: ECAD_RefDes

Operator: =

【What can be done】:

- Add ECAD_RefDes and Board Side attributes to BOM filter.
- BOM filters are divided into 5 categories: standard, physical attribute, file, ECAD and user-defined, and the attributes can be filtered as required.
- Filter conditions set by BOM filter can be saved in BOM template.

【Where it is】:

Drafting>>Layout>>BOM>>Filter

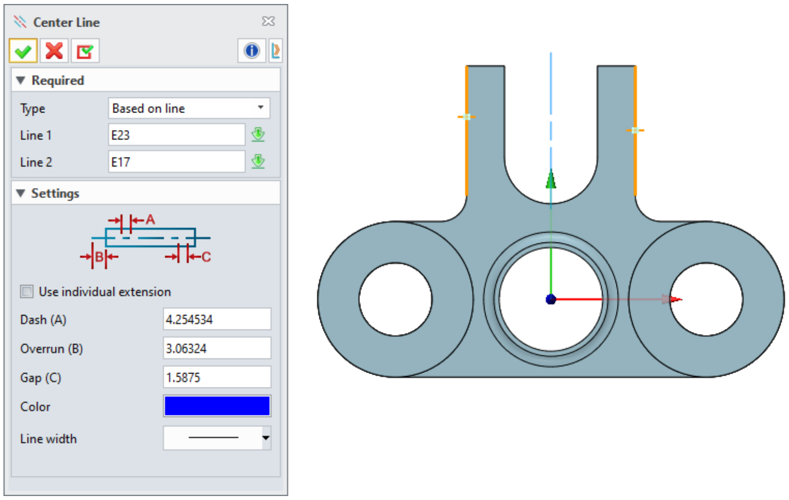
3.6 PMI

3.6.1 New Way to Create Center Line by Straight Line（ZW3D 2025 SP）

ZW3D 2025 SP optimizes the function of centerline creation in PMI, and adds a new way of creating by straight line, which provides a more convenient way of creating centerline, with a wider coverage of scenes, and greatly improves the efficiency of users in creating centerline.

【What can be done】:

- ✓ Create centerlines based on straight lines and straight edges



Center Line

Required

Type: Based on line

Line 1: E23

Line 2: E17

Settings

☐ Use individual extension

Dash (A): 4.254534

Overrun (B): 3.06324

Gap (C): 1.5875

Color: 🟦

Line width

【Where is it】:

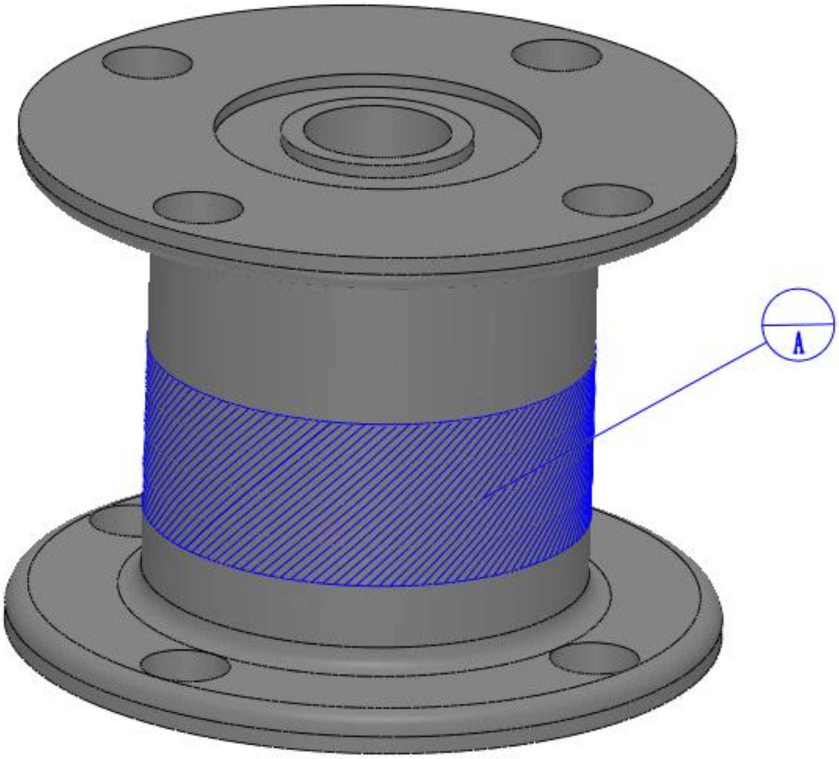
Part/Assembly >> PMI >> Properties

3.6.2 Datum Target Optimization （ZW3D 2025 SP）

ZW3D 2025 SP optimizes the datum target function in the PMI module, adds a new cylinder type to the original basis, and supports adding a datum target to the objects of the cylinder. After optimization, this function can cover a wider range of usage scenarios.

【What can be done】:

- ✓ Add the datum targets to the cylindrical objects.



【Where is it】:

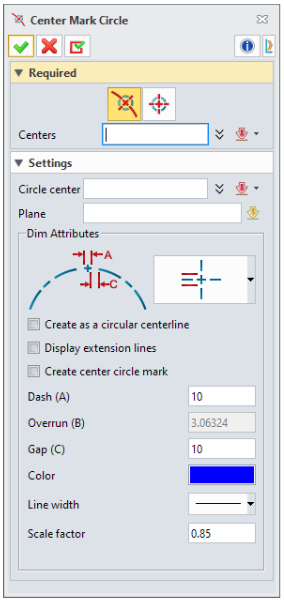
Part/Assembly >> PMI >> Annotation >> Datum Target

3.6.3 New Center Mark Circle（ZW3D 2025 SP）

ZW3D 2025 SP adds the center mark circle function in the PMI module, which supports the creation of center mark circles through the center and diameter modes. This function enriches the annotation types of PMI and satisfies more usage scenarios.

【What can be done】:

- ✓ Create center-marked circles through the center.



Center Mark Circle

Required

Centers

Settings

Circle center

Plane

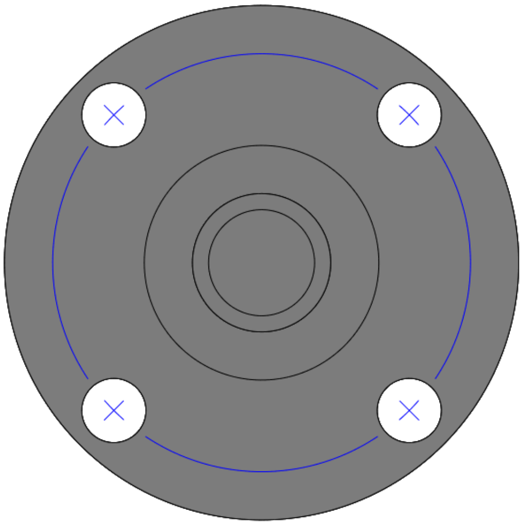
Dim Attributes

☐ Create as a circular centerline

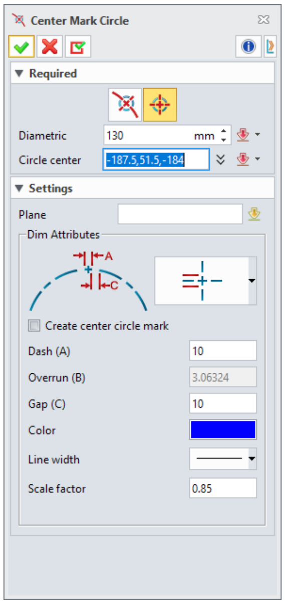
☐ Display extension lines

☐ Create center circle mark

Dash (A)	10
Overrun (B)	3.06324
Gap (C)	10
Color	🟦
Line width	
Scale factor	0.85



- ✓ Create a center marker circle by diameter



Center Mark Circle

Required

Diametric	130 mm
Circle center	-187.5,51.5,-184

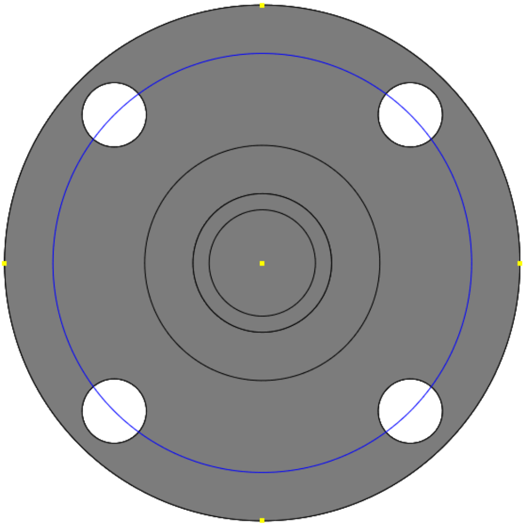
Settings

Plane

Dim Attributes

☐ Create center circle mark

Dash (A)	10
Overrun (B)	3.06324
Gap (C)	10
Color	🟦
Line width	
Scale factor	0.85



【Where is it】：

Part/Assembly >> PMI >> Annotation >> Center Mark Circle

4 Industry Application

4.1 Sheet Metal Design

The Transition function has been added to Twist Flange and Fold by Line in ZW3D 2025, which supports creating complex sheet metal features such as connection terminals and twisting copper bars for customers in 3C, electrical equipment, and other industries.

The Normal Cut command has been optimized and reconstructed in depth, and two kinds of normal cut methods have been added, namely Normal to the Near Side and Normal to the Far Side, to meet more design scenarios.

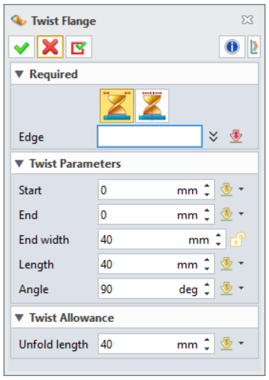
Profile flanges support multiple selections, which can meet customers’ core demand for creating multiple flanges in a single command when designing multiple local flanges, optimizing the design process.

4.1.1 New Twist Flange

3C, electrical equipment, and other industries usually use connection terminal/copper bars for electrical connection. The design of such parts usually needs to twist at a certain angle for the actual installation. The new Twist Flange of ZW3D 2025 can meet the design needs of such scenes.

【What can be done】:

- ✓ Support self-defining twist start width, end width, length, and twist angle as well as unfolding length.

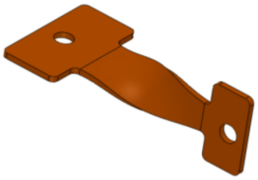
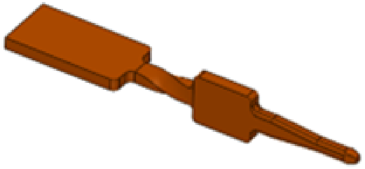


Edge
Start 0 mm
End 0 mm
End width 40 mm
Length 40 mm
Angle 90 deg
Unfold length 40 mm

【Note】:

- Twist area edge fillets or chamfers are limited by fillets, which may lead to errors (fillet broken surface or fillet broken surface after fillet). Therefore, this method is not recommended.

【Example】：

Twist Copper Bar	Connection Terminal
	

【Where is it】：

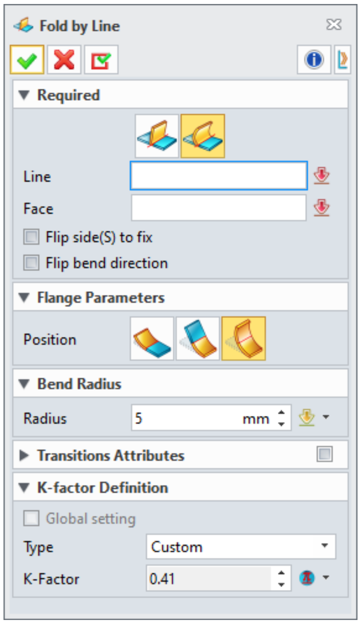
Sheet Metal Environment>Sheet Metal>>Flange>>Twist Flange

4.1.2 New Curl Transition in Fold by Line

3C, electrical equipment, and other industries usually use terminals for electrical connection, the round end of the terminal and the open plate need a smooth transition. ZW3D 2025 has added Curls Transition in Fold by Line that can satisfy the needs of this design scene.

【What can be done】：

- ✓ In the design process, users can customize the transition range through Sketch. Reversing the curl direction and customizing the curl radius and curl position are both supported.



Fold by Line

Required

Line:

Face:

☐ Flip side(S) to fix

☐ Flip bend direction

Flange Parameters

Position

Bend Radius

Radius: 5 mm

Transitions Attributes ☐

K-factor Definition

☐ Global setting

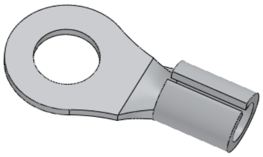
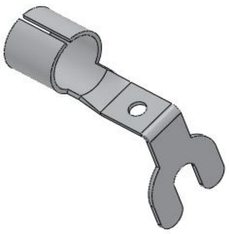
Type: Custom

K-Factor: 0.41

【Note】:

- The fillets or chamfers of the spline curve at the transition are limited by the fillet, which may lead to errors (fillet broken surface or fillet broken surface after fillet). Therefore, this method is not recommended.

【Example】:

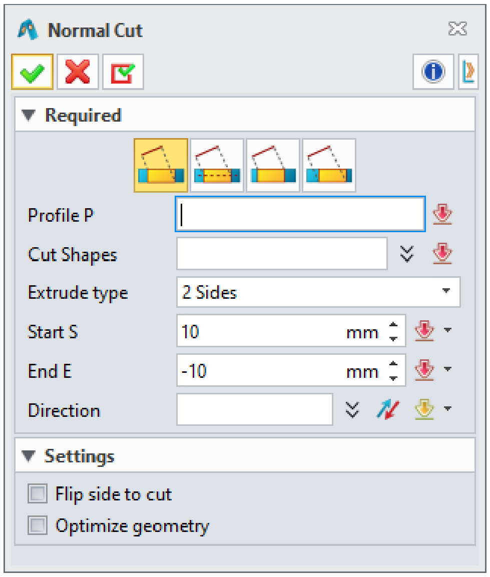
Connection Terminals	
	

【Where is it】:

Sheet Metal Environment>Sheet Metal>>Fold by Line>>Curl

4.1.3 New Normal Cut Methods

Two normal cut methods have been added, namely Normal to Near Side and Normal to Near Side, to satisfy more design scenes.



Normal Cut

Required

Profile P:

Cut Shapes:

Extrude type: 2 Sides

Start S: 10 mm

End E: -10 mm

Direction:

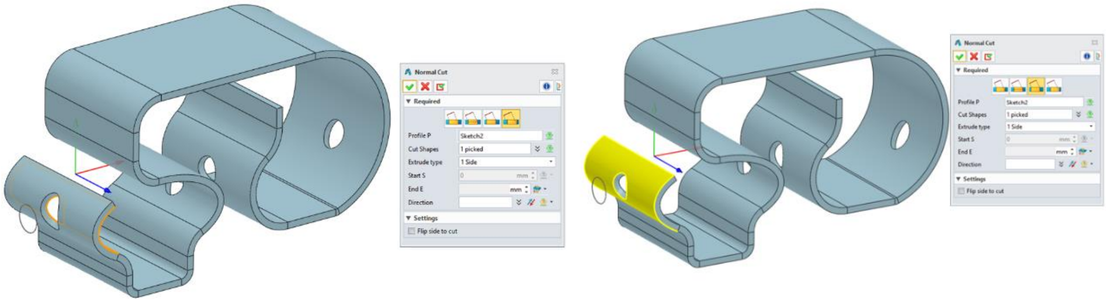
Settings

☐ Flip side to cut

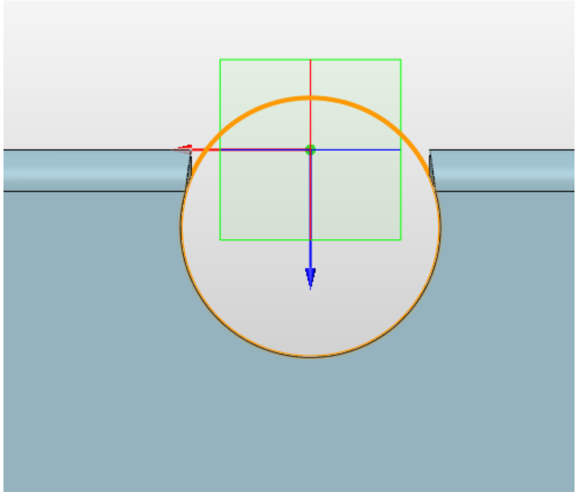
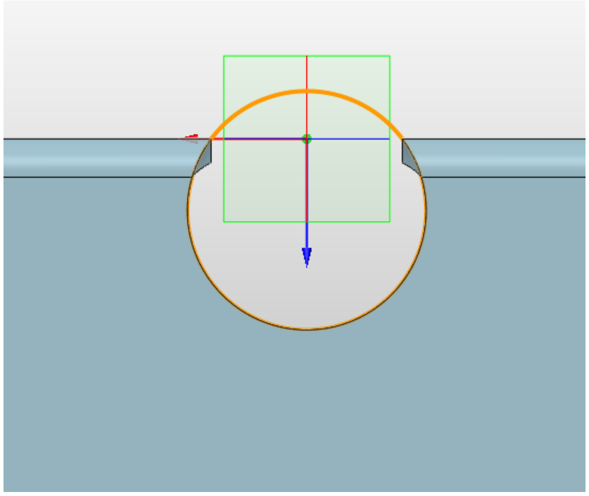
☐ Optimize geometry

【What can be done】:

- ✓ As shown in the following figure, when performing normal cutting according to different design requirements, users can project the area that requires to be cut (that is, the sketch profile) on the near-side sheet metal surface or the far-side sheet metal surface of the sketch plane to obtain the required design size.



【Example】：

Perpendicular to the Near Side	Perpendicular to the Far Side
	

【Where is it】：

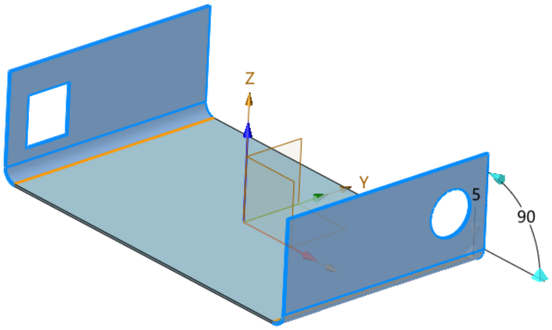
Sheet Metal Environment>Editing>>Normal Cut

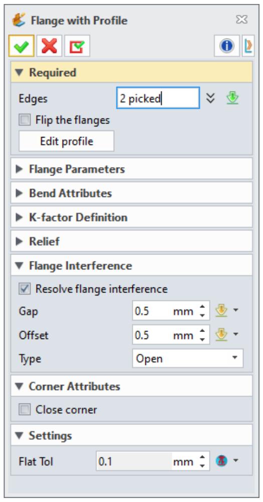
4.1.4 Flange with Profile Improvement

At the beginning of sheet metal design, when adding flanges after drawing the extruded plate, users are not sure whether the target bend to be designed is “full flanges” or “partial flanges”. In general, multiple "full flanges" will be added first. In the subsequent detailed design, users can modify the start and end of the flange width, as well as the length and angle, according to the position and purpose of the sheet metal parts in the assembly. The limitations of the full and local flange functions and the lack of multiple choices for flanges with profile lower customers’ design efficiency.

【What can be done】：

- ✓ ZW3D 2025 has improved Flange with Profile to support multiple choices. As shown below, users can create flanges with different profiles within a single command.





Flange with Profile

Required
Edges: 2 picked
☐ Flip the flanges
Edit profile

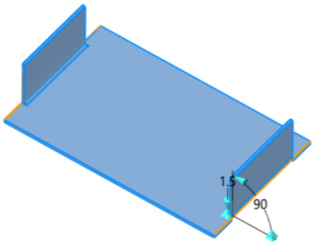
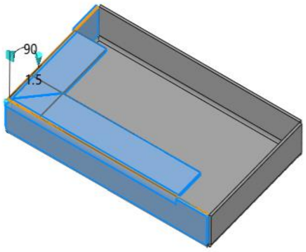
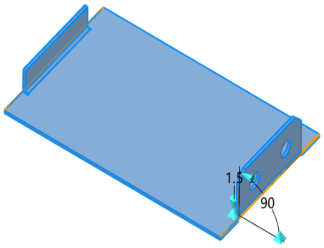
Flange Parameters
Bend Attributes
K-factor Definition
Relief

Flange Interference
☑ Resolve flange interference
Gap: 0.5 mm
Offset: 0.5 mm
Type: Open

Corner Attributes
☐ Close corner

Settings
Flat Tol: 0.1 mm

【Example】:

Support creating multiple-selected edges at once	Support multiple-selected edges to eliminate interference	Support multiple-selected edges to create different profiles
		

【Where is it】:

Sheet Metal Environment>Flange>>Flange with Profile

4.2 Structure & Weldment Design

Considering the customer's structural member design process and the actual engineering scene, we improve the process logic, ease of use, guidance, and correctness of each function, which strengthens the soft power of the Structural Member module that customers care about.

The new commands "**Edit Structure**" and "**Structure Wizard"** can greatly improve the design efficiency of customers. The overall improvement of the BOM function of joints and structural members makes the function results meet the actual needs of real engineering scenarios. Please check the following chapters and the help documentation for other improvements and detailed descriptions.

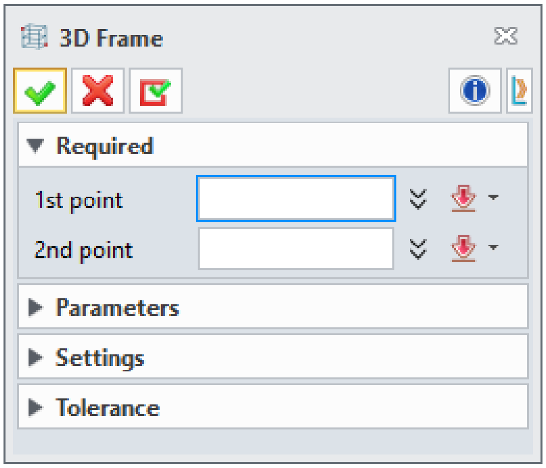
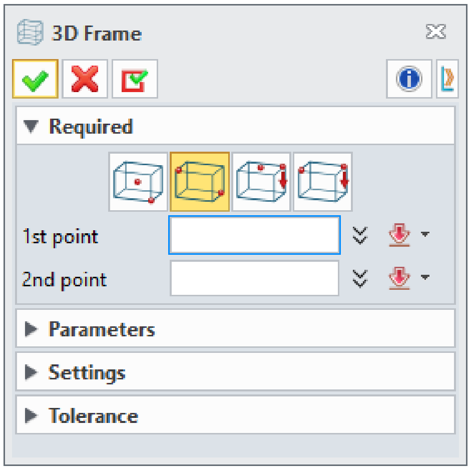
4.2.1 Structure Line Drawing

4.2.1.1 New Create Types in 3D Frame

In the real scene of 3D frame design, in addition to “Corner”, ZW3D 2025 has added three create types: “Center”, “Center–Height”, and “Corner–Height”.

【What can be done】：

- ✓ The 3D frame created through “Center”, “Center–Height”, and “Corner–Height” can be applied to many different user scenarios.

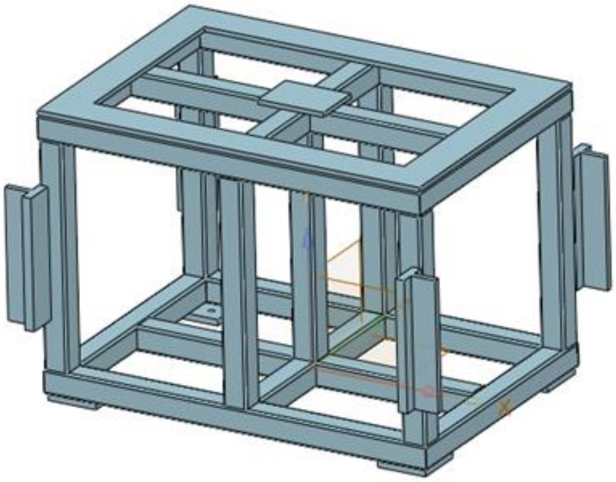
ZW3D 2024	ZW3D 2025
 3D Frame Required 1st point 2nd point Parameters Settings Tolerance	 3D Frame Required 1st point 2nd point Parameters Settings Tolerance

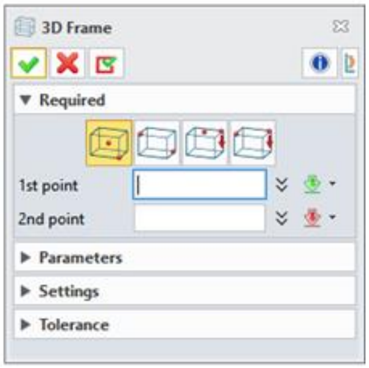
【Note】：

- Parameters and controls such as “Amont” and “Single Line” are not supported to be modified when editing a 3D frame.

【Example】：

1. The "Center-Height" type is used to create a 3D frame when you create a profile frame line based on the center-symmetry of the Z-axis in the WCS as shown below.





【Where is it】:

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

4.2.2 Structural Member Creation & Edition

4.2.2.1 Easy-to-use Structural Member Picking

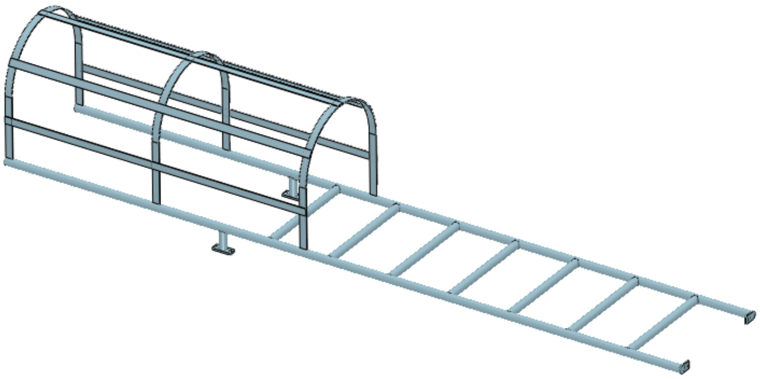
In ZW3D 2024, if users need to create line and curve structural members at the same time, the Structural Member command cannot select the linear lines and curves simultaneously. ZW3D 2025 has optimized the problem so that users can select both linear lines and curves to create structural members in batches.

【What can be done】:

✓ Select both linear lines and curves to create structural members in batches.

【Example】:

1. When the user creates the ladder barrier shown below, linear lines and curves are both selected by default to create the structural members.



【Where is it】:

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

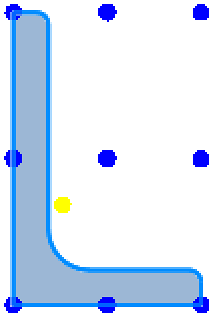
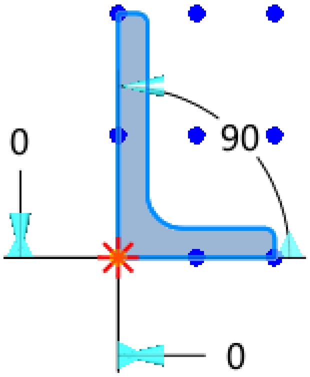
4.2.2.2 Structural Member Initial Default Coordinate Point and Drawing Area Display Optimization

When creating structural members in actual scenes, on the one hand, customers have different default anchor point requirements for different types of profile sections. However, ZW3D 2024 sets default anchor points for all types of structural member sections to the centroid of the structural member sections. On the other hand, the default anchor point is indistinct. In ZW3D 2025, on the one hand, default anchor points of various profiles have been optimized. On the other hand, the display of default anchor points has been clearer, and the orientation adjustment size display has been added, improving the correctness and ease of use of ZW3D.

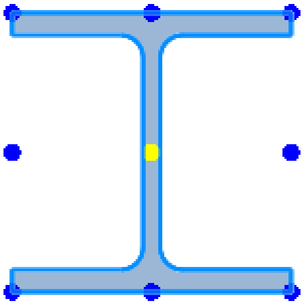
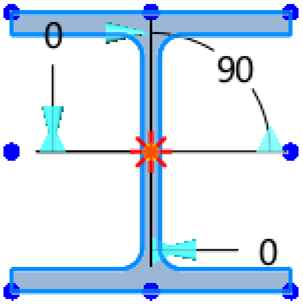
According to the actual design scene, default anchor points of each structure has been optimized as follows:

1. Angular steel: Adjust the corner point at the bottom left.

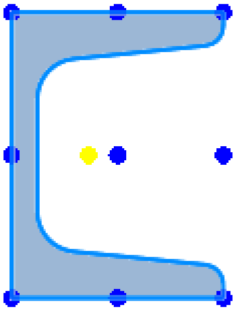
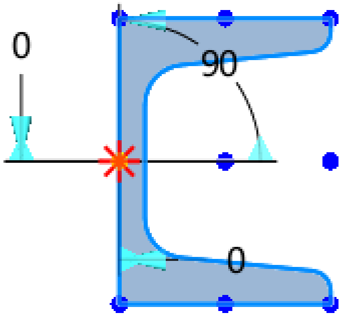
ZW3D 2024	ZW3D 2025

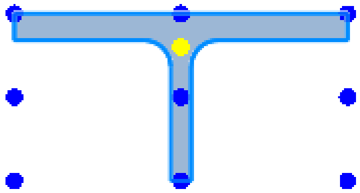
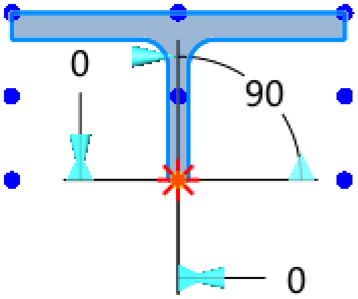
2. Bidirectional symmetry: Adjust the geometry center.

ZW3D 2024	ZW3D 2025
	

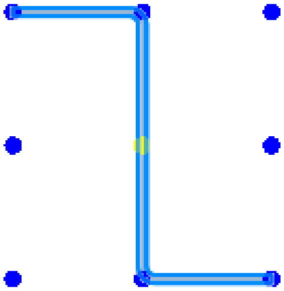
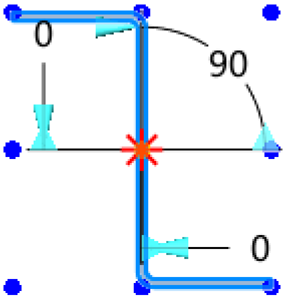
3. C type: Adjust at center left.

ZW3D 2024	ZW3D 2025
	

4. T type: Adjust at bottom center.

ZW3D 2024	ZW3D 2025
	

5. Z type: Adjust at geometry center.

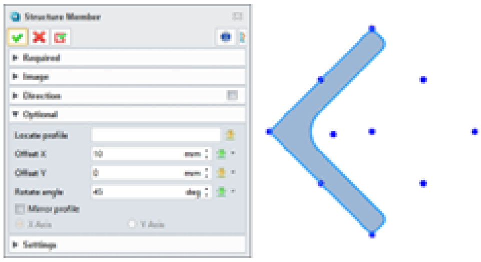
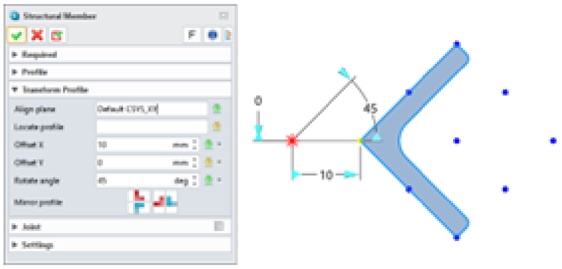
ZW3D 2024	ZW3D 2025
	

【What can be done】:

- ✓ Create different structural members and provide different types of default anchor points commonly used by users.
- ✓ Display the size change of structural member position in a clearer way in the drawing area.

【Example】:

1. When creating a structural member, select equal angular steel, move its section the axis X forward by 10mm without changing the axis Y, and rotate it by 45°. The effect is as follows:

ZW3D 2024	ZW3D 2025
	

【Where is it】：

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

4.2.2.3 ★Unify Structural Member Direction and Auto-Creating Joint Optimization

When creating structural members in batches, users always need to unify the structural member orientation and create suitable joints at the intersection between structural members. In ZW3D 2025, even if the selected structural member line does not form a closed ring, it can also unify structure orientation and create joints automatically. It can also create joint forms, including 4 major categories of joints Miter, End Trim, T-Joint, and Lap, and 9 minor categories.

【What can be done】：

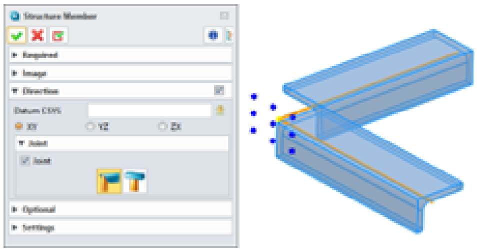
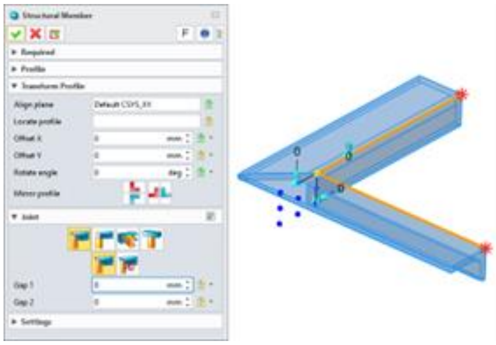
- ✓ Unify all structural member orientation parallel to the selected align planes.
- ✓ For the selected structure lines, when there is an interaction point between two lines, the joint can be processed automatically as well as create joint forms, including 4 major categories of joints Miter, End Trim, T-Joint, and Lap, and 9 minor categories.

【Note】：

- When selecting a lap joint type, the base and the tool of the structure are undefinable yet.
- Currently, ZW3D 2025 is unable to automatically deal with the joint position of three structure lines intersecting at the same point.

【Example】：

1. When users create two structure members and need to unify orientation and miter joints, the effect is as follows:

ZW3D 2024	ZW3D 2025
	

【Where is it】:

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

4.2.2.4 New Merge Collinear Paths

When creating structural members, users often find that two structural member lines are selected but only one structural member is created. ZW3D 2025 has solved this problem by providing the Merge Collinear Paths function.

【What can be done】:

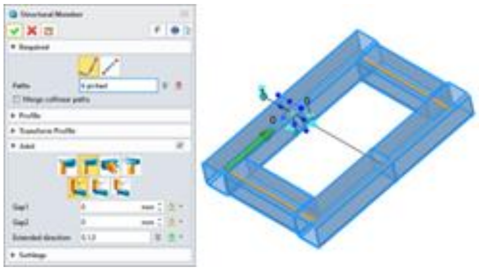
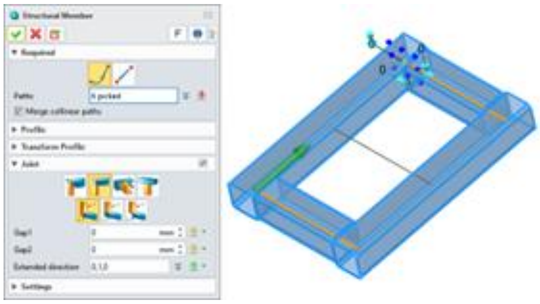
✓ In creating structural members in batches, the Merge Collinear Paths function can create a single structural member for multiple collinear structural member lines.

【Note】:

- Currently, the function cannot directly merge a linear structure line with a tangent bending structure line, or two tangent bending structure lines.

【Example】:

1. In the case of creating outer side structures, when “Merge collinear paths” is unchecked, 6 structural members are created; when “Merge collinear paths” is checked, 4 structural members are created.

“Merge collinear paths” checked	“Merge collinear paths” unchecked
	

【Where is it】:

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

4.2.2.5 New Create a Structural Folder

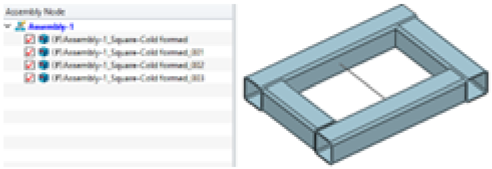
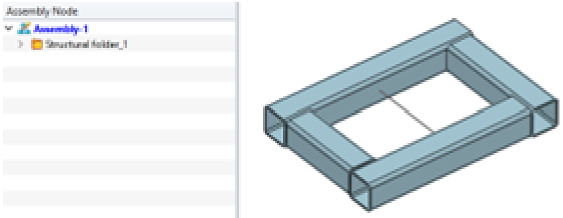
ZW3D 2025 has added the “Create a structural folder” checkbox for users to decide whether to uniformly save structural members created when executing a Structural Member command in one folder to avoid a messy assembly tree.

【What can be done】:

- ✓ In creating structural members in batches, save structural members created when executing a Structural Member command in one folder by checking “Create a structural folder”.

【Example】:

1. When creating the scenario in the example from the previous section, the results of checking “Create a structural folder” and unchecking “Create a structural folder” are as follows:

Checked “Create a structural folder”	Uncheck “Create a structural folder”
	

2. Based on the result shown in the above right figure, you can quickly edit all structures in the structural member folder in batches by using the Edit command in the right-click menu of the structural member.

【Where is it】：

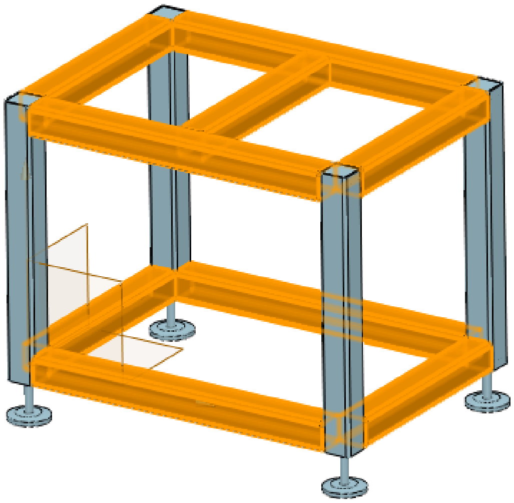
Assembly Environment >> Structure >> Profile >> Structural Member

4.2.2.6 ★New Batch Editing Structure Function

In the design process of an actual structural member frame, users always need to edit the type, specification, location, material, etc. of structures in batches because of the requirements of program modification, bearing capacity adjustment, equipment installation space, and frame overall appearance.

【What can be done】：

- ✓ This function can both batch edit a single item and multiple items of the selected structural member.
- ✓ To batch modify the specifications of the 9 highlighted secondary beams in the following figure, ZW3D 2024 takes 73 seconds, while ZW3D 2025 only takes 17 seconds by using the Batch Edit Structure function for structural members.

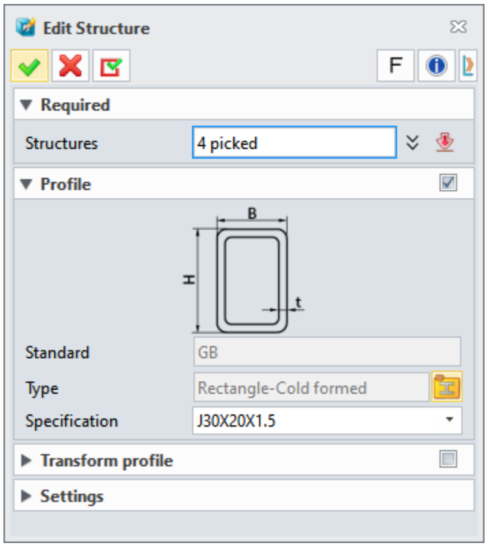


【Note】：

- Currently, the function cannot batch edit structural members and joints.

【Example】:

1. Use the “Edit Structure” command in the Assembly environment and check the profile to batch edit the profile standard, type, and specification.



Edit Structure

▼ Required

Structures: 4 picked

▼ Profile ☑

Standard: GB

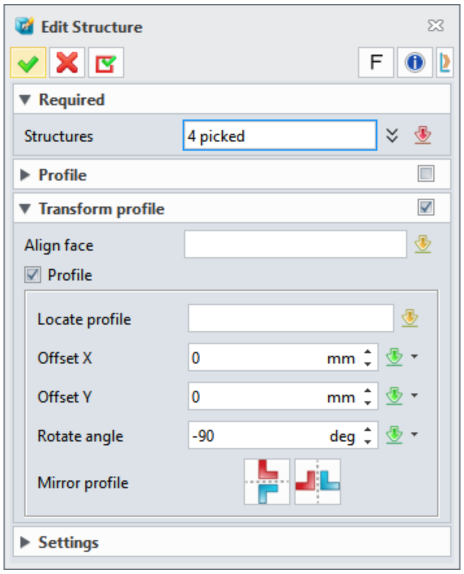
Type: Rectangle-Cold formed

Specification: J30X20X1.5

▶ Transform profile ☐

▶ Settings

2. Use the “Edit Structure” command in the Assembly environment and check the “Transform profile” option to batch edit align face, locate profile, offset, rotate angle, and mirror profile.



Edit Structure

▼ Required

Structures: 4 picked

▶ Profile ☐

▼ Transform profile ☑

Align face:

☑ Profile

Locate profile:

Offset X: 0 mm

Offset Y: 0 mm

Rotate angle: -90 deg

Mirror profile

▶ Settings

【Where is it】:

Assembly Environment >> Structure >> Profile >> Edit Structure

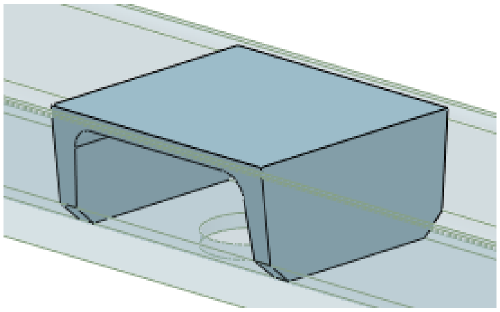
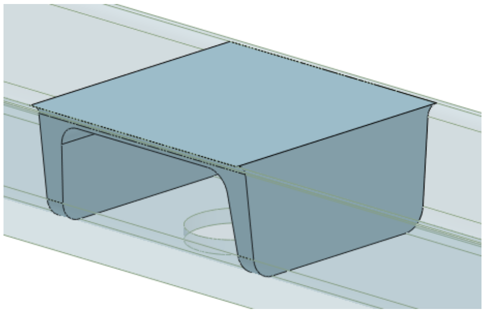
4.2.3 Joint Creation and Editing

4.2.3.1 New Simple Lap Function

In fact, simple lap scenes are more common in practical applications. There are two advantages to the result in practical engineering: 1. It is easy to process with flat cutting at the red circle of a base structural member; 2. Chamfers and fillets are much easier to match together with low accuracy.

【What can be done】：

- ✓ The effects of simple lap and tight lap are as follows:

Simple lap	Tight lap
	

【Example】：

1. Simple lap is much easier to use in matching fillets and chamfers in the application scene that requires low accuracy.
2. Tight lap can be used in the application scene that requires high accuracy.

【Where is it】：

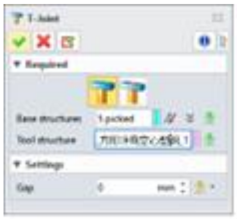
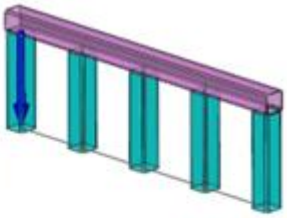
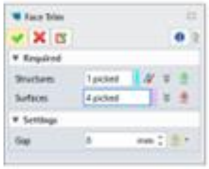
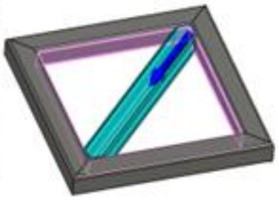
Assembly Environment >> Structure >> Advanced Joint >> Lap

4.2.3.2 ★New Batch Create Joint Function

In ZW3D 2024, the T-Joint and Face Trim commands can only create a single joint, not supporting creating joints in batches, which greatly decreases the design efficiency of customers. ZW3D 2025 supports editing T-Joints and Face Trim in batches.

【What can be done】：

✓ Create T-joints and face trim in baches.

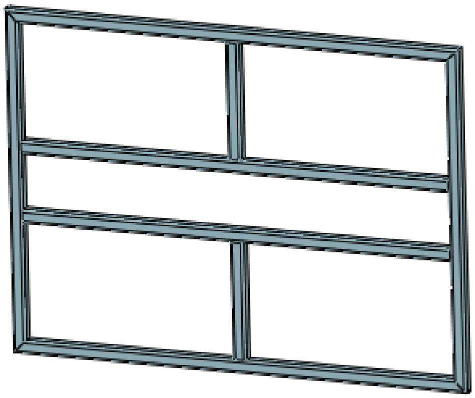
Create T-Joints in Batches	Create Face Trim in Batchse
 T-Joint Required Settings Base structure 1 picked Gap 0 mm	 Face Trim Required Settings Structures 1 picked Surfaces 4 picked Gap 0 mm

【Note】：

- When creating T-joints in batches, the function does currently not support multiple-selecting tool structures.

【Example】：

1. The following door frame can be created through editing T-Joint and Face Trim in batches.



【Where is it】：

Assembly Environment >> Structure >> Joint >> T-Joint/ Face Trim

4.2.4 Element Creating and Editing

4.2.4.1 Support Switching Element Types and Location Faces While Editing Element

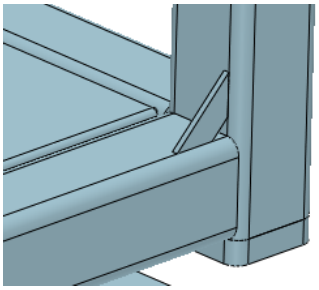
ZW3D 2025 has added the Edit Element command to support switching the element types and location faces, meeting the needs of modifying element types and location faces.

【What can be done】:

- ✓ Support switching element types with Edit Element.
- ✓ When executing the Edit Element command, you can also switch the element location surfaces if you do not need to switch the component type.

【Example】:

1. As the below triangular gusset shown, you can use the Edit Element command to switch the type to rectangular gusset.



【Where is it】:

Assembly Environment >> Structure >> Element >> Edit Element

4.2.5 Structural BOM

4.2.5.1 New Attribute Fields and Support Partial Online Editing

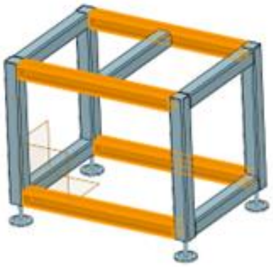
ZW3D 2025 has added 4 attribute fields: Designer, Manager, Supplier, and Cost, and supports online editing Description, Number, Designer, Manager, Supplier, and Cost.

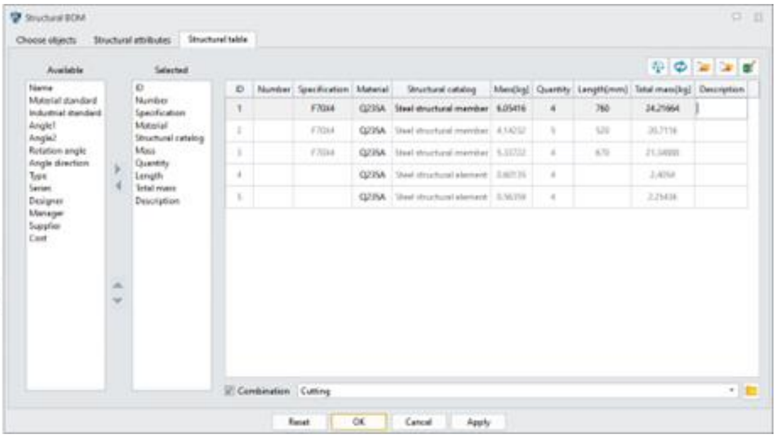
【What can be done】:

- ✓ When running the Structural BOM command, support displaying Designer, Manager, Supplier, and Cost.
- ✓ In the Structural table of Structural BOM, you can edit Description, Number, Designer, Manager, Supplier, and Cost online.

【Example】:

1. As shown below, double-click the Description field to edit the content.




Structural BOM
Choose objects
Structural attributes
Structural table
Available
Name
Material standard
Industrial standard
Angle1
Angle2
Rotation angle
Angle direction
Type
Series
Designer
Manager
Supplier
Cost
Selected
ID
Number
Specification
Material
Structural catalog
Mass
Quantity
Length
Total mass
Description
Reset
OK
Cancel
Apply

ID	Number	Specification	Material	Structural catalog	Mass(kg)	Quantity	Length(mm)	Total mass(kg)	Description
1		F7004	Q235A	Steel structural member	6.05416	4	760	24.21664	
2		F7004	Q235A	Steel structural member	[illegible]	5	520	[illegible]	
3		F7004	Q235A	Steel structural member	[illegible]	4	670	[illegible]	
4			Q235A	Steel structural element	[illegible]	4		[illegible]	
5			Q235A	Steel structural element	[illegible]	4		[illegible]	

☑ Combination: Cutting

【Where is it】:

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

4.2.5.2 ★Merge Rule Improvement

If Structural BOM is used for structural material procurement, it is necessary to combine rules by overall procurement. If Structural BOM is used for structural assembly or cutting before weldment, it is necessary to combine the rule of cutting. Besides, users can customize the combination rule of Structural BOM.

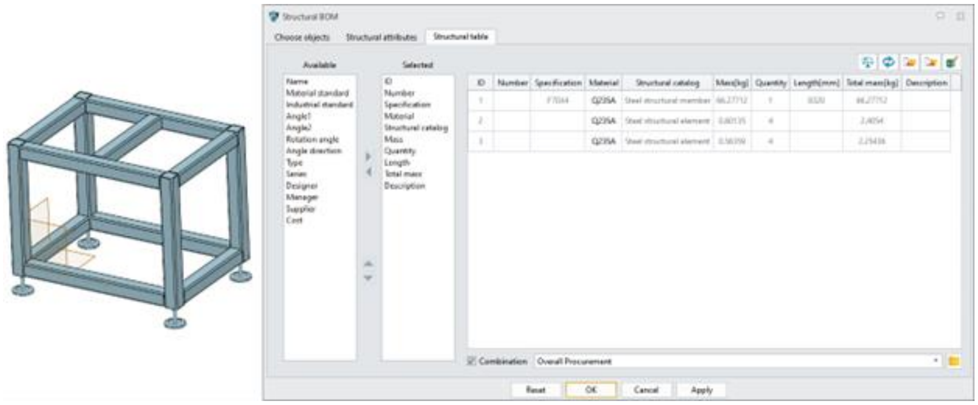
【What can be done】:

- ✓ ZW3D 2025 supports two combination rules: Overall Procumbent and Cutting.
- ✓ ZW3D 2025 supports user-defined combination rules in the combination library.

【Example】:

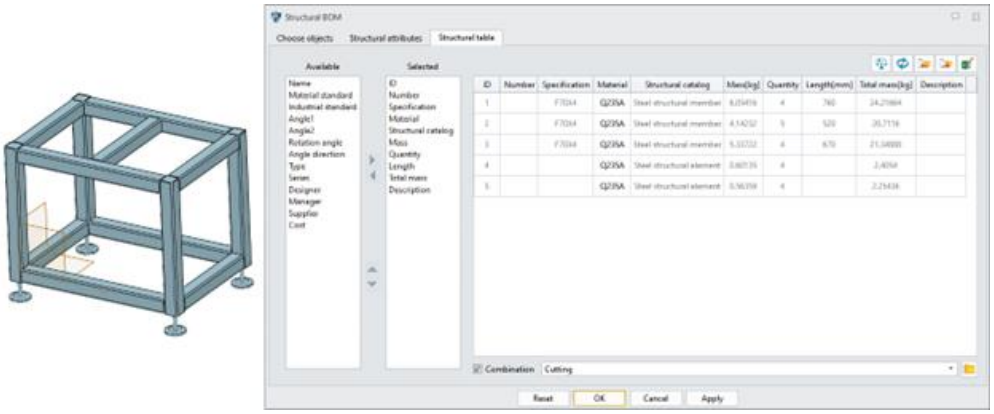
1. If users want to apply Structural BOM to structural material procurement, the combination rule

of Overall Procurement can be used.


Structural BOM
Choose objects
Structural attributes
Structural table
Available
Name
Material standard
Industrial standard
Angle1
Angle2
Rotation angle
Angle direction
Type
Series
Designer
Manager
Supplier
Cost
Selected
ID
Number
Specification
Material
Structural catalog
Mass
Quantity
Length
Total mass
Description
ID
Number
Specification
Material
Structural catalog
Mass(kg)
Quantity
Length(mm)
Total mass(kg)
Description
Reset
OK
Cancel
Apply

☑ Combination: Overall Procurement

2. If users want to apply Structural BOM to structural assembly or cutting before weldment, the combination rule of cutting can be used.


Structural BOM
Choose objects
Structural attributes
Structural table
Available
Name
Material standard
Industrial standard
Angle1
Angle2
Rotation angle
Angle direction
Type
Series
Designer
Manager
Supplier
Cost
Selected
ID
Number
Specification
Material
Structural catalog
Mass
Quantity
Length
Total mass
Description
ID
Number
Specification
Material
Structural catalog
Mass(kg)
Quantity
Length(mm)
Total mass(kg)
Description
Reset
OK
Cancel
Apply

☑ Combination: Cutting

【Where is it】:

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

4.2.5.3 ★Support Template Import & Export and Online Editing

ZW3D 2025 supports exporting and importing templates as well as editing templates in Excel.

【What can be done】:

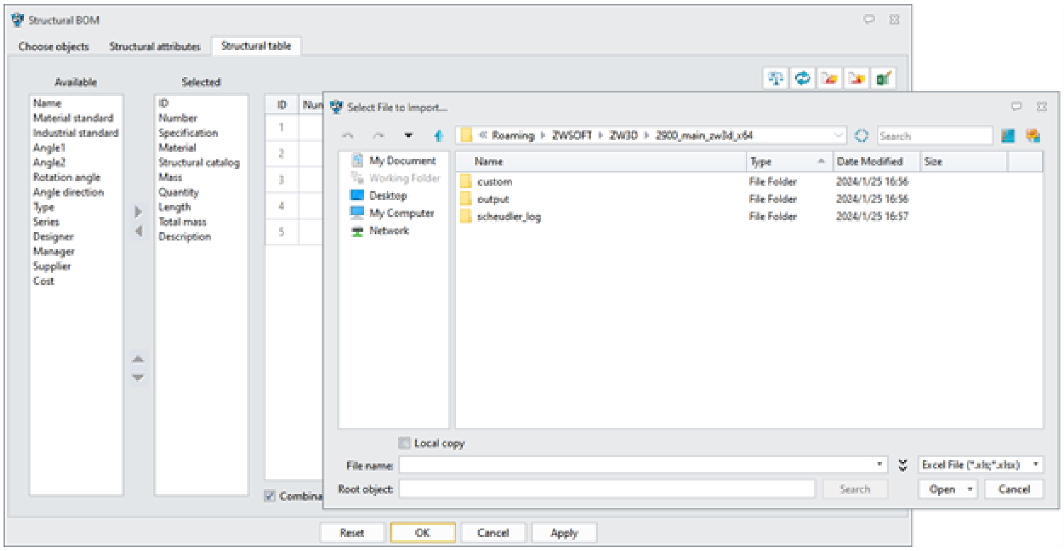
- ✓ Import in Structural BOM can make the template reusable.
- ✓ Edit in Excel in Structural BOM can edit formats and contents of BOM in Excel tables.

【Note】:

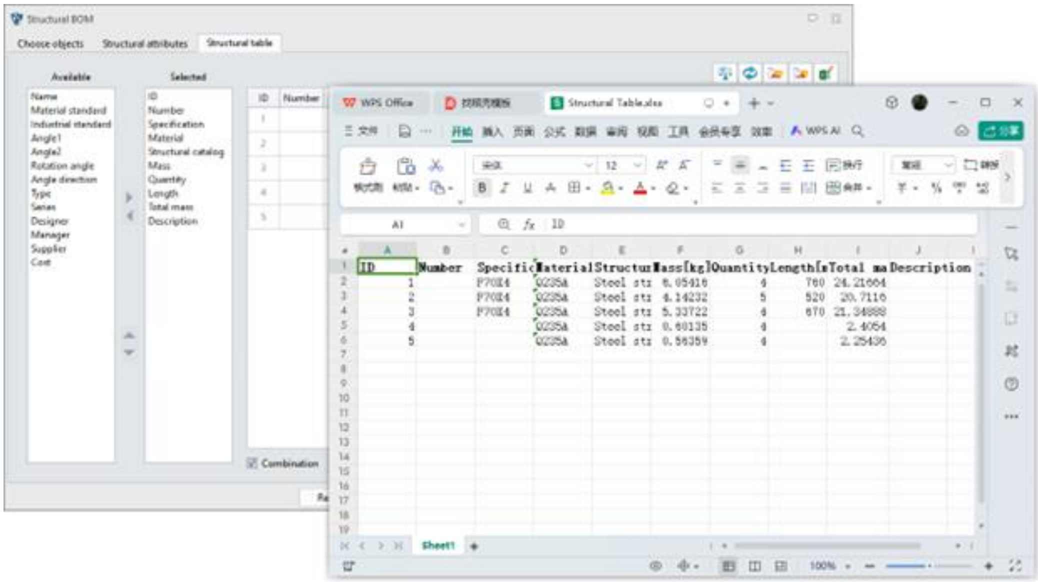
- Three format files: .xls, .xlsx, .csv can be imported.

【Example】：

1. Import in Structural BOM can make the template reusable.


Structural BOM
Choose objects
Structural attributes
Structural table
Available
Selected
Name
Material standard
Industrial standard
Angle1
Angle2
Rotation angle
Angle direction
Type
Series
Designer
Manager
Supplier
Cost
ID
Number
Specification
Material
Structural catalog
Mass
Quantity
Length
Total mass
Description
Combina
Reset
OK
Cancel
Apply
Select File to Import...
Roaming
ZWSOFT
ZW3D
2900_main_zw3d_x64
Search
My Document
Working Folder
Desktop
My Computer
Network
Name
Type
Date Modified
Size
custom
output
scheudler_log
File Folder
2024/1/25 16:56
2024/1/25 16:57
Local copy
File name:
Excel File (*.xls;*.xlsx)
Root object:
Open
Cancel

2. Edit in Excel in Structural BOM can edit formats and contents of BOM in Excel tables.


Structural BOM
Choose objects
Structural attributes
Structural table
Available
Selected
Combination
WPS Office
Structural Table.xlsx
A1
Sheet1
100%

ID	Number	Specific	Material	Structur	Mass[kg]	Quantity	Length[m	Total ma	Description
1		P70X4	Q235A	Steel str	6.05416	4	760	24.21664	
2		P70X4	Q235A	Steel str	4.14232	5	520	20.7116	
3		P70X4	Q235A	Steel str	5.33722	4	670	21.34888	
4			Q235A	Steel str	0.60135	4		2.4054	
5			Q235A	Steel str	0.56359	4		2.25436	

【Where is it】：

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

4.2.5.4 New Accuracy Mass Statistics Function

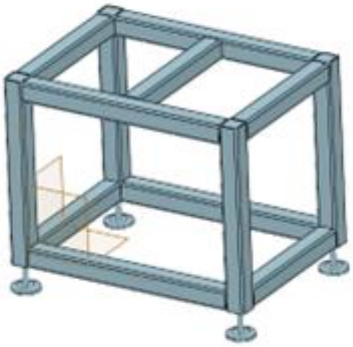
ZW3D 2025 has added Exact Mass Statistics to support user switching between cutting mass and exact mass to meet customers' needs in various scenarios.

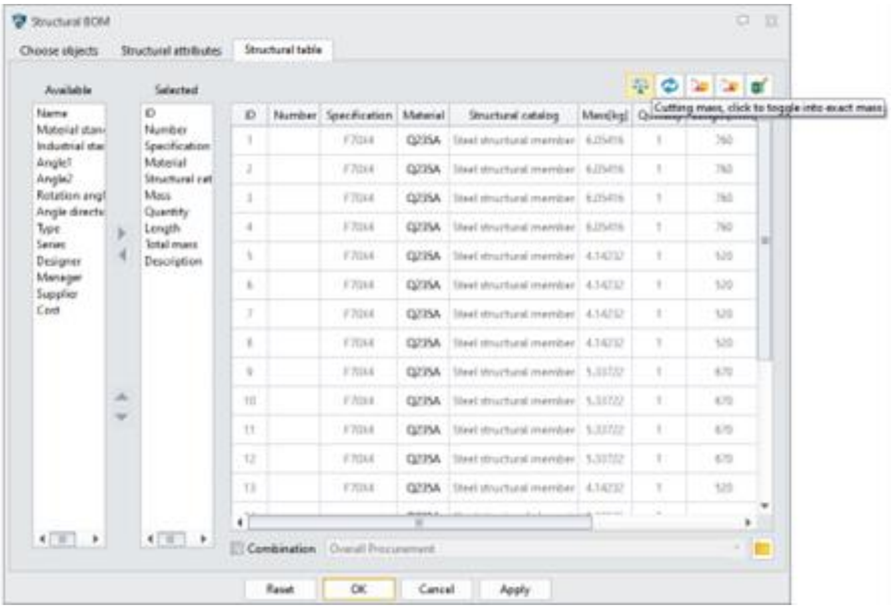
【What can be done】:

- ✓ Click the Mass button to switch between cutting mass and exact mass.

【Example】:

1. Click the Mass button to switch between cutting mass and exact mass in the following scenario.




Structural BOM
Choose objects
Structural attributes
Structural table
Available
Selected
Name
Material stan
Industrial sta
Angle1
Angle2
Rotation angl
Angle directi
Type
Series
Designer
Manager
Supplier
Cost
ID
Number
Specification
Material
Structural cat
Mass
Quantity
Length
Total mass
Description
ID
Number
Specification
Material
Structural catalog
Mass(kg)
Cutting mass, click to toggle into exact mass
Combination
Overall Procurement
Reset
OK
Cancel
Apply

【Where is it】:

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

4.2.6 Other

4.2.6.1 ★Structure Wizard Function

ZW3D 2025 has added Structure Wizard to guide and quickly create structural member standards.

【What can be done】:

- ✓ Create a new profile.

- ✓ Create based on the existing profiled.
- ✓ Create based on the existing sketches.
- ✓ Edit the existing profiles.

【Note】:

- The Structure Wizard command needs to be used with explanatory documents.

【Where is it】:

Assembly Environment >> Structure >> Library >> Structure Wizard

4.2.6.2 ★UI/UE of Structure Member Module Entire Upgrade

ZW3D 2025 has improved the UI and UE of the Structure Member module that lacked guidance in the 2024 version, which enhances the guidance for users.

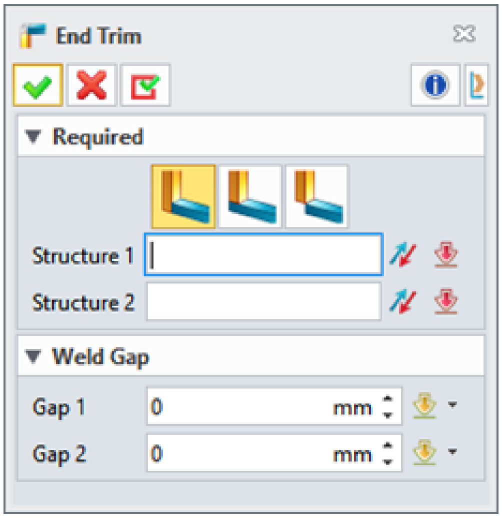
【What can be done】:

- ✓ Promote the guidance of Structure Member module design.

【Example】:

1. UI improvement in End Trim.

ZW3D 2024	ZW3D 2025



End Trim

Required

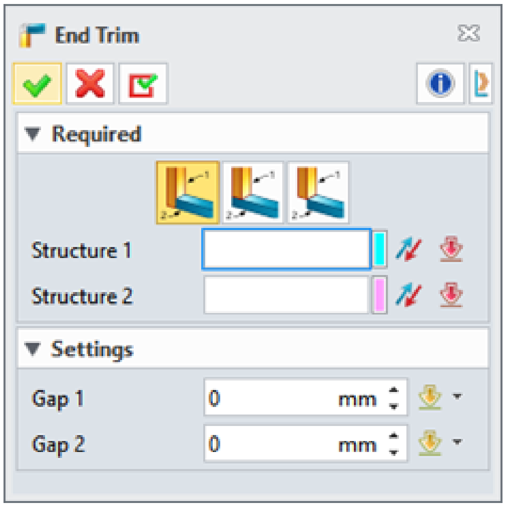
Structure 1

Structure 2

Weld Gap

Gap 1 0 mm

Gap 2 0 mm



End Trim

Required

Structure 1

Structure 2

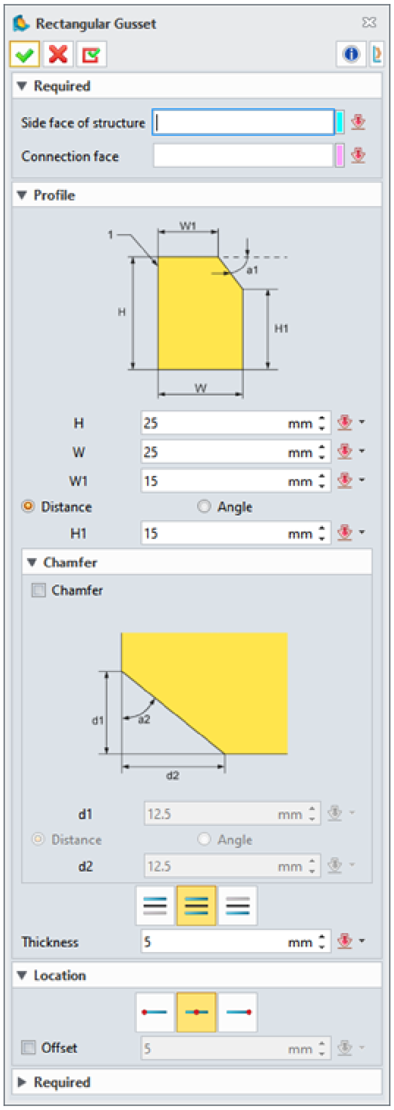
Settings

Gap 1 0 mm

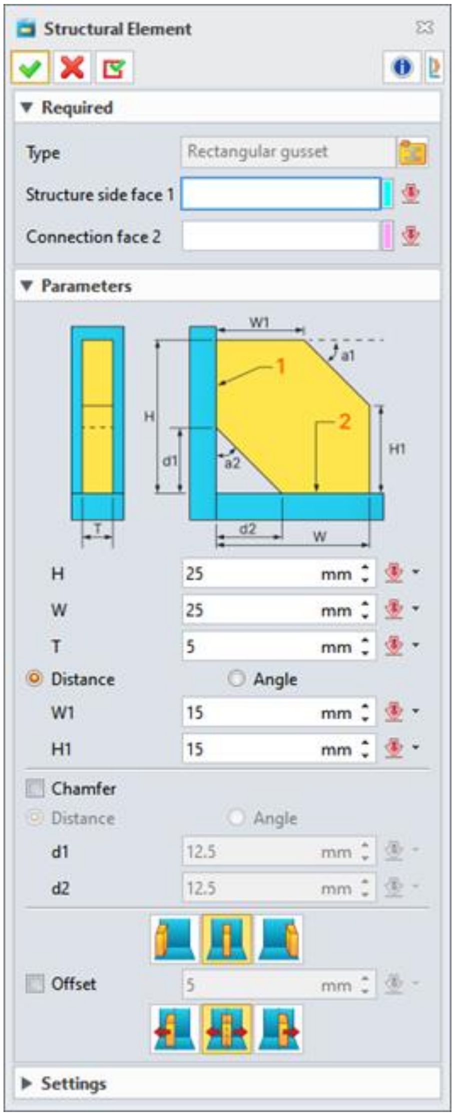
Gap 2 0 mm

2. UI improvement in Rectangular Gusset

ZW3D 2024	ZW3D 2025


Rectangular Gusset
Required
Profile
1
W1
a1
H
H1
W
Chamfer
d1
a2
d2
Location
Required

Side face of structure
Connection face
H 25 mm
W 25 mm
W1 15 mm
● Distance ○ Angle
H1 15 mm
☐ Chamfer
d1 12.5 mm
○ Distance ○ Angle
d2 12.5 mm
Thickness 5 mm
☐ Offset 5 mm


Structural Element
Required
Parameters
W1
a1
1
2
H
H1
d1
a2
T
d2
W
Settings

Type Rectangular gusset
Structure side face 1
Connection face 2
H 25 mm
W 25 mm
T 5 mm
● Distance ○ Angle
W1 15 mm
H1 15 mm
☐ Chamfer
○ Distance ○ Angle
d1 12.5 mm
d2 12.5 mm
☐ Offset 5 mm

【Where is it】：

Assembly Environment >> Structure

4.3 Mold Design

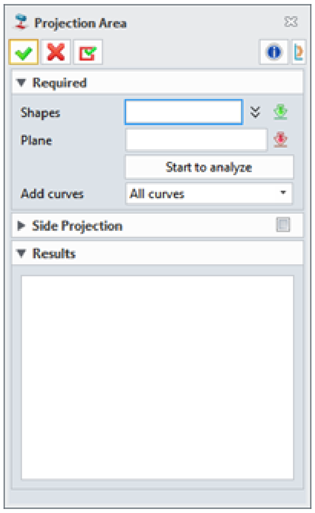
4.3.1 ★New Projection Area Function

In the design of the mold, it is necessary to select the model of the injection molding machine according to the projection area of the product in the mold opening direction. The Projection Area function in ZW3D 2025 can meet the design requirements of such scenarios.

【What can be done】：

ZW3D 2025 has added the Project Area command to support two projection methods: All curves and Exterior loops.

- ✓ All curves: Project the whole product to the projection area. In this case, the projection area corresponding to the product internal through hole will be subtracted from the calculation result.
- ✓ Exterior loops: Only the outer contour of the product will be projected and only the projected area of it will be calculated in this case. If only the outer contour of the product is projected, only the projected area of the outer contour of the product is calculated in this case.



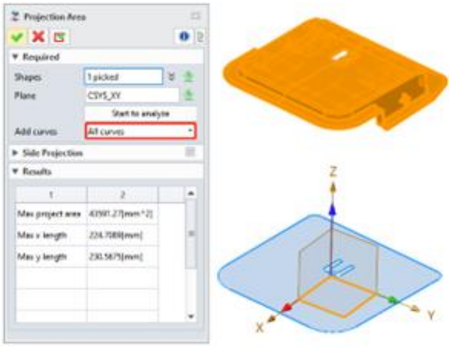
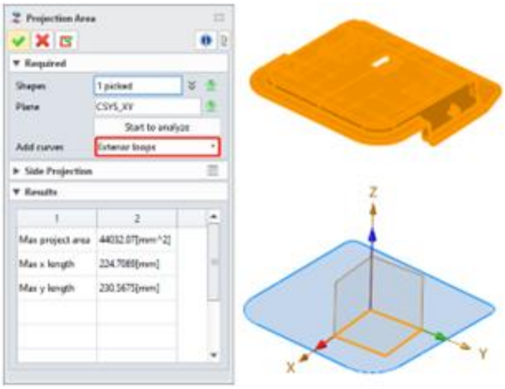
Add curves: All curves

【Note】：

- After the "Side projection" check box is checked, the software needs to perform multiple projection query calculations on the side direction of the projector plane, resulting in low efficiency of functional calculation.

【Example】：

All curves projection	Exterior loops projection

 Projection Area Required Shapes: 1 picked Plane: CSYS_XY Start to analyze Add curves: All curves Side Projection Results Max project area: 43591.27[mm^2] Max x length: 224.7089[mm] Max y length: 230.5675[mm]	 Projection Area Required Shapes: 1 picked Plane: CSYS_XY Start to analyze Add curves: Exterior loops Side Projection Results Max project area: 44032.07[mm^2] Max x length: 224.7089[mm] Max y length: 230.5675[mm]

【Where is it】:

Inquire >> Measure >> Projection Area

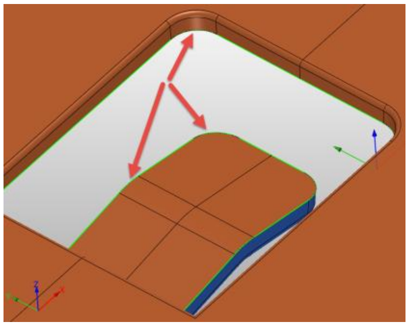
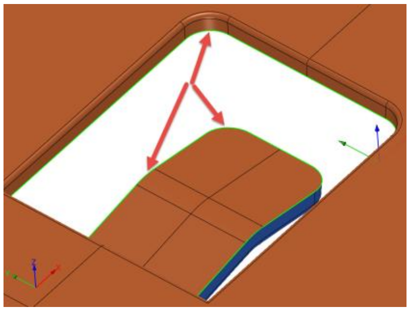
4.3.2 Parting Line Display Effect Optimization

In the mold design, the drawing of the parting line part can not only guide the parting design work of mold design engineers, but also help customers upstream to better understand the mold structure. Therefore, the display quality of the parting line greatly affects the scheme design report.

ZW3D 2025 enhances the display effect of the product parting line.

【What can be done】

✓ The display effect of parting line is enhanced.

ZW3D 2024	ZW3D 2025
	

【Where is it】:

Part/Assembly Environment >> Mold >> Parting Design >> Region Definition

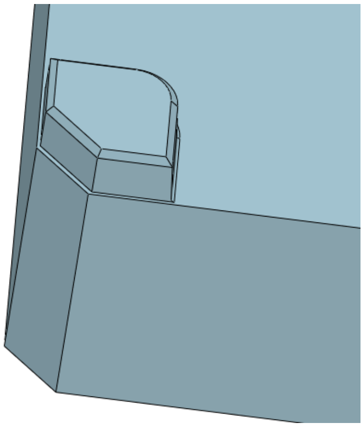
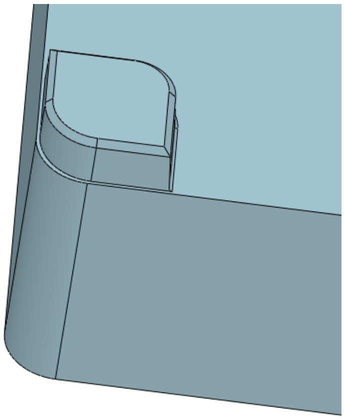
4.3.3 Taper Lock Function Improvement

When designing molds, the four sides of the mold core need to be chamfered, which requires the taper lock to adapt the chamfer (rounded corner) of the mold core when it is generated. The Taper Lock function of ZW3D 2025 can automatically identify the top chamfer (rounded corner) of the mold core and adapt the taper lock features for the chamfer to meet the design needs of such scenarios.

【What can be done】:

- ✓ The Taper Lock command can recognize the chamfer (fillet) on the mold core and adapt the chamfer (fillet) with the same size value for the taper lock features.

【Example】:

The Side Chamfer of the Mold Core	The Side Fillet of the Mold Core
	

【Where is it】:

Part/Environment >> Mold >> Detail Design >> Taper Lock

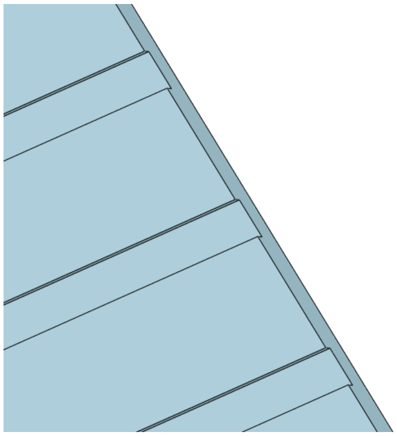
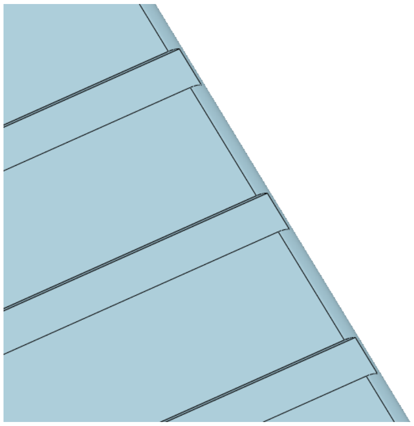
4.3.4 Airvent Function Improvement

In mold airvent design, it is often necessary to lead the airvent groove to the edge of the workpiece, and the edge of the workpiece often has a chamfer (fillet) feature, which requires that the airvent groove can pass through the chamfer (fillet) feature when it is generated. The airvent function of ZW3D 2025 can identify the edge chamfer (fillet) characteristics of the workpiece and make corresponding adaptations to meet the design needs of this type of scene.

【What can be done】:

- ✓ The Airvent command can identify the chamfer (fillet), and the generated airvent groove can pass through the corresponding chamfer (fillet) feature.

【Example】:

Side Chamfer	Side Fillet
	

【Where is it】:

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

4.3.5 Add Rename the Components to Trim Tool Function

When trimming a component (pin) with the Trim tool, users can check the newly added "Rename the components" option to generate the trimmed component (pin) with a new component name, by which components (pin) with the same name will not be trimmed at the same time.

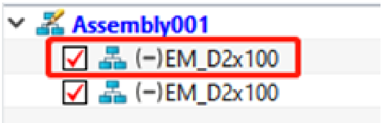
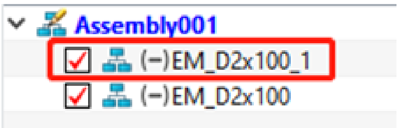
【What can be done】:

- ✓ When the "Rename the components" option is checked, the trimmed component (pin) will generate a new component name.

【Note】:

- When the “Rename the components” option is checked, a new component name with a suffix is automatically generated. The function does not support renaming components.

【Example】:

Pins before Being Trimmed	Pins after Being Trimmed
 Assembly001 ☑ (−)EM_D2x100 ☑ (−)EM_D2x100	 Assembly001 ☑ (−)EM_D2x100_1 ☑ (−)EM_D2x100

【Where is it】:

Part/Environment >> Mold >> Library >> Trim Tool

4.3.6 New Electrode Project Function

When designing the electrode, the workpiece properties need to be pre-set, such as the workpiece project name, part name, part version, and design information to facilitate the electrode list and spark drawing to secure the related property information. ZW3D 2025 has added the Electrode Project function to meet this design requirement.

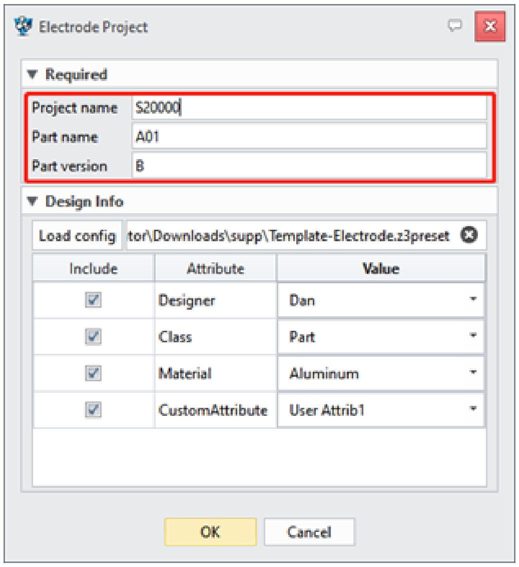
【What can be done】:

- ✓ In the Electrode Project function, users can set the project name, part name & version, design information, and customized information of the document that needs electrode design.

Electrode list and spark drawing can quickly read the corresponding attribute information.

【Example】：

1. Pre-set project name, part name, and part version in the Electrode Project function.


Electrode Project
Required
Design Info
OK
Cancel

Project name S20000

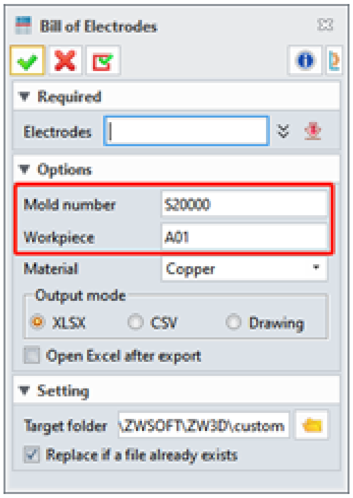
Part name A01

Part version B

Load config tor\Downloads\supp\Template-Electrode.z3preset

Include	Attribute	Value
☑	Designer	Dan
☑	Class	Part
☑	Material	Aluminum
☑	CustomAttribute	User Attrib1

2. Electrode list and spark drawing can quickly get the corresponding information.


Bill of Electrodes
Required
Electrodes
Options
Setting

Mold number S20000

Workpiece A01

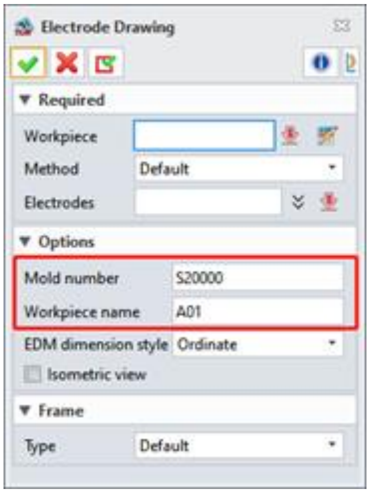
Material Copper

Output mode: ● XLSX ○ CSV ○ Drawing

☐ Open Excel after export

Target folder \ZWSOFT\ZW3D\custom

☑ Replace if a file already exists


Electrode Drawing
Required
Workpiece
Electrodes
Options
Frame

Method Default

Mold number S20000

Workpiece name A01

EDM dimension style Ordinate

☐ Isometric view

Type Default

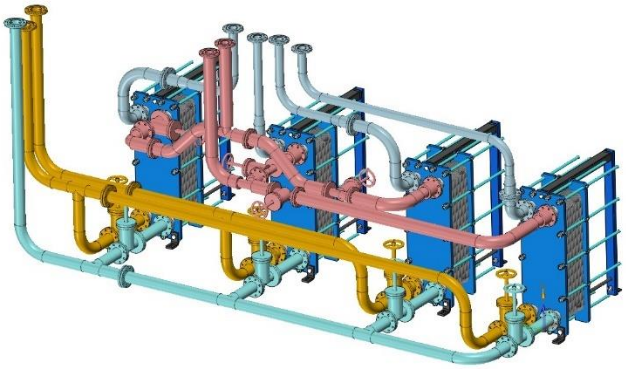
【Where is it】：

Part/Assembly Environment >> Electrode >> Start >> Electrode Project

4.4 Piping & Tubing Design

4.4.1 Pipe Modeling Enhancement

The 3D Modeling has added new types of pipe wall fittings and optimized pipeline parameters, inserted pipe standard fittings, breakpoints, and rapid sizing functions. The added functionality enhances pipeline modeling capabilities and expands supported design scenarios. The optimized functions focus on improving the design experience and fully solving the modeling efficiency.

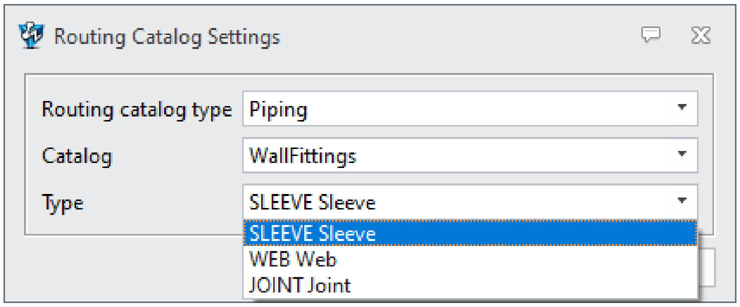


4.4.1.1 New WallFittings Type

n the **Routing Design** process, it is usually necessary to add protective casing, inner pipe clamp, web, pipe seat, etc. Such standard parts will not cause pipe disconnection. Users can define the "WallFittings" standard type to add standard parts to the pipe.

【What can be done】:

- ✓ Set the type of pipe wall fittings and complete the library definition of pipe wall fittings which can be used to add pipe wall fittings in the pipe.



Routing catalog type: Piping

Catalog: WallFittings

Type: SLEEVE Sleeve

【Note】：

- Currently, the pipe wall fittings cannot perforate the pipe.

【Example】：

1. The library definition of routing standard parts, such as thermometer joint base, pressure gauge joint base, and other pipe bases, can be set to the “joint”.
2. The library definition of routing standard parts, such as protection bushing, inner pipe clamp, and other protection classes, can be set to the “Sleeve”.
3. The library definition of routing standard parts, such as web, seals, and other non-standard classes, can be set to the “Web”.

【Where is it】：

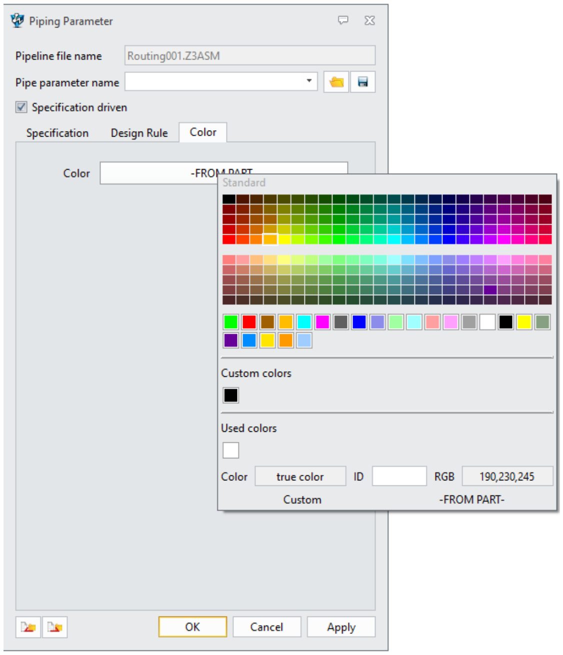
Part >> Insert >> Routing >> Set Routing Catalog

4.4.1.2 New Pipe Color Presetting

In the routing design process, users usually want to arrange the routes according to their actual colors. ZW3D 2025 has added the “**Color**” option to control the newly created pipes to follow the set color. If you don't need to set the color, change it to "FROM PART" or "Reset to default".

【Where can be done】:

- ✓ If a color is set, the newly arranged pipes will display the corresponding color.
- ✓ If no color is set (FROM PART by default), the newly arranged pipes will show the color saved in the library file.


Piping Parameter
Specification
Design Rule
Color
Standard
Custom colors
Used colors
Custom
OK
Cancel
Apply

Pipeline file name: Routing001.Z3ASM

Pipe parameter name:

☑ Specification driven

Color: -FROM PART-

Color: true color ID: RGB: 190,230,245

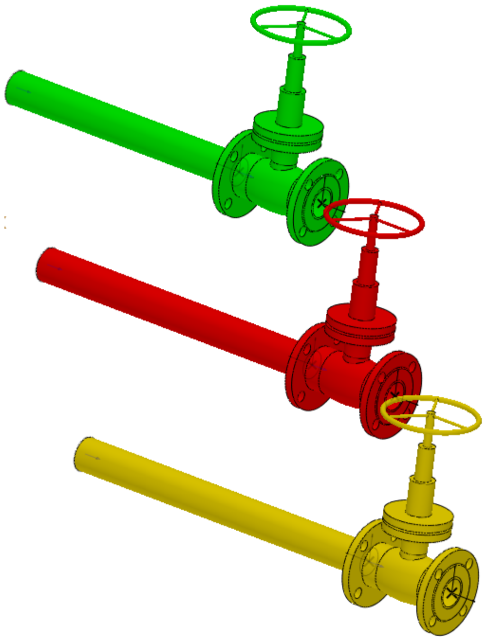
-FROM PART-

【Note】:

- The color setting only takes effect on the newly built pipes and their standard parts. The existing pipe remains unchanged, and the modified pipes remain their original color.

【Example】:

1. Before designing fire extinguishing pipes, use the "**Color**" option and set the color to red. The arranged pipes will be in red color.
2. Before designing yellow painted pipes, use the "**Color**" option and set the color to yellow. The arranged pipes will be in yellow color.



【**Where is it**】：

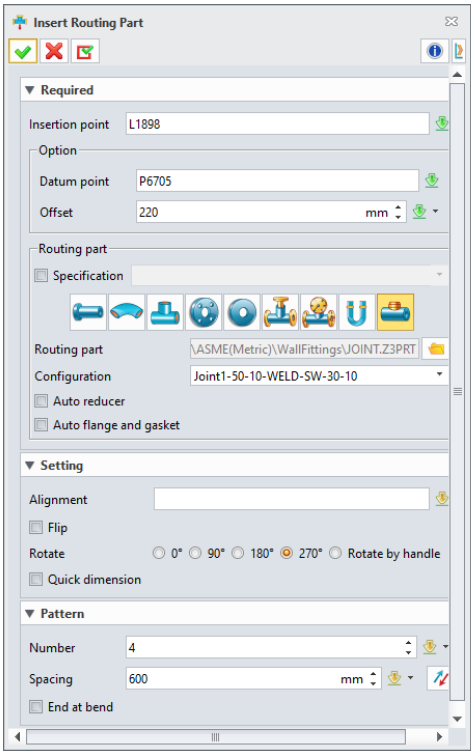
Piping/Tubing >> Piping Rule >> Piping Parameter >> Color

4.4.1.3 ★Insert Routing Part Optimization

In the routing design process, the optimized "Insert Routing Part" function can be used to add various types of routing standard parts, including protective sleeves, inner clamps, and thermometer seats. It supports precise positioning to add locations, realizing dimension addition. With the Pattern option, standard parts can be added quickly and precisely, greatly improving the routing design efficiency.

【**What can be done**】：

- ✓ Support to add protective sleeves inner clamp, instrument base, and other wall fittings, the pipe to maintain a continuous state.
- ✓ Adding pipe fittings based on inflection point and special point offset distance.
- ✓ Add pipe fittings are automatically dimensioned.
- ✓ Standard parts can be added along the routing pattern.



Insert Routing Part

Required

Insertion point L1898

Option

Datum point P6705

Offset 220 mm

Routing part

☐ Specification

Routing part \ASME(Metric)\WallFittings\JOINT.Z3PRT

Configuration Joint1-50-10-WELD-SW-30-10

☐ Auto reducer

☐ Auto flange and gasket

Setting

Alignment

☐ Flip

Rotate ○ 0° ○ 90° ○ 180° ● 270° ○ Rotate by handle

☐ Quick dimension

Pattern

Number 4

Spacing 600 mm

☐ End at bend

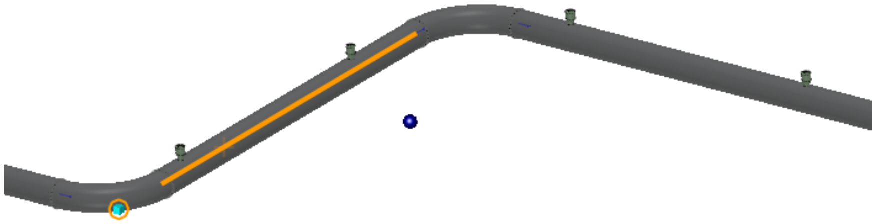
【Note】:

- The actual position of the added pipe standard components are determined by the offset distance.
- The position of the array must be on the straight section of the pipe. The bending position or the section beyond the pipe is invalid.

【Example】:

1. In the routing design, **Optional** in the I**nsert Routing Part** command allows you to set the datum point and offset distance. The pipe fittings can be added at a certain distance from the inflection point or special point.
2. In the double-wall pipe design, " **WallFittings**" in the "**Insert Routing Part**" command allows you to add an inner pipe clamp. After the pipe fittings are added, the pipe remains continuous and can be defined to spools to output spool drawings.
3. In the routing design, "**Quick dimension**" in the "**Insert Routing Part**" command allows dimensions to be automatically marked when adding standard parts.

4. n the double-wall pipe design, the "**Insert Routing Part**" command "**Pattern**" option allows you to add pipe standard parts in batches along the pipe direction.



【Where is it】：

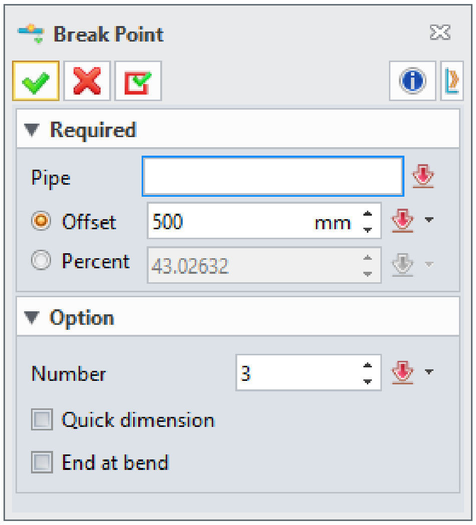
Piping/Tubing >> Route >> Insert Routing Part

4.4.1.4 Break Point Optimization

In the routing design process, the optimized "**Break Point**" command can add break points in the middle of pipes and dimensions. By arraying break points, users can accurately add the break point location, thus improving the efficiency of routing design.

【What can be done】：

- ✓ Automatically judge the start point.
- ✓ Break pipes in batches along the line direction.
- ✓ Add dimensions to the break point location.



Break Point

Required

Pipe

● Offset 500 mm

○ Percent 43.02632

Option

Number 3

☐ Quick dimension

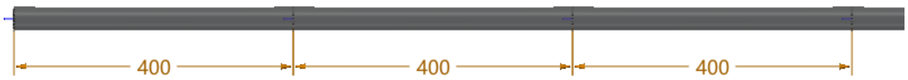
☐ End at bend

【Note】：

- The break points of flexible pipe are based on the length of the spline center line.

【Example】：

1. In routing design, you can automatically determine the datum point by using the "**Pipe**" option of the "**Break Point**" command. The connection point near the selected pipe center line is the reference point.
2. In routing design, you can batch break points to standard pipe length according to the specified length by using the "**Pattern**" option of the "**Break Point**" command to make full use of the length of the pipe. And when the Quick Dimension is on, the fixed length size can be marked.



【Where is it】：

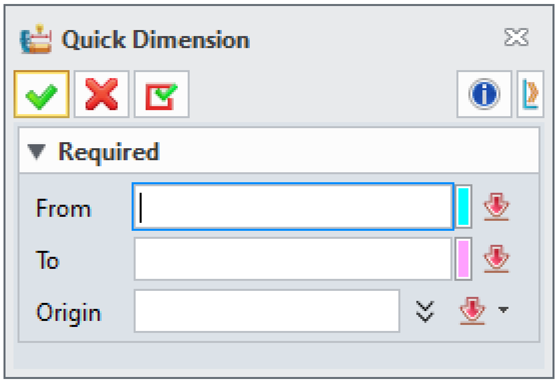
Piping/Tubing >> Route >> Break Point

4.4.1.5 Quick Dimension Optimization

In the routing design process, users often need to label the integer dimensions to precisely control the position. By the optimized “**Quick Dimension**” function, users can directly input the target length or angle for dimension without double-clicking again to edit and enter.

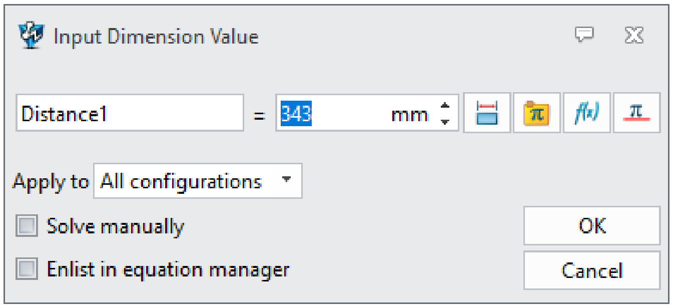
【What can be done】:

- ✓ After marking the size, you can enter the target size into the pop-up window and continue to dimension the size.



【Example】:

1. The dimensioned pipe length value is not the target value, which needs to be modified to an integer or a target value.



Distance1 = 343 mm

Apply to All configurations

☐ Solve manually

☐ Enlist in equation manager

【Where is it】:

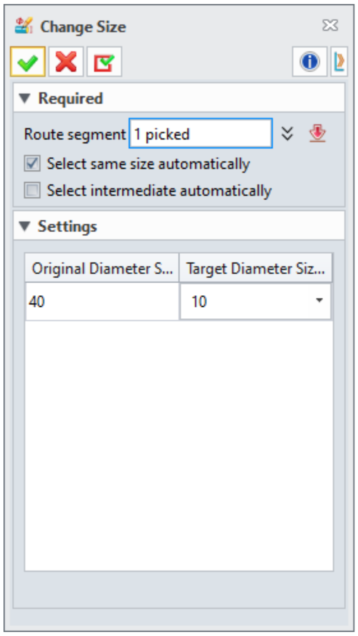
Piping/Tubing >> Constraint >> Quick Dimension

4.4.1.6 Change Size Optimization (ZW3D 2025 SP)

During the pipeline design process, you need to adjust the size of the pipe to modify the pipeline. Through the optimized "Change size" function, the "Select same size automatically" can automatically select the part of the same specification. It reduces the selection and increase design efficiency.

【What can be done】:

- ✓ Automatic select the fittings of the same size, including reducer.
- ✓ Automatic select the fittings of the same size, including branch.



Required

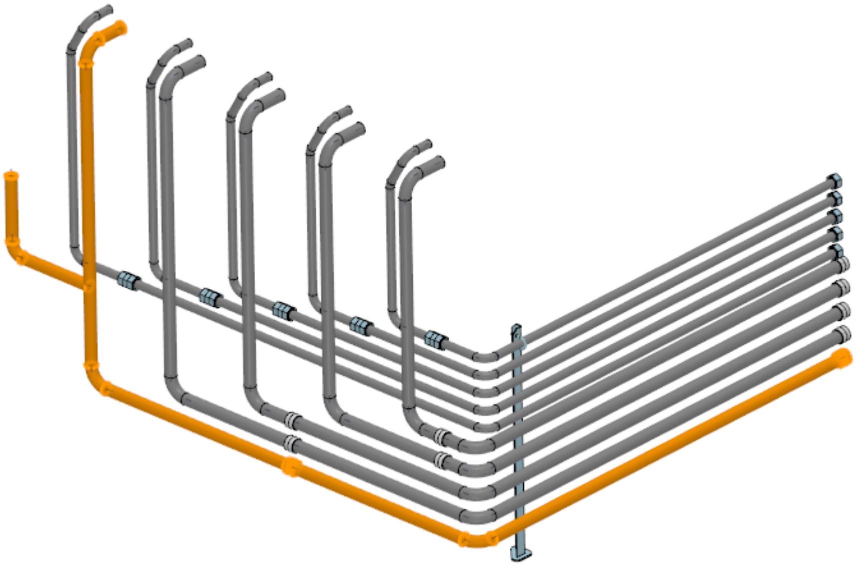
Route segment: 1 picked

☑ Select same size automatically

☐ Select intermediate automatically

Settings

Original Diameter S...	Target Diameter Siz...
40	10



【Note】:

- The part automatic selected must be connected continuously.

【Example】:

1. When designing the pipeline, it needs to change the design. it is necessary to change the size of a pipe fitting in the pipeline, and the size of the same size connected with it also needs to be changed, so that the pipeline is connected correctly.

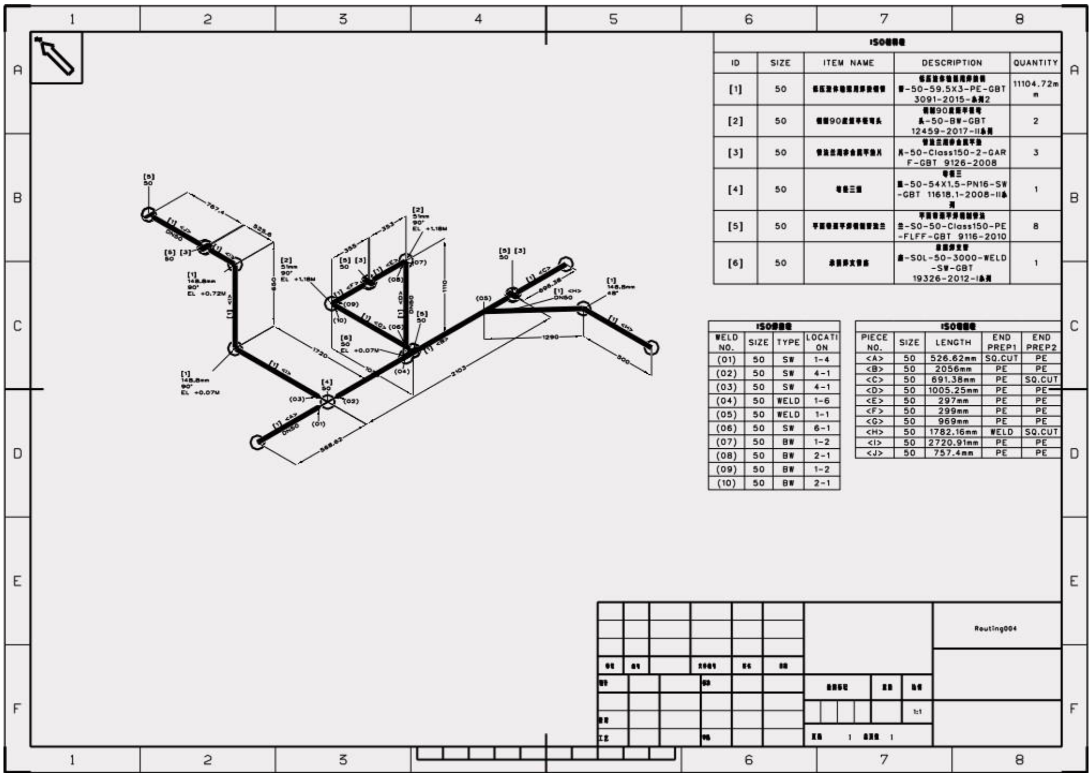
2. When designing the pipeline, it needs to change the design.it is necessary to change the size of the local pipeline, and the same size is automatically selected, so that the pipeline is connected correctly.

【Where is it】:

Piping/Tubing >> Modify >> Change size

4.4.2 Routing Drawing Optimization

The optimized drawing functions of the Routing module include spool drawing and ISO drawing. The optimized spool drawing focuses on improving the rationality check. The optimized ISO drawing mainly enhances automatic subdrawing and diversified labeling capabilities to meet the customized needs of different customers.



ISO[illegible]

ID	SIZE	ITEM NAME	DESCRIPTION	QUANTITY
[1]	50	[illegible]	[illegible]-50-59.5X3-PE-GBT 3091-2015-[illegible]2	11104.72mm
[2]	50	[illegible]90[illegible]	[illegible]90[illegible]-50-BW-GBT 12459-2017-II[illegible]	2
[3]	50	[illegible]	[illegible]-50-Class150-2-GARF-GBT 9126-2008	3
[4]	50	[illegible]	[illegible]-50-54X1.5-PN16-SW-GBT 11618.1-2008-II[illegible]	1
[5]	50	[illegible]	[illegible]-SO-50-Class150-PE-FLFF-GBT 9116-2010	8
[6]	50	[illegible]	[illegible]-SOL-50-3000-WELD-SW-GBT 19326-2012-I[illegible]	1

ISO[illegible]

WELD NO.	SIZE	TYPE	LOCATION
(01)	50	SW	1-4
(02)	50	SW	4-1
(03)	50	SW	4-1
(04)	50	WELD	1-6
(05)	50	WELD	1-1
(06)	50	SW	6-1
(07)	50	BW	1-2
(08)	50	BW	2-1
(09)	50	BW	1-2
(10)	50	BW	2-1

ISO[illegible]

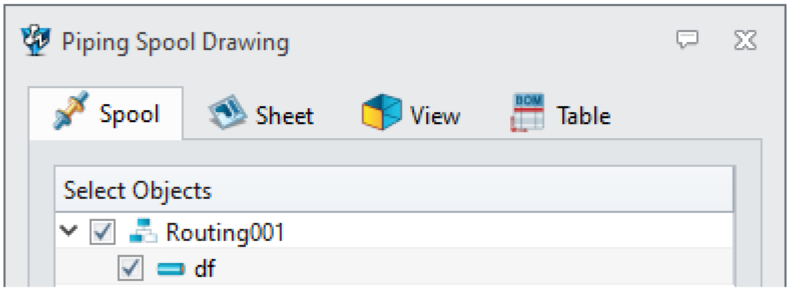
PIECE NO.	SIZE	LENGTH	END PREP1	END PREP2
<A>	50	526.62mm	SQ.CUT	PE
<B>	50	2056mm	PE	PE
<C>	50	691.38mm	PE	SQ.CUT
<D>	50	1005.25mm	PE	PE
<E>	50	297mm	PE	PE
<F>	50	299mm	PE	PE
<G>	50	969mm	PE	PE
<H>	50	1782.16mm	WELD	SQ.CUT
<I>	50	2720.91mm	PE	PE
<J>	50	757.4mm	PE	PE

4.4.2.1 ★Spool Drawing Optimization

After routing design is completed, users need to complete the output of the prefabricated spool drawings according to the manufacturing process requirements. When a spool drawing is generated through the optimized "**Spool Drawing"** function, the Spool Selection List will automatically determine whether the spool definition is reasonable to avoid abnormal spool definition due to the wrong definition or model deletion or modification. By automatically checking the reasonableness of the spool, users can avoid production errors due to wrong drawing guidance.

【What can be done】:

- ✓ Black spool names indicate the reasonable spool definition.
- ✓ Blue spool names indicate spool definition with the possible abnormal conditions.
- ✓ Red spool names indicate the wrong spool definition.



【Note】:

- The definition of the Spool command does not have a check mechanism, and thus the rationality of spools cannot be checked.
- Blue spool names do not generate spool drawings by default, but users can choose to generate spool drawings; Red spool names have no selection to generate spool drawings, which requires users to modify the spool definition.
- Both Routing BOM (spool only) and Bending Data have the spool rationality check mechanism.

【Example】:

1. When a spool is defined, the bent pipe definition does not include bending, and the spool will be displayed in red.
2. When a spool is defined, it only includes standard parts, and the spool will be shown in blue.

【Where is it】:

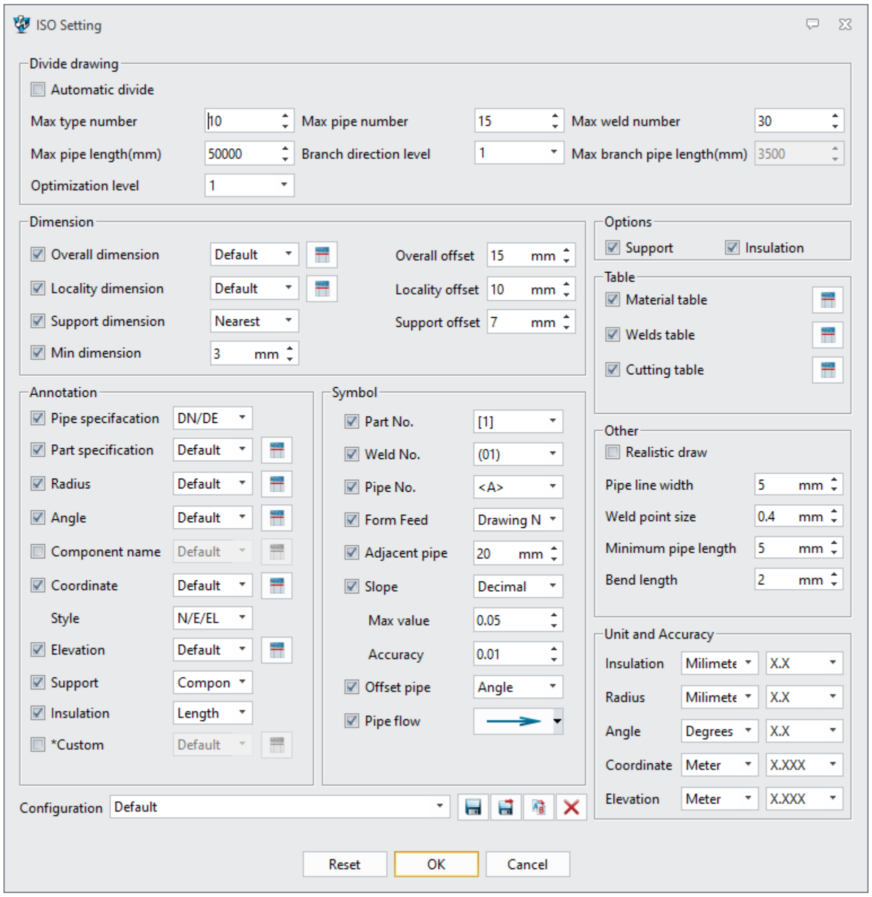
Piping/Tubing >> Drawing >> Spool Drawing

4.4.2.2 ★ISO Setting Optimization

After the 3D design of routing, various styles of ISO drawings need to be generated. The optimized "ISO settings" allow users to set boundaries for Automatic divide, automatically dimension and position, automatically annotate attributes and positions, and form the general configuration. By controlling the configurations, different styles of ISO drawings can be generated to output customized ISO drawings.

【What can be done】:

- ✓ Set the boundary conditions for enabling Automatic divide.
- ✓ Set the content and position of dimensioning and dimension by length and angle dimensions.
- ✓ Set the content and location of annotation and annotate by the leader annotation.
- ✓ Set the style annotation of the symbol, including BOM number styles, flow directions, and offset tube symbols.
- ✓ Set the unit and precision of the dimensioning.
- ✓ Different settings records can be saved and shared.



ISO Setting

Divide drawing
☐ Automatic divide
Max type number: 10
Max pipe number: 15
Max weld number: 30
Max pipe length(mm): 50000
Branch direction level: 1
Max branch pipe length(mm): 3500
Optimization level: 1

Dimension
☑ Overall dimension: Default
☑ Locality dimension: Default
☑ Support dimension: Nearest
☑ Min dimension: 3 mm
Overall offset: 15 mm
Locality offset: 10 mm
Support offset: 7 mm

Options
☑ Support ☑ Insulation

Table
☑ Material table
☑ Welds table
☑ Cutting table

Annotation
☑ Pipe specifacation: DN/DE
☑ Part specification: Default
☑ Radius: Default
☑ Angle: Default
☐ Component name: Default
☑ Coordinate: Default
Style: N/E/EL
☑ Elevation: Default
☑ Support: Compon
☑ Insulation: Length
☐ *Custom: Default

Symbol
☑ Part No.: [1]
☑ Weld No.: (01)
☑ Pipe No.: <A>
☑ Form Feed: Drawing N
☑ Adjacent pipe: 20 mm
☑ Slope: Decimal
Max value: 0.05
Accuracy: 0.01
☑ Offset pipe: Angle
☑ Pipe flow: →

Other
☐ Realistic draw
Pipe line width: 5 mm
Weld point size: 0.4 mm
Minimum pipe length: 5 mm
Bend length: 2 mm

Unit and Accuracy
Insulation: Milimete X.X
Radius: Milimete X.X
Angle: Degrees X.X
Coordinate: Meter X.XXX
Elevation: Meter X.XXX

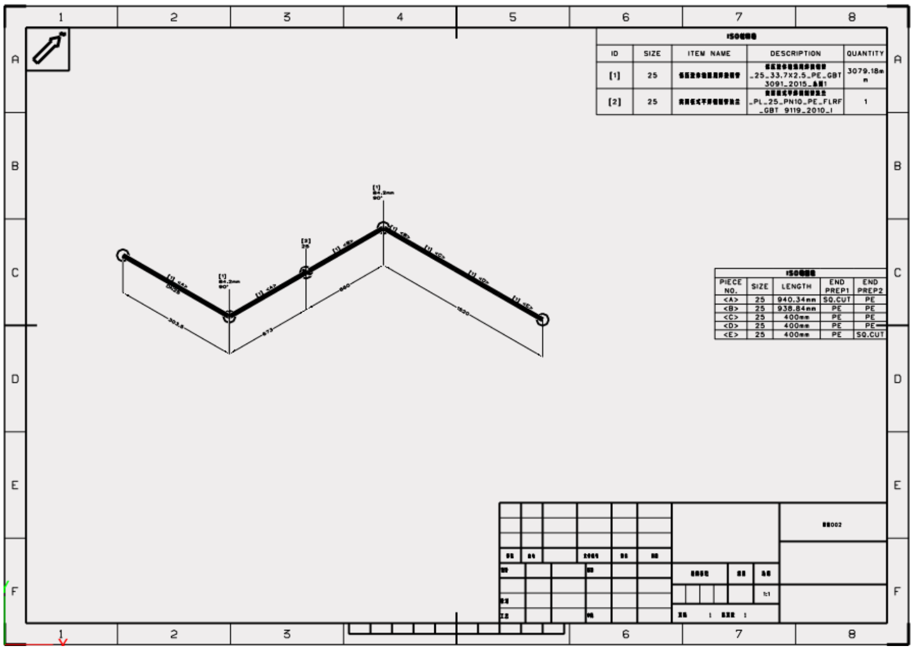
Configuration: Default

【Note】:

- Default is an editable text that cannot be translated. Users can change it to the corresponding language or abbreviation as required.
- Automatic divide is only the default option, and whether to enter Automatic divide is determined by Automatic divide of "ISO Generate".

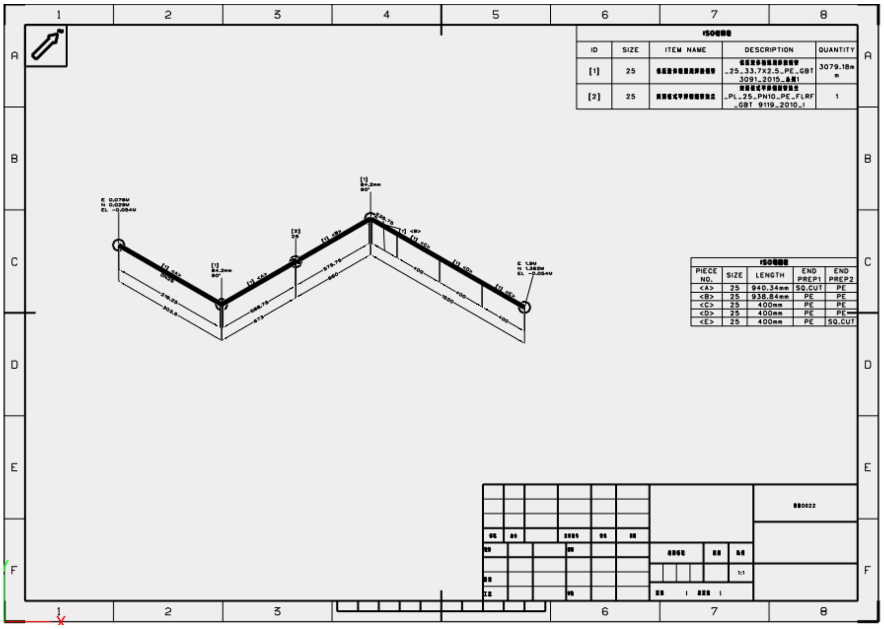
【Example】:

1. When generating an ISO drawing, you can change the result of the automatic dividing by using the boundary of the "**Divide** " option to set reasonable dividing boundary conditions according to the size and layout of the drawing template and the density of the standard parts of the actual routing model.
2. When generating an ISO drawing, you will need to annotate the core size information according to the purpose of the drawing. The fewer dimensions, the clearer the layout expression. The more dimensions, the more accurate the on-site production expression. You can change the dimensioning effect of ISO drawings by using the Settings of the "**Dimension**" option to set a reasonable annotation location.
3. When generating an ISO drawing, the core information that needs to be annotated varies depending on the purpose of the drawing. The fewer annotations, the clearer the layout expression. The more annotations, the more accurate the on-site production expression. You can change the leader annotation effect of ISO drawings by using the Settings of the "**Annotation**" option to set a reasonable annotation content and location.
4. When generating ISO drawings, users can set different symbols according to the usage habits of drawings to improve the ability of drawing information expression. You can maintain the usage habit requirements of ISO drawings by using the Settings of the "**Symbol**" option to set a reasonable style.



ID	SIZE	ITEM NAME	DESCRIPTION	QUANTITY
[1]	25	[illegible]	[illegible]_25_33.7X2.5_PE_GBT 3091_2015_[illegible]1	3079.18mm
[2]	25	[illegible]	[illegible]_PL_25_PN10_PE_FLRF_GBT 9119_2010_I	1

PIECE NO.	SIZE	LENGTH	END PREP1	END PREP2
<A>	25	940.34mm	SQ.CUT	PE
<B>	25	938.84mm	PE	PE
<C>	25	400mm	PE	PE
<D>	25	400mm	PE	PE
<E>	25	400mm	PE	SQ.CUT



ID	SIZE	ITEM NAME	DESCRIPTION	QUANTITY
[1]	25	[illegible]	[illegible]_25_33.7X2.5_PE_GBT 3091_2015_[illegible]1	3079.18mm
[2]	25	[illegible]	[illegible]_PL_25_PN10_PE_FLRF_GBT 9119_2010_I	1

PIECE NO.	SIZE	LENGTH	END PREP1	END PREP2
<A>	25	940.34mm	SQ.CUT	PE
<B>	25	938.84mm	PE	PE
<C>	25	400mm	PE	PE
<D>	25	400mm	PE	PE
<E>	25	400mm	PE	SQ.CUT

【Where is it】:

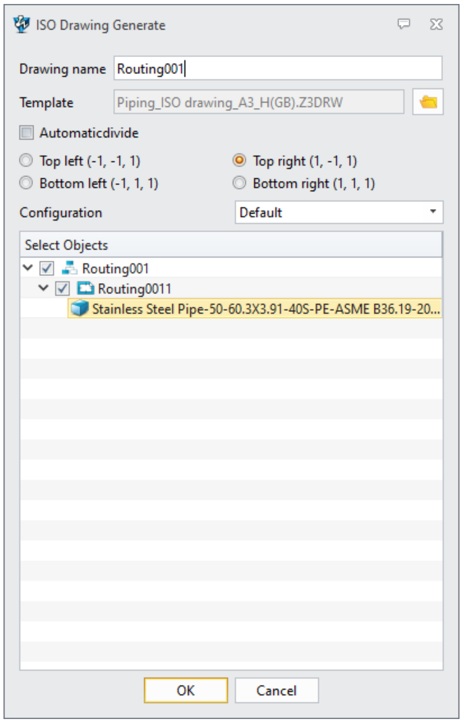
Piping/Tubing >> Drawing >> ISO Setting

4.4.2.3 ISO Generation Optimization

After the 3D design of routing, various styles of ISO drawings need to be generated. The optimized “ISO settings” allow users to generate ISO drawings with different styles, including drawing name, template, Automatic dividing, and configuration. By generating various ISO drawings, it is easy for users to compare the effects of the drawings.

【What can be done】:

- ✓ Set drawing names.
- ✓ Select the target template.
- ✓ Whether to turn on the Automatic divide option.
- ✓ Select one of the four views.
- ✓ Select the annotated general configuration.



ISO Drawing Generate

Drawing name: Routing001

Template: Piping_ISO drawing_A3_H(GB).Z3DRW

☐ Automaticdivide

○ Top left (-1, -1, 1)　● Top right (1, -1, 1)

○ Bottom left (-1, 1, 1)　○ Bottom right (1, 1, 1)

Configuration: Default

Select Objects

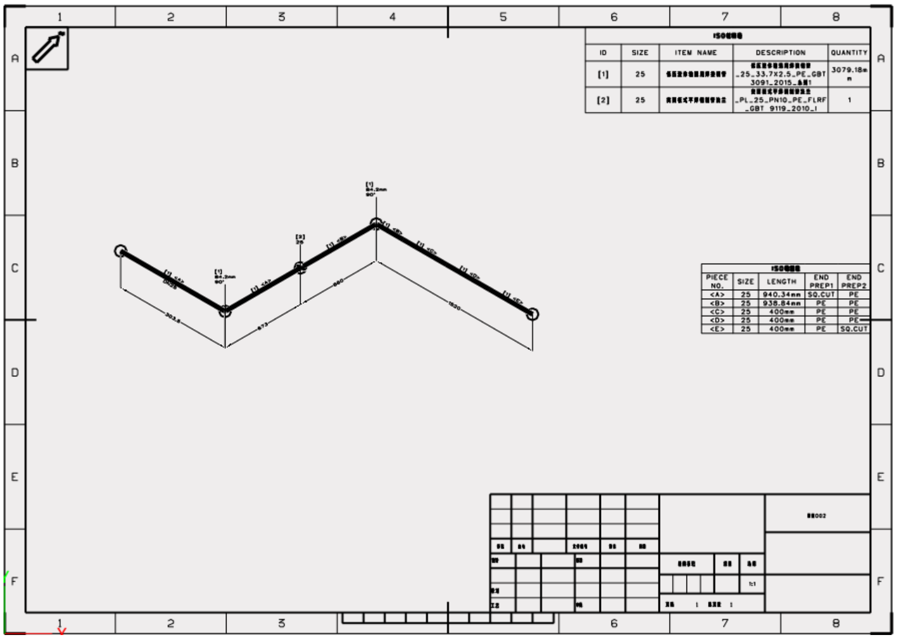
- ☑ Routing001
  - ☑ Routing0011
    - Stainless Steel Pipe-50-60.3X3.91-40S-PE-ASME B36.19-20...

【Note】:

- Piping standard parts must be correctly defined.
- Closed loop pipes cannot generate ISO drawings.

【Example】:

1. When the model has a certain scale, multiple ISO drawings need to be generated. You can use the "**Automatic divide**" option to divide complex routing models into multiple ISO drawings.
2. You need to audit, product, install, maintain, and repair the same model according to the ISO drawing, which requires to generate drawings with varied dimension styles. You can use the "**Configuration**" option of "ISO Setting" to generate different styles of ISO drawings.

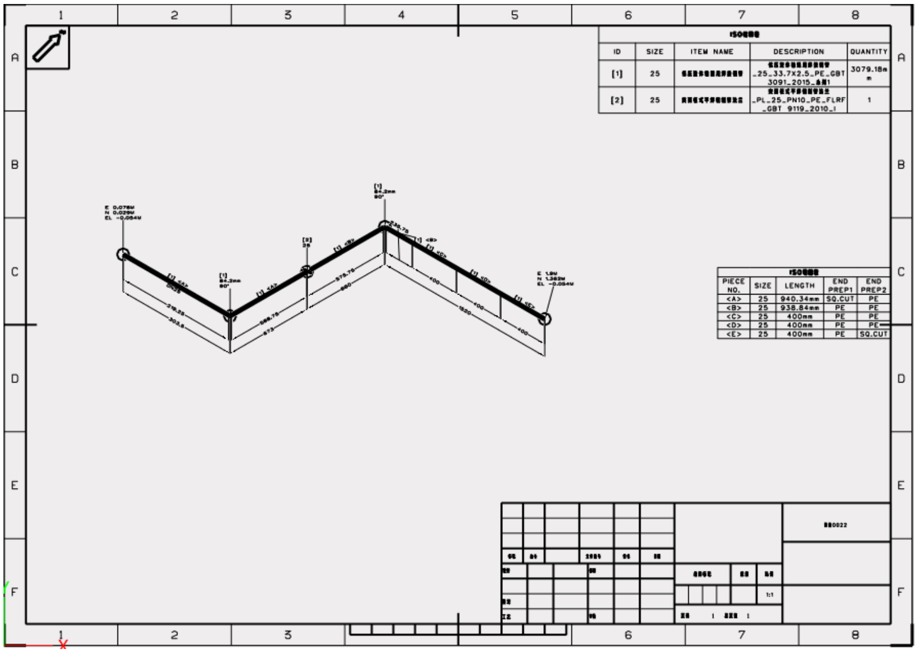


ISO[illegible]

ID	SIZE	ITEM NAME	DESCRIPTION	QUANTITY
[1]	25	[illegible]	[illegible]_25_33.7X2.5_PE_GBT 3091_2015_[illegible]1	3079.18mm
[2]	25	[illegible]	[illegible]_PL_25_PN10_PE_FLRF_GBT 9119_2010_I	1

ISO[illegible]

PIECE NO.	SIZE	LENGTH	END PREP1	END PREP2
<A>	25	940.34mm	SQ.CUT	PE
<B>	25	938.84mm	PE	PE
<C>	25	400mm	PE	PE
<D>	25	400mm	PE	PE
<E>	25	400mm	PE	SQ.CUT



ISO[illegible]

ID	SIZE	ITEM NAME	DESCRIPTION	QUANTITY
[1]	25	[illegible]	[illegible]_25_33.7X2.5_PE_GBT 3091_2015_[illegible]1	3079.18mm
[2]	25	[illegible]	[illegible]_PL_25_PN10_PE_FLRF_GBT 9119_2010_I	1

ISO[illegible]

PIECE NO.	SIZE	LENGTH	END PREP1	END PREP2
<A>	25	940.34mm	SQ.CUT	PE
<B>	25	938.84mm	PE	PE
<C>	25	400mm	PE	PE
<D>	25	400mm	PE	PE
<E>	25	400mm	PE	SQ.CUT

【Where is it】:

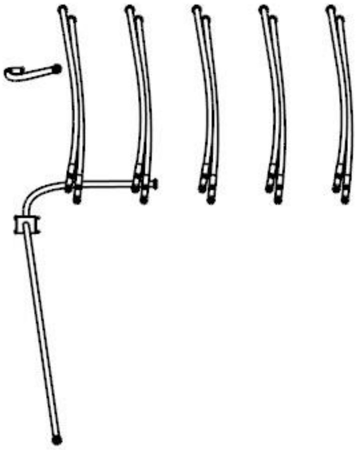
Piping/Tubing >> Drawing >> ISO Generate

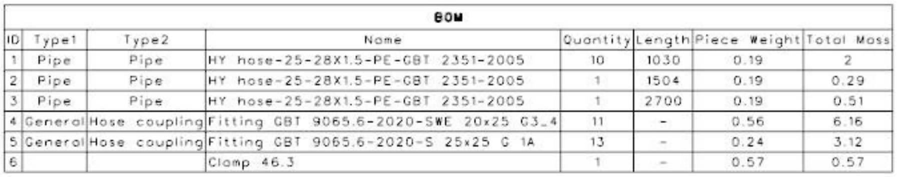
4.4.2.4 Routing BOM Optimization (ZW3D 2025 SP)

After the hydraulic hose design is completed, the output line drawings are required for manufacturing. With the optimized "Routing BOM" function, it is possible to realize the same size and length of hoses, and the statistics can be combined. Fully solve the need for different direction hose which need to combine statistics.

【What can be done】:

- ✓ Hoses of the same specification and length are combined for statistics.
- ✓ one-yard polymorphism.





BOM							
ID	Type1	Type2	Name	Quantity	Length	Piece Weight	Total Mass
1	Pipe	Pipe	HY hose-25-28X1.5-PE-GBT 2351-2005	10	1030	0.19	2
2	Pipe	Pipe	HY hose-25-28X1.5-PE-GBT 2351-2005	1	1504	0.19	0.29
3	Pipe	Pipe	HY hose-25-28X1.5-PE-GBT 2351-2005	1	2700	0.19	0.51
4	General	Hose coupling	Fitting GBT 9065.6-2020-SWE 20x25 G3_4	11	-	0.56	6.16
5	General	Hose coupling	Fitting GBT 9065.6-2020-S 25x25 G 1A	13	-	0.24	3.12
6			Clamp 46.3	1	-	0.57	0.57

【Note】:

- The size and length must be the same.

【Example】:

1. When the pipeline is designed, the hoses of different directions and the same size are all set to the same length for the convenience of manufacturing. When the drawing BOM is counted, the same length hose is counted according to the number.
2. When the pipeline is designed, the copied hose assembly, when the drawing BOM is counted, the standard length of the hose is counted according to the number.

【Where is it】:

Drawing >> Routing >> Routing BOM

4.5 Harness Design

4.5.1 Standard Part Definition Improvement

4.5.1.1 New Std Part Con View Setting Function

When defining standard parts, users need to define standard connector views to facilitate the correct projection of connector locations and view directions of harness standard parts.

【What can be done】：

- ✓ Define different standard connector views according to different types of standards parts.
- ✓ The views of the standard connectors display in different ways in the nailboard.

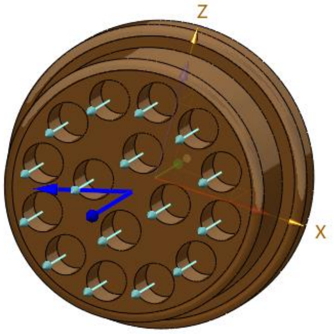
【Note】：

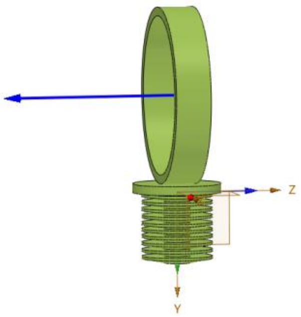
The UI and the setting methods of the Std Part Con View command are different according to different types of standards.

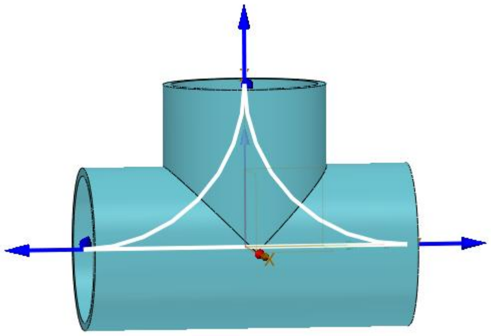
- Connector: The front view of the leading direction of the harness needs to be set.
- Fixture and Leading part: The UI is the same. The side view of the leading direction of the harness needs to be set.

【Example】：

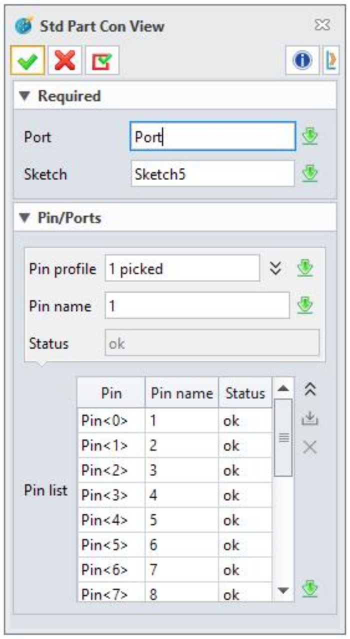
- ✓ Create a connector, fixture, or leading part in the Part/Assembly environment and finish setting ports and leading parts.







- ✓ Complete defining view by the **Std Part Con View** command.


Std Part Con View
Required
Pin/Ports

Port: Port

Sketch: Sketch5

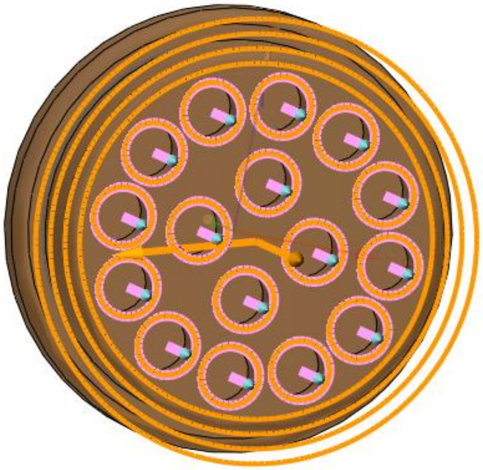
Pin profile: 1 picked

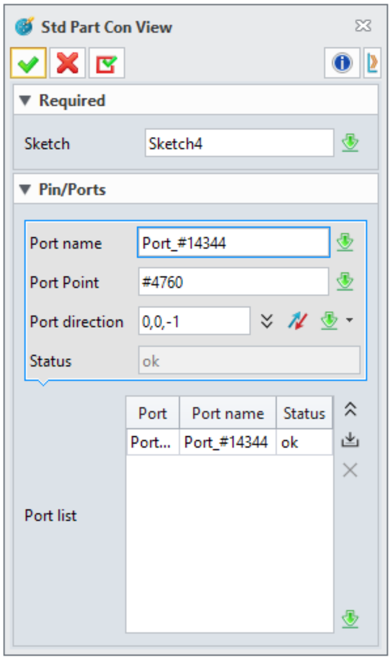
Pin name: 1

Status: ok

Pin list

Pin	Pin name	Status
Pin<0>	1	ok
Pin<1>	2	ok
Pin<2>	3	ok
Pin<3>	4	ok
Pin<4>	5	ok
Pin<5>	6	ok
Pin<6>	7	ok
Pin<7>	8	ok




Std Part Con View
Required
Pin/Ports

Sketch: Sketch4

Port name: Port_#14344

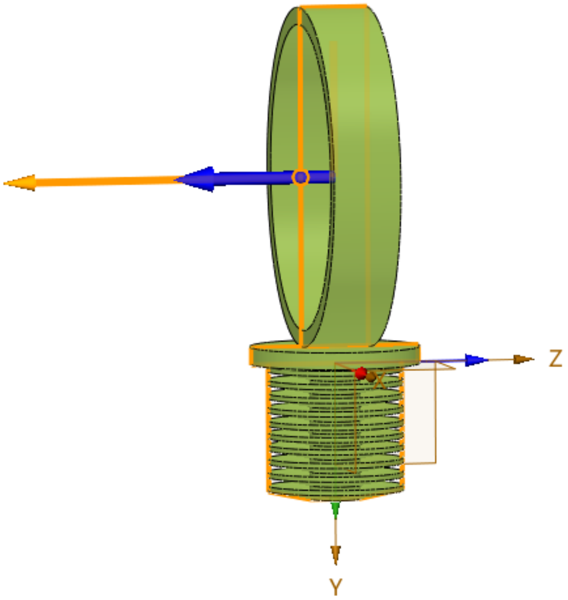
Port Point: #4760

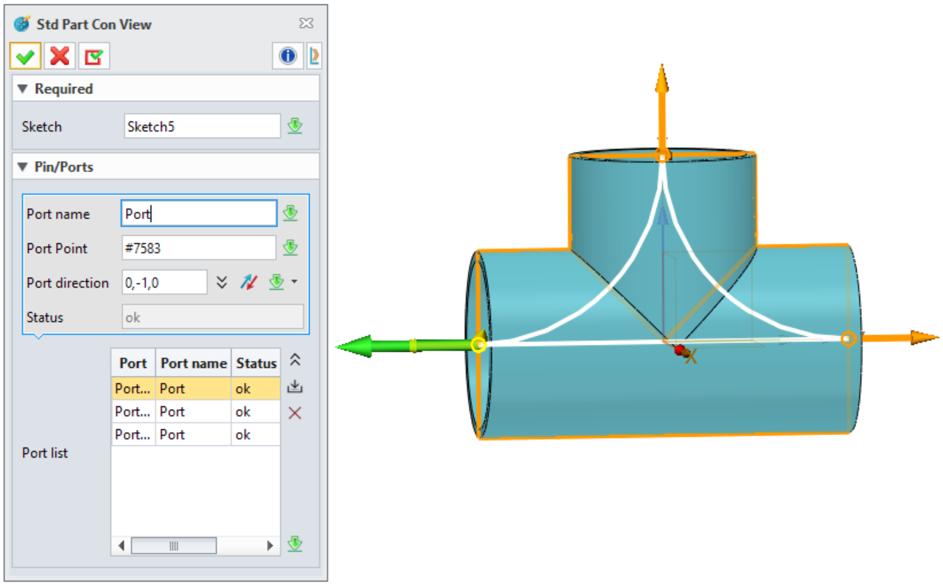
Port direction: 0,0,-1

Status: ok

Port list

Port	Port name	Status
Port...	Port_#14344	ok





Sketch: Sketch5

Port name: Port

Port Point: #7583

Port direction: 0,-1,0

Status: ok

Port list

Port	Port name	Status
Port...	Port	ok
Port...	Port	ok
Port...	Port	ok

【Where is it】：

Insert >> Harness >> Std Part Con View

4.5.1.2 New Attributes to Define Wire and Cable

At present, the Harness module cannot simulate the relaxed state of harnesses under the action of gravity. But to facilitate the calculation of the length of the actual harnesses in relaxed state, the attribute of "Relaxation amount" has been added to the definition of the Wire, Cable, and Protection, which can define the relaxation ratio for each kind of wire. The harness length after the increase of relaxation amount can be automatically reflected in the statistics and measurement of the harness length in the later stage.

【What can be done】：

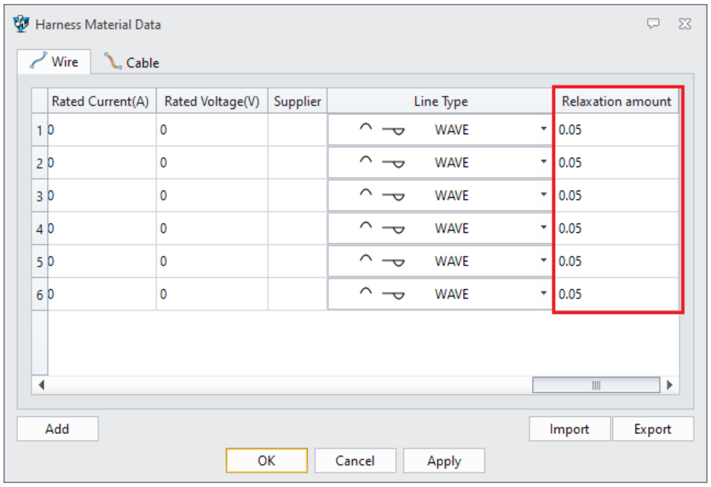
✓ Define the relaxation amount for any type of harness material and express it in the measured length.

【Note】：

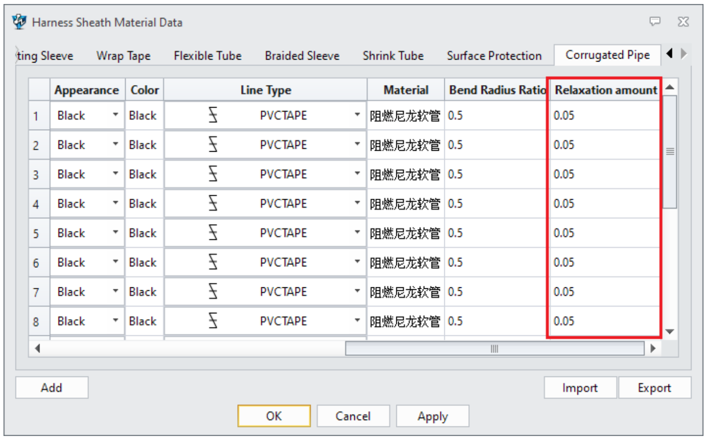
- The relaxation amount is a percentage coefficient, which is equal to adding a certain length to the original length.
- The relaxation amount cannot be simulated in the actual model, which means the effect of harness relaxation cannot be displayed.

【Example】：

1. Add any material in the “Define Wire/Cable” or “Define Protection” command to define “Relaxation amount”.


Harness Material Data
Wire
Cable
Add
Import
Export
OK
Cancel
Apply

	Rated Current(A)	Rated Voltage(V)	Supplier	Line Type	Relaxation amount
1	0	0		WAVE	0.05
2	0	0		WAVE	0.05
3	0	0		WAVE	0.05
4	0	0		WAVE	0.05
5	0	0		WAVE	0.05
6	0	0		WAVE	0.05


Harness Sheath Material Data
ting Sleeve
Wrap Tape
Flexible Tube
Braided Sleeve
Shrink Tube
Surface Protection
Corrugated Pipe
Add
Import
Export
OK
Cancel
Apply

	Appearance	Color	Line Type	Material	Bend Radius Ratio	Relaxation amount
1	Black	Black	PVCTAPE	阻燃尼龙软管	0.5	0.05
2	Black	Black	PVCTAPE	阻燃尼龙软管	0.5	0.05
3	Black	Black	PVCTAPE	阻燃尼龙软管	0.5	0.05
4	Black	Black	PVCTAPE	阻燃尼龙软管	0.5	0.05
5	Black	Black	PVCTAPE	阻燃尼龙软管	0.5	0.05
6	Black	Black	PVCTAPE	阻燃尼龙软管	0.5	0.05
7	Black	Black	PVCTAPE	阻燃尼龙软管	0.5	0.05
8	Black	Black	PVCTAPE	阻燃尼龙软管	0.5	0.05

【Where is it】：

Harness Assembly >> Attribute >> Define Wire/Cable

Harness Assembly >> Attribute >> Define Protection

4.5.2 Harness 3D Modeling Enhancement

4.5.2.1 ★Harness Creation Enhancement

4.5.2.1.1 ★New Free Method to Create Protection

The new creation mode of the protection allows you to create the protection as you create cables without attaching cables. The protection can be nested among multiple layers.

【What can be done】:

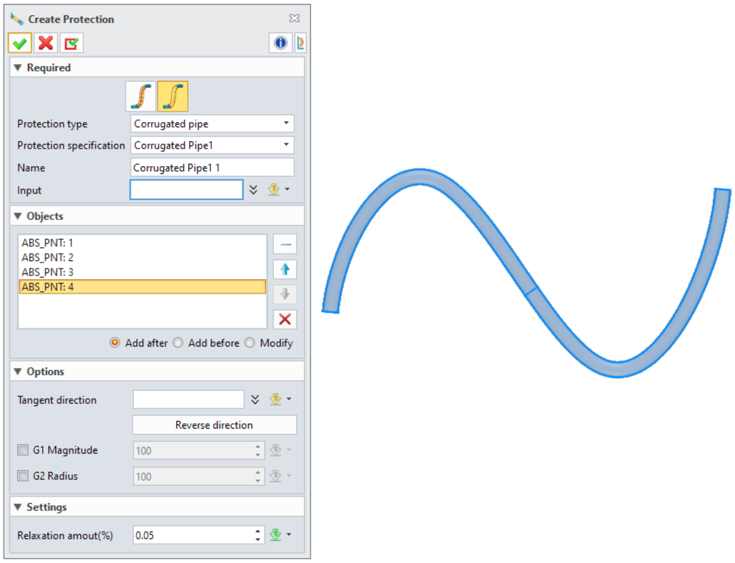
- ✓ Create protection without any limitations.
- ✓ Multiple layers can be nested to protection and managed by section.

【Note】:

- Creating a protection in the PIN of a connector is not recommended because protections lack electrical connections. Even if the protection connects the PIN, its electrical information cannot be identified.

【Example】:

1. Use the "General" type in the "Create Protection" command in the Harness Part environment and select any type of protection and material to create protections at will.



Create Protection

Required

Protection type: Corrugated pipe

Protection specification: Corrugated Pipe1

Name: Corrugated Pipe1 1

Input:

Objects

ABS_PNT: 1
ABS_PNT: 2
ABS_PNT: 3
ABS_PNT: 4

● Add after ○ Add before ○ Modify

Options

Tangent direction:

Reverse direction

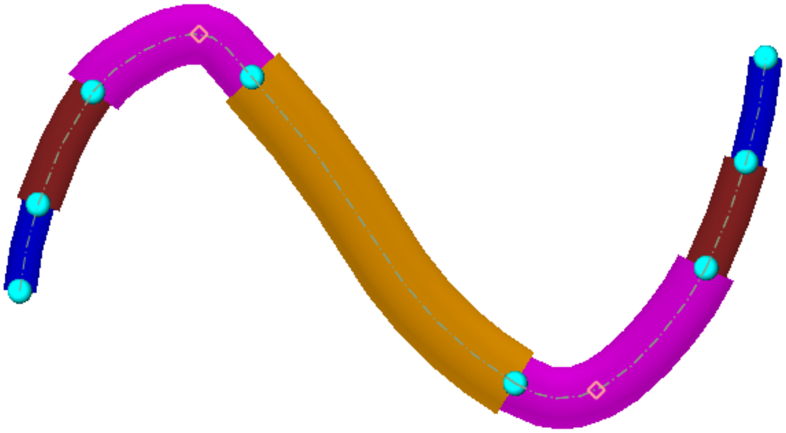
☐ G1 Magnitude: 100

☐ G2 Radius: 100

Settings

Relaxation amout(%): 0.05

2. Select the "Attached" method in the "Create Protection" command and continue creating protection based on the existing protection.



【Where is it】:

Harness Part >> Harness Route >> Create Protection >> General

4.5.2.1.2 ★New Create Branch & Merge Harness

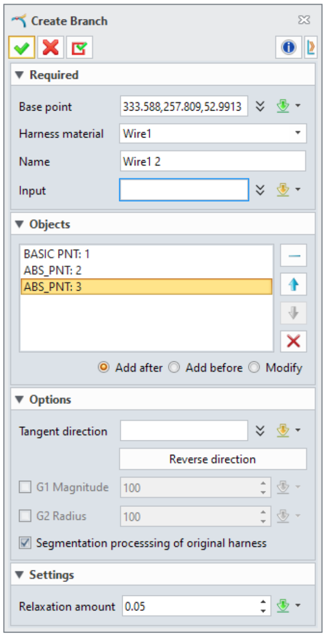
There are riveting and other ways of harness. In this case, it is reflected in the 3D harness modeling that the harness is derived from a point of the existing harness in the tangent direction. In this case, the "Create Branch" command can be used to create such a harness. For branches and main lines, the "Merge Harness" command is allowed to merge them into a single harness.

【What can be done】:

- ✓ Create a branch of the same type at any point of the existing harness.
- ✓ Allow to break the existing harness into two harnesses at the branch point.
- ✓ Support merging two connected and tangent harnesses of the same material into one harness.

【Example】:

1. Use the "Create Branch" command to create a branch at any point of the existing harness and check the "Segmentation processing of original harness" option to break the original harness into two harnesses.



Create Branch

Required

Base point: 333.588,257.809,52.9913

Harness material: Wire1

Name: Wire1 2

Input:

Objects

BASIC PNT: 1
ABS_PNT: 2
ABS_PNT: 3

● Add after ○ Add before ○ Modify

Options

Tangent direction:

Reverse direction

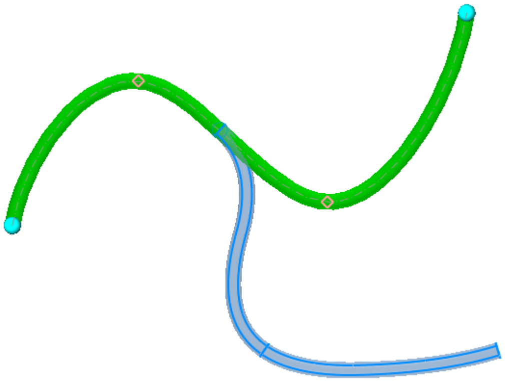
☐ G1 Magnitude: 100

☐ G2 Radius: 100

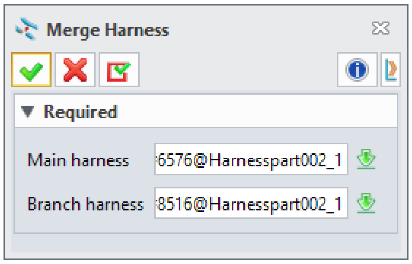
☑ Segmentation processing of original harness

Settings

Relaxation amount: 0.05



2. Used the “Merge Harness” command to merge the two harnesses that are created in the previous step into on harness.

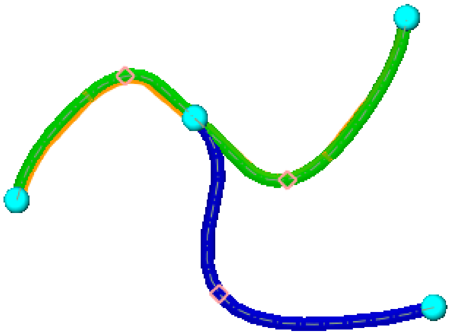


Merge Harness

Required

Main harness: 6576@Harnesspart002_1

Branch harness: 8516@Harnesspart002_1



【Where is it】：

Harness Part >> Harness Route >> Create Branch/ Merge Harness

4.5.2.2 Editing Harness Ability Enhancement

4.5.2.2.1 New Adjustment to Harness Wire

Harness errors often occur during the harness layout process. If you want to adjust the harness material, you can use the "Modify Harness" command to adjust.

【What can be done】：

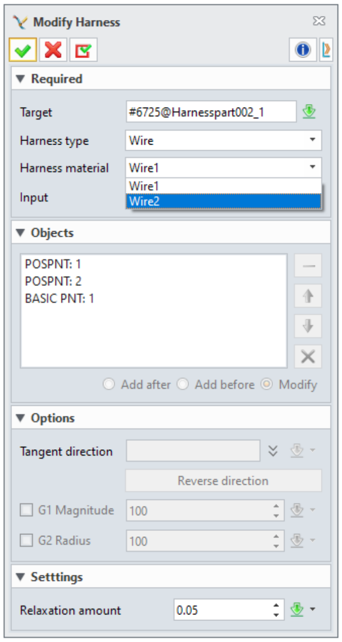
- ✓ Replace material of harness with the existing material in the library.

【Note】:

- Only support adjusting the harness of the same type at present.

【Example】:

1. Use the “**Modify Harness**” command to replace harness material with other materials.



Modify Harness

Required

Target: #6725@Harnesspart002_1

Harness type: Wire

Harness material: Wire1

Wire1

Wire2

Input

Objects

POSPNT: 1
POSPNT: 2
BASIC PNT: 1

○ Add after ○ Add before ● Modify

Options

Tangent direction

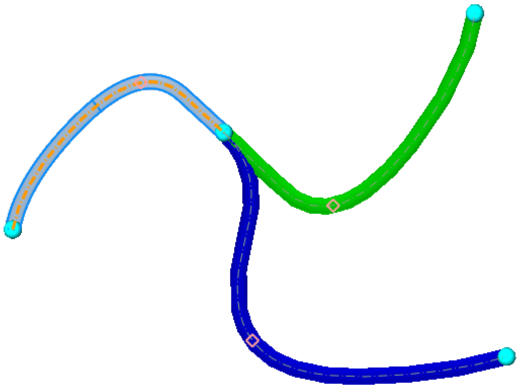
Reverse direction

☐ G1 Magnitude: 100

☐ G2 Radius: 100

Setttings

Relaxation amount: 0.05



【Where is it】:

Harness Part >> Edit>> Modify Harness

4.5.2.2.2 New Cut Harness

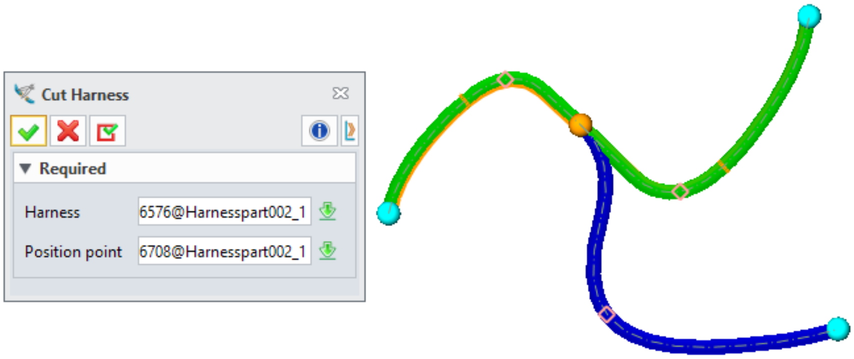
After creating a branched harness, it is often necessary to adjust the material used by some of the harnesses. In this case, you can use the new "Cut Harness" command to break one harness into two harnesses at any position point.

【What can be done】:

- ✓ Support cutting a harness into two harnesses at any position point.

【Example】:

1. Add a position point in the middle of harness and use the “Cut Harness” command to break the selected harness on the selected position point.



Harness 6576@Harnesspart002_1
Position point 6708@Harnesspart002_1

【Where is it】:

Harness Part >> Edit>> Cut Harness

4.5.2.3 Adapt Harness for the Platform

4.5.2.3.1 ★Adapt Harness for Dimension Measurement

Harness needs to verify distance information between dimensions and other parts. Thus, users are allowed to measure related dimensions such as length, radius, and distance.

【What can be done】:

- ✓ Length measure: Allow to measure harness in full length or partial length.
- ✓ Radius measure: Allow to measure the minimum bending radius of the harness and the bend radius of the selected harness at any point.
- ✓ Distance measure: Allow to measure the maximum/minimum distance between the harness and other entities.

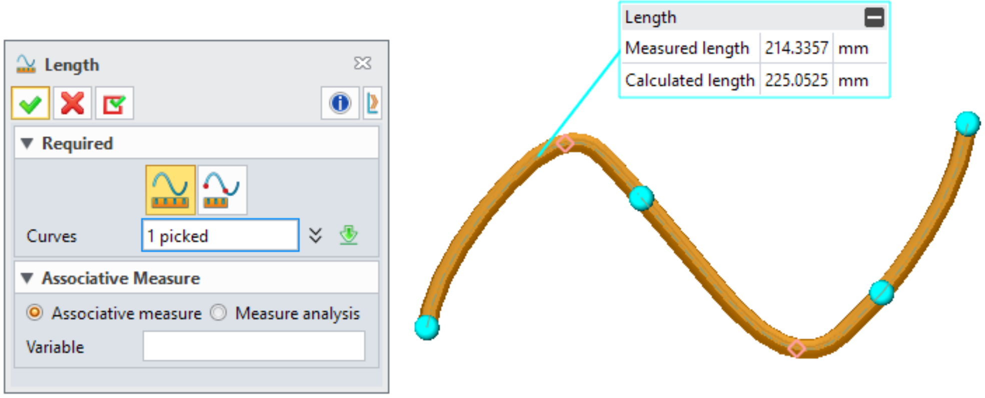
【Note】:

- The measured length can only be operated in the Harness Part environment.

【Example】:

1. Create a harness and add multiple position points. Use the “**Length**” command of “**Measure**”

which provides the Full length and Partial length measurements.



Length

▼ Required

Curves: 1 picked

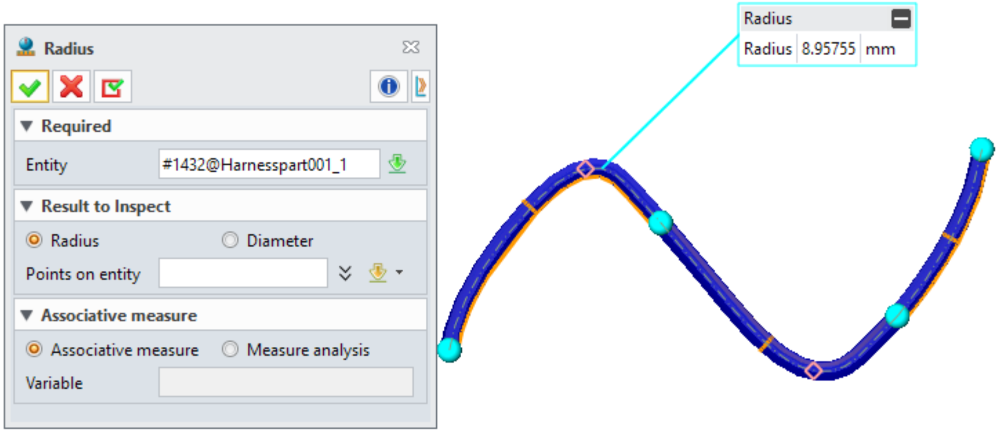
▼ Associative Measure

● Associative measure ○ Measure analysis

Variable

Length		
Measured length	214.3357	mm
Calculated length	225.0525	mm

2. Create a harness. Use the “**Length**” command of “**Measure**”, select the harness to measure its maximum bending radius, and then pick any point on the harness to measure the bending radius of the picked point.



Radius

▼ Required

Entity: #1432@Harnesspart001_1

▼ Result to Inspect

● Radius ○ Diameter

Points on entity

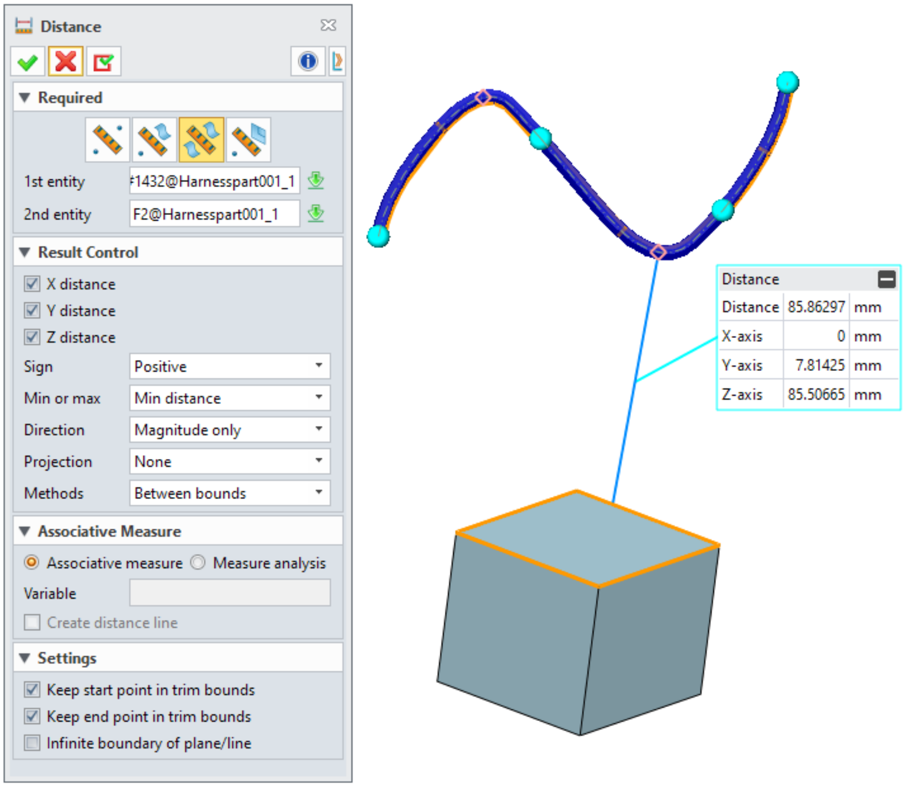
▼ Associative measure

● Associative measure ○ Measure analysis

Variable

Radius		
Radius	8.95755	mm

3. Create a harness and any cube. Use the “**Distance**” command in “**Measure**” to measure the maximum/minimum distance between the harness and the cube.



Distance

Required

1st entity 1432@Harnesspart001_1

2nd entity F2@Harnesspart001_1

Result Control

☑ X distance

☑ Y distance

☑ Z distance

Sign: Positive

Min or max: Min distance

Direction: Magnitude only

Projection: None

Methods: Between bounds

Associative Measure

● Associative measure ○ Measure analysis

Variable

☐ Create distance line

Settings

☑ Keep start point in trim bounds

☑ Keep end point in trim bounds

☐ Infinite boundary of plane/line

Distance		
Distance	85.86297	mm
X-axis	0	mm
Y-axis	7.81425	mm
Z-axis	85.50665	mm

【Where is it】：

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

4.5.2.3.2 Adapt Harness for Drawing Sheet, Export, and Import

Harness in ZW3D 2025 is adapted to the related functions of Drawing Sheet, Import, and Export in the platform. If there is a harness, you can correctly project the harness to a drawing sheet without affecting the correctness of the drawing sheet. You can also export the harness as a no-parameter file in other formats and import it to ZW3D without affecting the correct display of the harness.

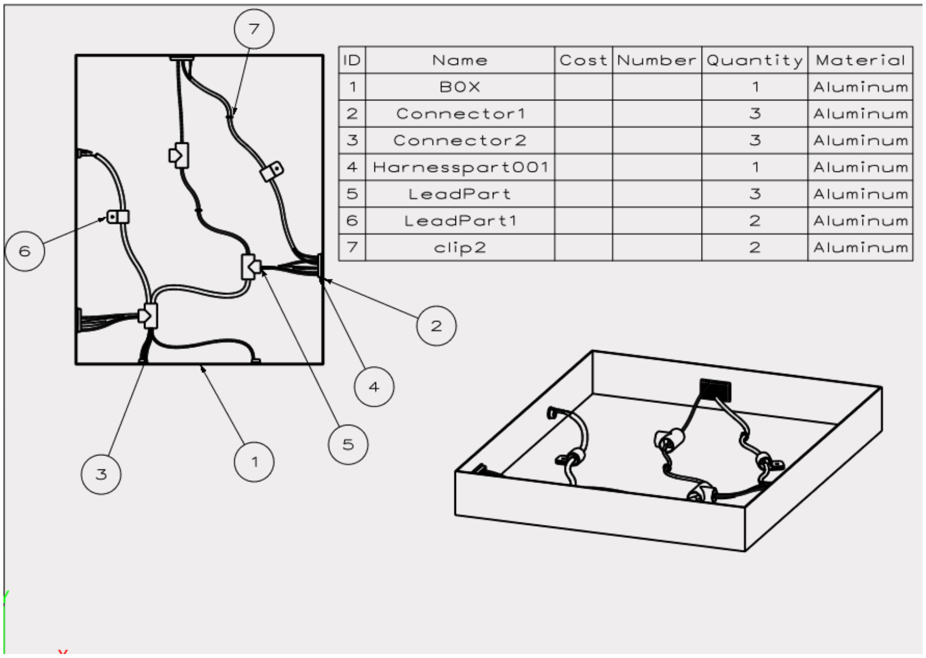
【What can be done】：

✓ Harness supports drawing sheet projection.

✓ Harness supports drawing sheets for dimensioning.

✓ Harness supports drawing sheets for generating tables and ball marks.

✓ Harness supports exporting 3D and 2D to files in other formats.



ID	Name	Cost	Number	Quantity	Material
1	BOX			1	Aluminum
2	Connector1			3	Aluminum
3	Connector2			3	Aluminum
4	Harnesspart001			1	Aluminum
5	LeadPart			3	Aluminum
6	LeadPart1			2	Aluminum
7	clip2			2	Aluminum

【Note】:

- The actual length of the harness cannot be marked in the drawing sheet.

【Example】:

1. Create a model with a harness. Project and dimension it in the drawing sheet, and then export it as a file in other formats.
2. Create a model with a harness and export it as a file in another format.

【Where is it】:

New>> Drawing Sheet

Data Exchange>> Import >> Import/Quick Import/Multi-Import

Data Exchange>> Import >> Import/Multi-Import

4.5.2.3.3 New Harness Diameter Inquiry Function

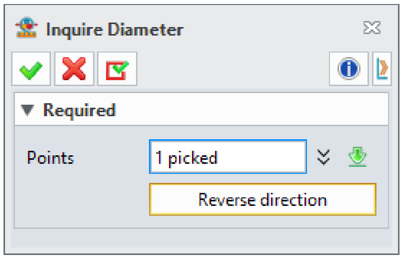
For the harnesses attached together, their protections, fixtures, and leading parts need to be selected according to the minimum diameter of their outer tangential circle. Therefore, the "Inquire Diameter" function has been added to query the diameter of the outer tangential circle of harnesses at each position point.

【What can be done】:

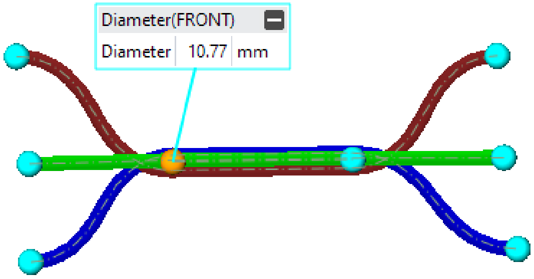
- ✓ Inquire about the minimum diameter of the outer tangential circle in two directions.

【Example】:

1. Create multiple harnesses attached together. Use the “Inquire Diameter” command to pick a position point to inquire about the minimum outer tangential circle’s diameter.



Points: 1 picked



Diameter(FRONT)
Diameter 10.77 mm

【Where is it】:

Harness Part >> Inquire >> Inquire Diameter

4.5.2.4 New Harness Statistics Table

4.5.2.4.1 New Harness 3D BOM

The “Harness 3D BOM” command has been added to the Harness Assembly and Harness Part environments to carry out quick statistics of harness material information.

【What can be done】:

- ✓ Allow to export harness 3D BOM information to files in the Excel format.
- ✓ There are differences between the Harness Part environment and the Harness Assembly environment. The Harness Part environment only allows the material information statistics according to material names, while the Harness Assembly environment also allows the material information statistics based on harness names.

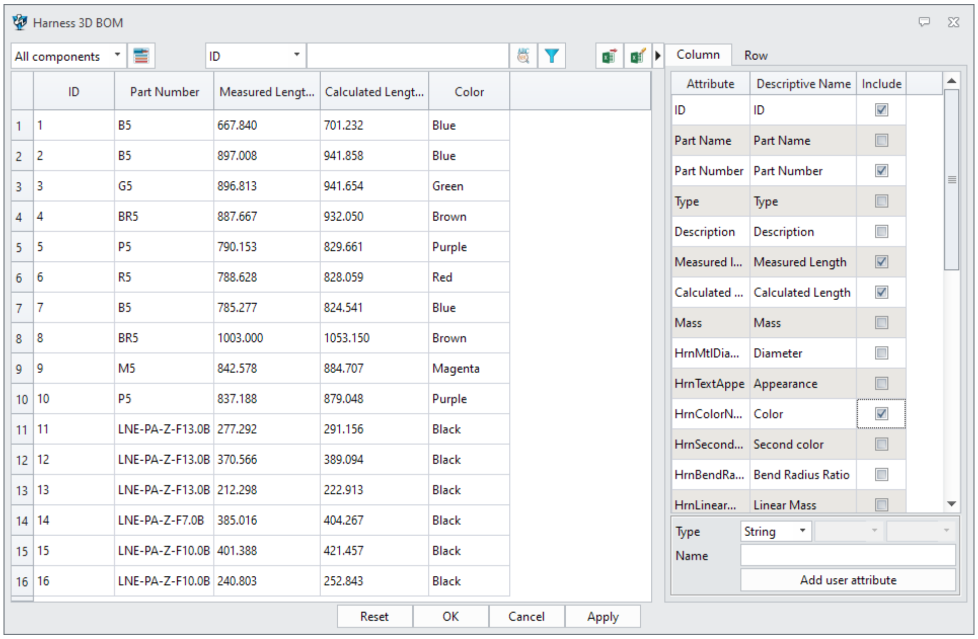
【Note】:

- Only the wire material is counted by Harness 3D BOM. As for other standard material information, it needs to use Harness 3D in the Assembly environment.

- All properties collected in the Harness 3D BOM are derived from the properties of cable and protection.

【Example】:

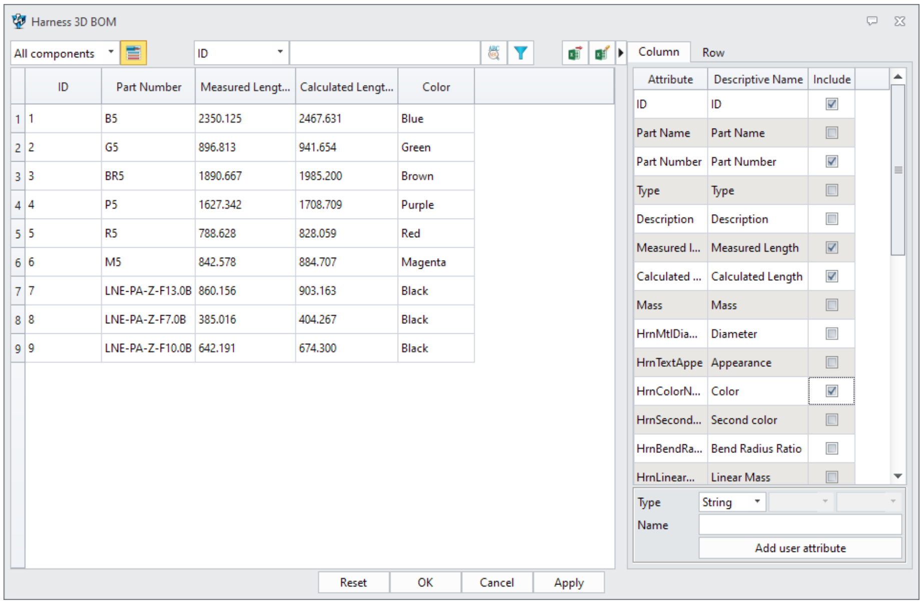
1. To create a complex harness, use the "**Harness 3D BOM"** function to collect material information about harnesses.



	ID	Part Number	Measured Lengt...	Calculated Lengt...	Color
1	1	B5	667.840	701.232	Blue
2	2	B5	897.008	941.858	Blue
3	3	G5	896.813	941.654	Green
4	4	BR5	887.667	932.050	Brown
5	5	P5	790.153	829.661	Purple
6	6	R5	788.628	828.059	Red
7	7	B5	785.277	824.541	Blue
8	8	BR5	1003.000	1053.150	Brown
9	9	M5	842.578	884.707	Magenta
10	10	P5	837.188	879.048	Purple
11	11	LNE-PA-Z-F13.0B	277.292	291.156	Black
12	12	LNE-PA-Z-F13.0B	370.566	389.094	Black
13	13	LNE-PA-Z-F13.0B	212.298	222.913	Black
14	14	LNE-PA-Z-F7.0B	385.016	404.267	Black
15	15	LNE-PA-Z-F10.0B	401.388	421.457	Black
16	16	LNE-PA-Z-F10.0B	240.803	252.843	Black

Attribute	Descriptive Name	Include
ID	ID	☑
Part Name	Part Name	☐
Part Number	Part Number	☑
Type	Type	☐
Description	Description	☐
Measured l...	Measured Length	☑
Calculated ...	Calculated Length	☑
Mass	Mass	☐
HrnMtlDia...	Diameter	☐
HrnTextAppe	Appearance	☐
HrnColorN...	Color	☑
HrnSecond...	Second color	☐
HrnBendRa...	Bend Radius Ratio	☐
HrnLinear...	Linear Mass	☐

Type: String
Name:
Add user attribute



	ID	Part Number	Measured Lengt...	Calculated Lengt...	Color
1	1	B5	2350.125	2467.631	Blue
2	2	G5	896.813	941.654	Green
3	3	BR5	1890.667	1985.200	Brown
4	4	P5	1627.342	1708.709	Purple
5	5	R5	788.628	828.059	Red
6	6	M5	842.578	884.707	Magenta
7	7	LNE-PA-Z-F13.0B	860.156	903.163	Black
8	8	LNE-PA-Z-F7.0B	385.016	404.267	Black
9	9	LNE-PA-Z-F10.0B	642.191	674.300	Black

Attribute	Descriptive Name	Include
ID	ID	☑
Part Name	Part Name	☐
Part Number	Part Number	☑
Type	Type	☐
Description	Description	☐
Measured l...	Measured Length	☑
Calculated ...	Calculated Length	☑
Mass	Mass	☐
HrnMtlDia...	Diameter	☐
HrnTextAppe	Appearance	☐
HrnColorN...	Color	☑
HrnSecond...	Second color	☐
HrnBendRa...	Bend Radius Ratio	☐
HrnLinear...	Linear Mass	☐

Type: String

Name:

Add user attribute

【Where is it】:

Harness Part >> Table/BOM >> Harness 3D BOM

4.5.2.4.2 ★New Global Wiring Table

For manual wiring, the wiring relationships of harnesses may be incorrectly connected. Use the "Global Wiring Table" function to quickly identify the actual wiring relationships of harnesses and screen the electrical information in the future.

【What can be done】:

- ✓ Global Wiring Table allows the wiring information statistics for all wires connected to connectors in the current Harness Part environment.
- ✓ Global Wiring Table can be output as an EXCEL table.

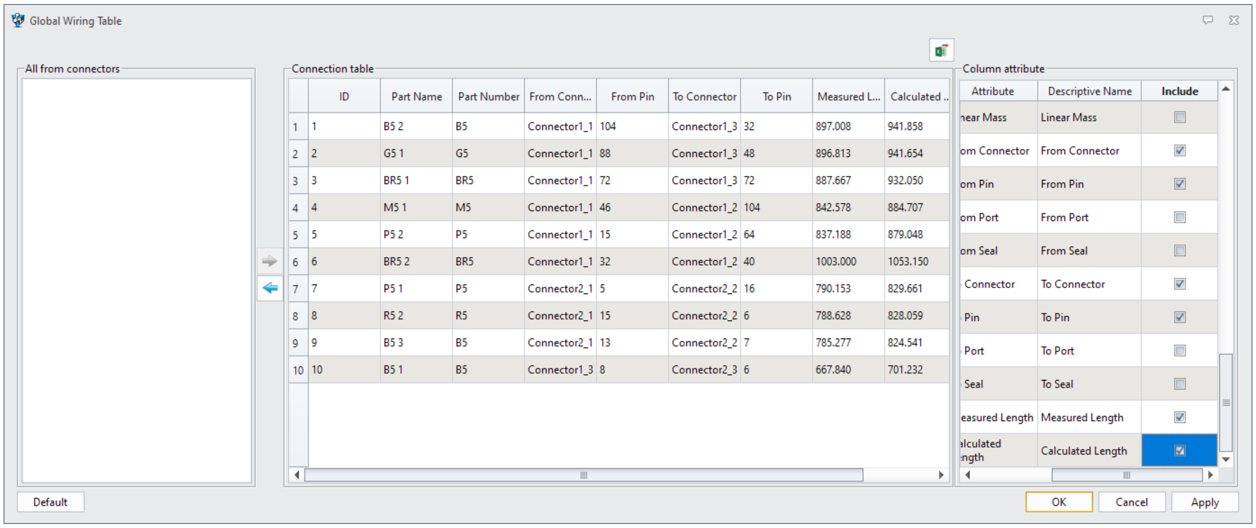
【Note】:

- Global Wiring Table only collects information about electrical connections and related properties of wires.

【Example】:

1. Create a complex harness. Use **Global Wiring Table** and select the start connector to identify all

the wiring relationships connected to this connector.



Connection table

	ID	Part Name	Part Number	From Conn...	From Pin	To Connector	To Pin	Measured L...	Calculated ..
1	1	B5 2	B5	Connector1_1	104	Connector1_3	32	897.008	941.858
2	2	G5 1	G5	Connector1_1	88	Connector1_3	48	896.813	941.654
3	3	BR5 1	BR5	Connector1_1	72	Connector1_3	72	887.667	932.050
4	4	M5 1	M5	Connector1_1	46	Connector1_2	104	842.578	884.707
5	5	P5 2	P5	Connector1_1	15	Connector1_2	64	837.188	879.048
6	6	BR5 2	BR5	Connector1_1	32	Connector1_2	40	1003.000	1053.150
7	7	P5 1	P5	Connector2_1	5	Connector2_2	16	790.153	829.661
8	8	R5 2	R5	Connector2_1	15	Connector2_2	6	788.628	828.059
9	9	B5 3	B5	Connector2_1	13	Connector2_2	7	785.277	824.541
10	10	B5 1	B5	Connector1_3	8	Connector2_3	6	667.840	701.232

Column attribute

Attribute	Descriptive Name	Include
near Mass	Linear Mass	☐
om Connector	From Connector	☑
om Pin	From Pin	☑
om Port	From Port	☐
om Seal	From Seal	☐
Connector	To Connector	☑
Pin	To Pin	☑
Port	To Port	☐
Seal	To Seal	☐
easured Length	Measured Length	☑
alculated ngth	Calculated Length	☑

【Where is it】:

Harness Part >> Table >> Global Wiring Table

4.5.2.4.3 New Electrical Schematic Table

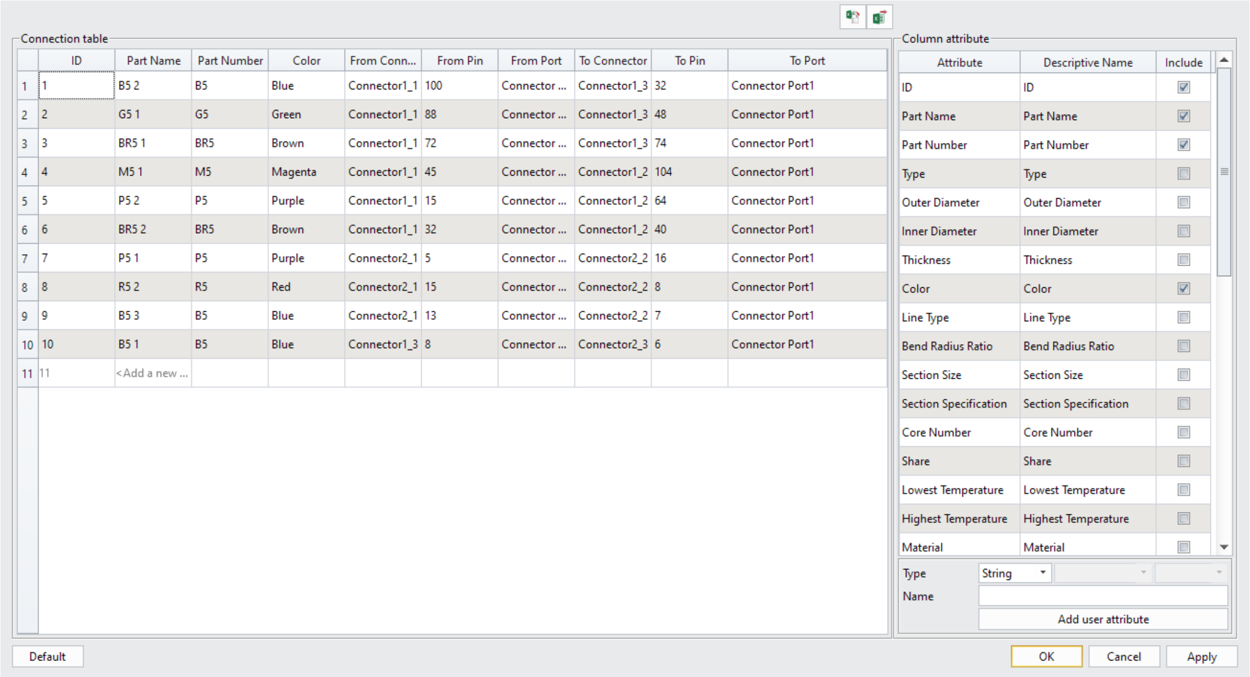
To check whether the electrical connections of 3D wiring are correct, the Electrical Schematic Table function has been added, which can edit, import, and export electrical schematic tables.

【What can be done】:

- ✓ Electrical schematic tables can be edited directly in ZW3D.
- ✓ Electrical schematic tables scan input and output files in Excel format.

【Example】:

1. Open the "**Electrical Schematic Table**" command to enter the wiring information in the electrical schematic diagram, or use the global wiring table output in Excel format to enter the information into the table directly.



Connection table

	ID	Part Name	Part Number	Color	From Conn...	From Pin	From Port	To Connector	To Pin	To Port
1	1	B5 2	B5	Blue	Connector1_1	100	Connector ...	Connector1_3	32	Connector Port1
2	2	G5 1	G5	Green	Connector1_1	88	Connector ...	Connector1_3	48	Connector Port1
3	3	BR5 1	BR5	Brown	Connector1_1	72	Connector ...	Connector1_3	74	Connector Port1
4	4	M5 1	M5	Magenta	Connector1_1	45	Connector ...	Connector1_2	104	Connector Port1
5	5	P5 2	P5	Purple	Connector1_1	15	Connector ...	Connector1_2	64	Connector Port1
6	6	BR5 2	BR5	Brown	Connector1_1	32	Connector ...	Connector1_2	40	Connector Port1
7	7	P5 1	P5	Purple	Connector2_1	5	Connector ...	Connector2_2	16	Connector Port1
8	8	R5 2	R5	Red	Connector2_1	15	Connector ...	Connector2_2	8	Connector Port1
9	9	B5 3	B5	Blue	Connector2_1	13	Connector ...	Connector2_2	7	Connector Port1
10	10	B5 1	B5	Blue	Connector1_3	8	Connector ...	Connector2_3	6	Connector Port1
11	11	<Add a new ...								

Column attribute

Attribute	Descriptive Name	Include
ID	ID	☑
Part Name	Part Name	☑
Part Number	Part Number	☑
Type	Type	☐
Outer Diameter	Outer Diameter	☐
Inner Diameter	Inner Diameter	☐
Thickness	Thickness	☐
Color	Color	☑
Line Type	Line Type	☐
Bend Radius Ratio	Bend Radius Ratio	☐
Section Size	Section Size	☐
Section Specification	Section Specification	☐
Core Number	Core Number	☐
Share	Share	☐
Lowest Temperature	Lowest Temperature	☐
Highest Temperature	Highest Temperature	☐
Material	Material	☐

Type: String
Name:

【Where is it】:

Harness Part >> Table >> Electrical Schematic Table

4.5.2.4.4 New Wiring Info Compare Check

When there is a global wiring table and an electrical schematic table, you can use the “**Wiring Info Compare Check**” function to quickly identify the correctness of the actual 3D electrical connection information based on the two tables.

【What can be done】:

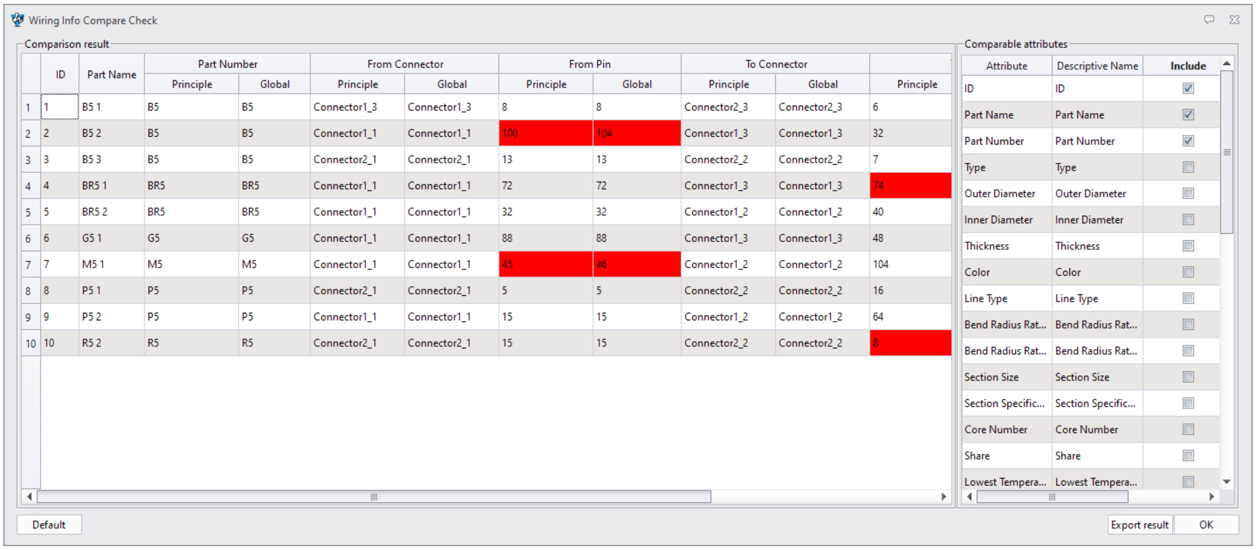
- ✓ When the global wiring table information is different from the result of the electrical schematic table, it will be displayed in a special color in the corresponding line.
- ✓ The Wiring Info Compare Check list can be output in Excel format.

【Note】:

- The electrical information comparison check list is based on the electrical schematic table. If no electrical schematic table or global wiring table is available, this command cannot be used.

【Example】:

Create a harness. After identifying the global wiring table and editing the electrical schematic table, directly open the " **Wiring Info Compare Check** " command, and you can automatically check the different properties of the global wiring table and the electrical schematic table.



	ID	Part Name	Part Number		From Connector		From Pin		To Connector		
			Principle	Global	Principle	Global	Principle	Global	Principle	Global	Principle
1	1	B5 1	B5	B5	Connector1_3	Connector1_3	8	8	Connector2_3	Connector2_3	6
2	2	B5 2	B5	B5	Connector1_1	Connector1_1	100	104	Connector1_3	Connector1_3	32
3	3	B5 3	B5	B5	Connector2_1	Connector2_1	13	13	Connector2_2	Connector2_2	7
4	4	BR5 1	BR5	BR5	Connector1_1	Connector1_1	72	72	Connector1_3	Connector1_3	74
5	5	BR5 2	BR5	BR5	Connector1_1	Connector1_1	32	32	Connector1_2	Connector1_2	40
6	6	G5 1	G5	G5	Connector1_1	Connector1_1	88	88	Connector1_3	Connector1_3	48
7	7	M5 1	M5	M5	Connector1_1	Connector1_1	45	46	Connector1_2	Connector1_2	104
8	8	P5 1	P5	P5	Connector2_1	Connector2_1	5	5	Connector2_2	Connector2_2	16
9	9	P5 2	P5	P5	Connector1_1	Connector1_1	15	15	Connector1_2	Connector1_2	64
10	10	R5 2	R5	R5	Connector2_1	Connector2_1	15	15	Connector2_2	Connector2_2	8

Attribute	Descriptive Name	Include
ID	ID	☑
Part Name	Part Name	☑
Part Number	Part Number	☑
Type	Type	☐
Outer Diameter	Outer Diameter	☐
Inner Diameter	Inner Diameter	☐
Thickness	Thickness	☐
Color	Color	☐
Line Type	Line Type	☐
Bend Radius Rat...	Bend Radius Rat...	☐
Bend Radius Rat...	Bend Radius Rat...	☐
Section Size	Section Size	☐
Section Specific...	Section Specific...	☐
Core Number	Core Number	☐
Share	Share	☐
Lowest Tempera...	Lowest Tempera...	☐

【Where is it】：

Harness Part >> Table >> Wiring Info Compare Check

4.5.2.5 Harness Manager and Attributes Adjustment

4.5.2.5.1 Harness Manager Enhancement

The Harness Object Manager of ZW3D 2025 has added statistical attribute information and can express hierarchical relationships, which helps users identify related information about harnesses.

【What can be done】：

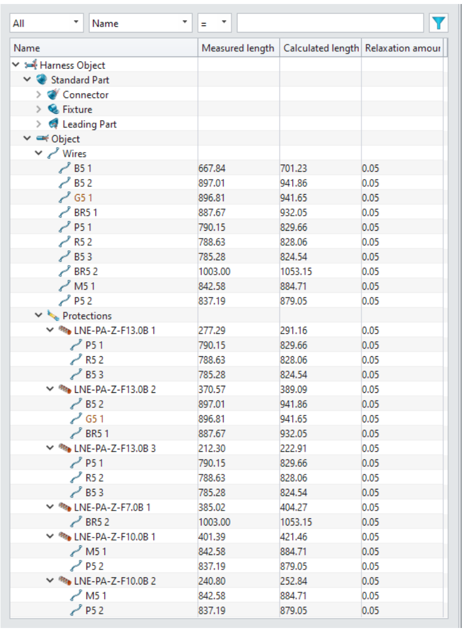
- ✓ The options of harness calculation length, measurement length, and relaxation amount are added for users to easily view harness length and attribute information.
- ✓ The new Harness Manager adds a hierarchical display, which can express the package relationship between harnesses.
- ✓ The exception display function has been added for a lower bending radius than the set value. When the actual bending radius of the harness is lower than the set value, the corresponding harness will be displayed in a different color on the manager.

【Note】：

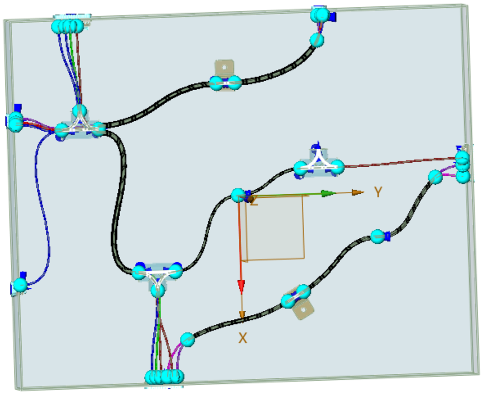
- The hierarchical display in the Harness Object Manager can only display the nested relationships of its first layer.

【Example】：

1. Create a complex harness. Check **Cable Harnesses Object Manager**, and you can find the hierarchical relationship between the protection and the cable. The harnesses with an abnormal bending radius are displayed in a special color.
2. Open **Property Manager**, and you can find the statistical harness information increasing. Select all the properties, and the related information can be displayed in **Wiring Harness Object Manager**.



Name	Measured length	Calculated length	Relaxation amour
Harness Object			
Standard Part			
Connector			
Fixture			
Leading Part			
Object			
Wires			
B5 1	667.84	701.23	0.05
B5 2	897.01	941.86	0.05
G5 1	896.81	941.65	0.05
BR5 1	887.67	932.05	0.05
P5 1	790.15	829.66	0.05
R5 2	788.63	828.06	0.05
B5 3	785.28	824.54	0.05
BR5 2	1003.00	1053.15	0.05
M5 1	842.58	884.71	0.05
P5 2	837.19	879.05	0.05
Protections			
LNE-PA-Z-F13.0B 1	277.29	291.16	0.05
P5 1	790.15	829.66	0.05
R5 2	788.63	828.06	0.05
B5 3	785.28	824.54	0.05
LNE-PA-Z-F13.0B 2	370.57	389.09	0.05
B5 2	897.01	941.86	0.05
G5 1	896.81	941.65	0.05
BR5 1	887.67	932.05	0.05
LNE-PA-Z-F13.0B 3	212.30	222.91	0.05
P5 1	790.15	829.66	0.05
R5 2	788.63	828.06	0.05
B5 3	785.28	824.54	0.05
LNE-PA-Z-F7.0B 1	385.02	404.27	0.05
BR5 2	1003.00	1053.15	0.05
LNE-PA-Z-F10.0B 1	401.39	421.46	0.05
M5 1	842.58	884.71	0.05
P5 2	837.19	879.05	0.05
LNE-PA-Z-F10.0B 2	240.80	252.84	0.05
M5 1	842.58	884.71	0.05
P5 2	837.19	879.05	0.05



【Where is it】：

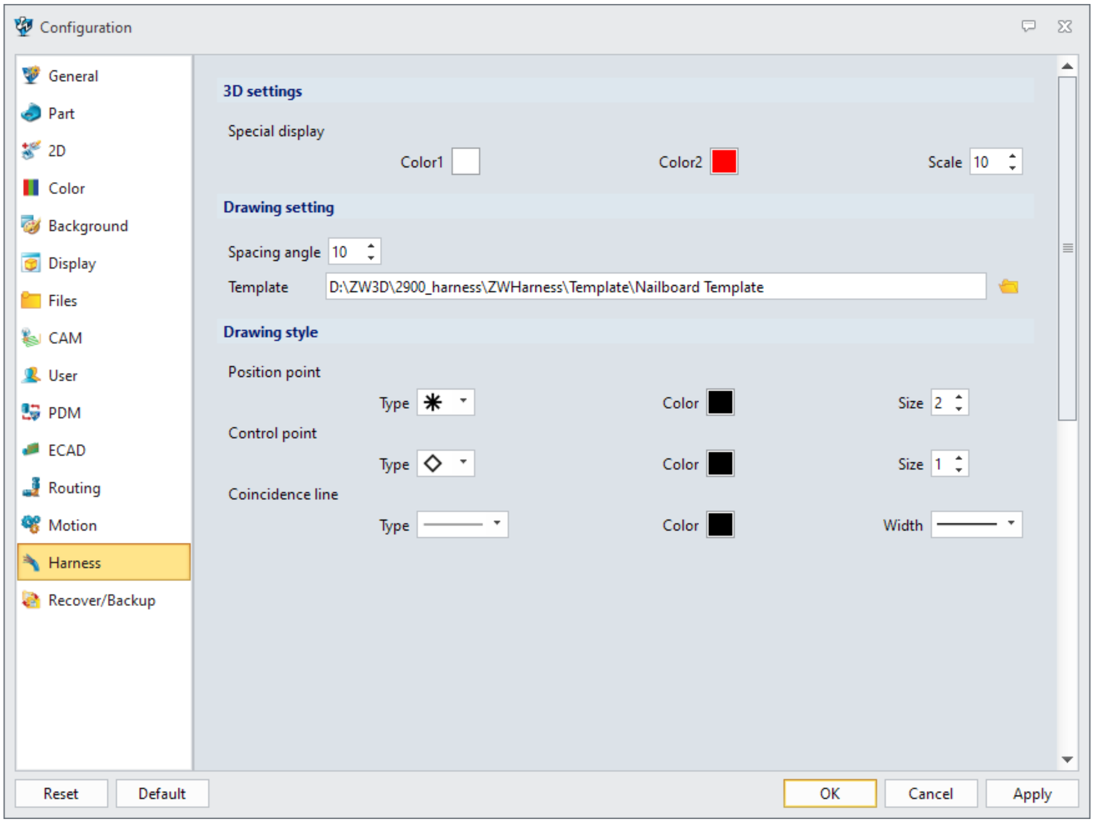
Harness Part >> Manager >> Harness Object Manager

4.5.2.5.2 New Harness Attribute Setting

In ZW3D 2025, when the actual bending radius of the harness is lower than the set value, the appearance of the harness will be displayed in an abnormal way in 3D. For the abnormally displayed appearance, you can set its color and proportion in the properties. In addition, you can also set the line and point shape as well as template position in the Harness Nailboard in Attribute.

【Example】:

1. In any environment, open the **Harness** section in **Configuration** to set it up.


Configuration
General
Part
2D
Color
Background
Display
Files
CAM
User
PDM
ECAD
Routing
Motion
Harness
Recover/Backup
Reset
Default
OK
Cancel
Apply

3D settings

Special display

Color1 ⬜ Color2 🟥 Scale 10

Drawing setting

Spacing angle 10

Template D:\ZW3D\2900_harness\ZWHarness\Template\Nailboard Template

Drawing style

Position point

Type ✳ Color ■ Size 2

Control point

Type ◇ Color ■ Size 1

Coincidence line

Type —— Color ■ Width ——

【Where is it】:

Configuration >> Harness

4.5.3 ★New Harness Nailboard

4.5.3.1 ★New Nailboard Command

For the drawings required for harness processing, ZW3D 2025 provides the "Nailboard" command, which can automatically flatter 3D harnesses, and automatically generate the required standard part view, harness table, harness dimension, and other information.

【What can be done】:

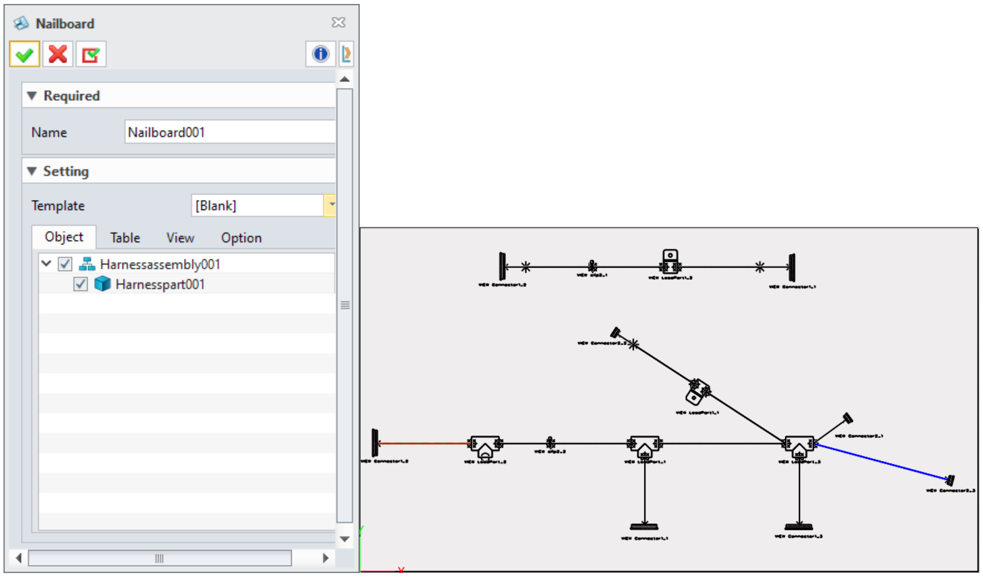
- ✓ Set sheet templates.
- ✓ Select the type of automatically generated tables.
- ✓ Select the color hatching type for the connector PIN, including Loop hatch and Fan hatch.
- ✓ Choose whether to generate dimensions.
- ✓ Used for linear expression.

【Note】:

- If there is a loop harness in 3D Harness, the nailboard cannot be correctly output.
- Currently, users need to adjust the overlap problem in the nailboard.
- As for the problem of the nailboard exceeding the range of the drawing, users need to adjust the flattening state or drawing size.

【Example】:

1. Create a harness part and return to the Harness Assembly environment. Use the "Nailboard" command, select the harness parts and tables of the nailboard that need to be output, choose whether to output views and dimensions, and finally a nailboard is generated.



Nailboard

Required

Name Nailboard001

Setting

Template [Blank]

Object Table View Option

☑ Harnessassembly001

☑ Harnesspart001

【Where is it】:

Harness Assembly >> Drawing Sheet >> Nailboard

4.5.3.2 New Nailboard Editing Command

4.5.3.2.1 ★Rotate Segment Function

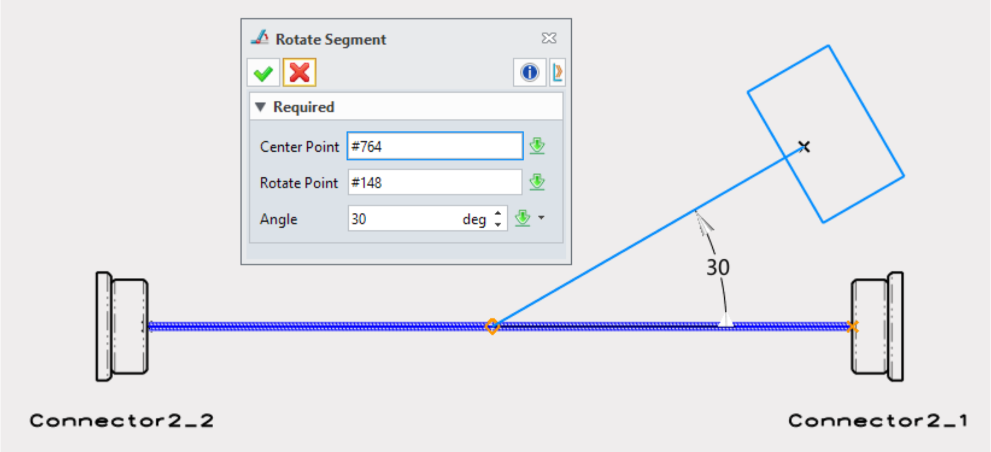
When the 3D harness is flattened into a nailboard, some harnesses are out of the sheet range or overlap. In this case, you can run the “Rotate Cable” command to adjust the flattening layout to meet the sheet requirements.

【What can be done】：

- ✓ When rotating the harness, both the rotation center and the rotation point can be position points and control points.
- ✓ When rotating the harness, you can input the angle in manual or directly adjust it in the sheet manually.

【Example】：

1. After creating any harness to generate the nailboard, run the **Rotate Segment** command to select the fixed point and rotation point to rotate the segment and adjust the flattening state.



Center Point #764

Rotate Point #148

Angle 30 deg

【Where is it】：

Nailboard >> Edit >> Rotate Segment

4.5.3.2.2 ★Shrink Segment Function

When the 3D harness is flattened to be a nailboard, some harnesses may be out of the sheet range. In this case, run the “Shrink Segment” command to adjust the flattening layout to meet the sheet range requirements.

【What can be done】:

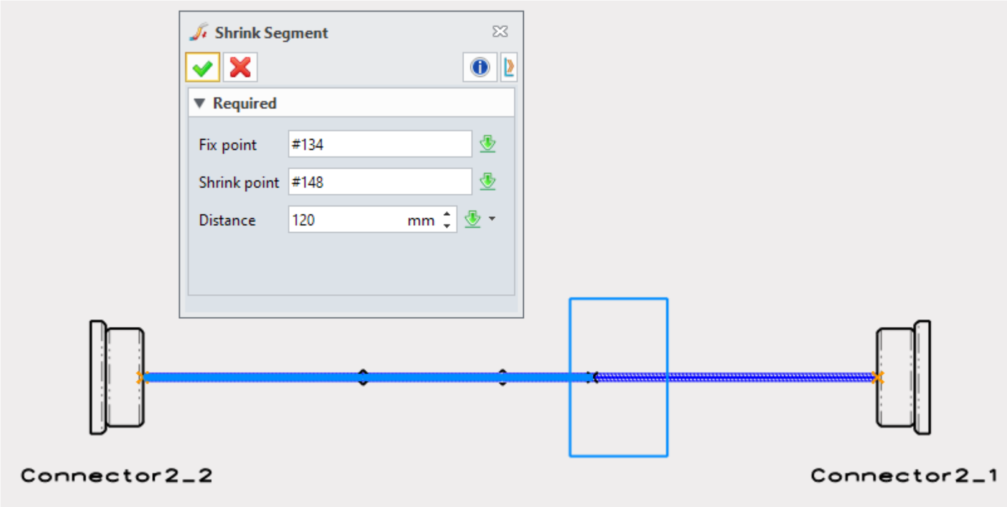
- ✓ When shrinking segments, the fixed point and the moving point are position points.
- ✓ Shrinking segments does not affect the correctness of dimensions.
- ✓ The shrunk segments can be restored.

【Note】:

- Segments cannot be extended.
- The segment length cannot be shortened to 0.

【Example】:

1. Create a harness and enter the Nailboard environment. Use the “Shrink Segment” command and select the fixed point and the moving point to shorten the segment to the specific length.



Shrink Segment

Required

Fix point #134

Shrink point #148

Distance 120 mm

【Where is it】:

Nailboard >> Edit >> Shrink Segment

4.5.3.2.3 ★Insert Control Point Function

When the existing position points are not enough for users to adjust the flattening layout of the harness, use the "Insert Control Point" function to add arbitrary points at the desired positions.

【What can be done】:

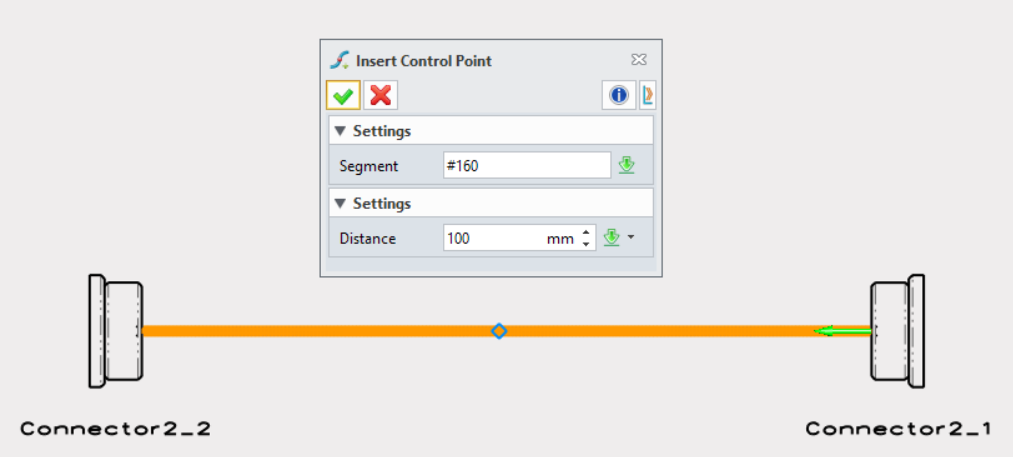
- ✓ Control points do not affect the 3D layout and are not displayed in the 3D harness.
- ✓ Unlike position points, control points can be deleted in the nailboard.
- ✓ Control points has the same function as position points in the nailboard. Both can be properly dimensioned and referenced.

【Note】:

- New control points cannot overlap the existing position points and control points.

【Example】:

1. Create a harness and enter the Nailboard environment. Use the "Insert Control Point" command to select any point on the cable to add a control point and set the distance information.



Insert Control Point

Settings

Segment #160

Settings

Distance 100 mm

【Where is it】:

Nailboard >> Edit >> Insert Control Point

4.5.3.2.4 Regeneration Function

When the 3D harness or the standard parts are changed, two regeneration methods are provided in the nailboard, and users can choose according to their needs.

【What can be done】：

- ✓ Regen: At this point, regenerate all the views, tables, and the harness flattening state.
- ✓ Regen All Tables: At this point, only regenerate the various table information of the nailboard.

【Where is it】：

Nailboard >> Edit >> Regen

Nailboard >> Edit >> Regen All Tables

4.5.3.3 New View Command

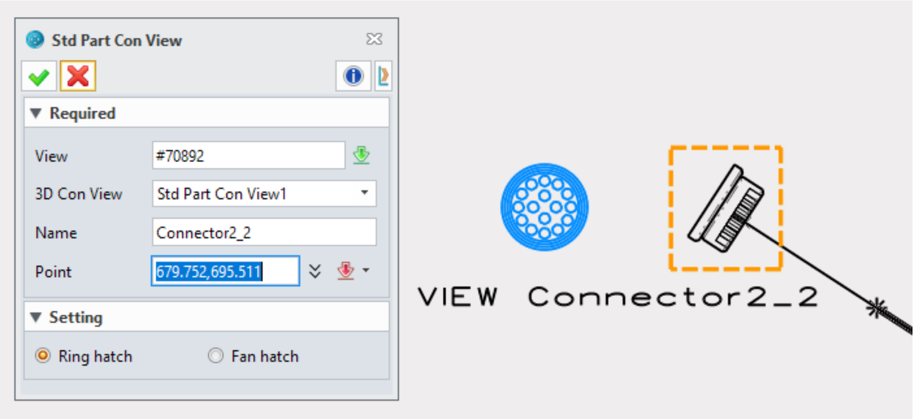
After generating the nailboard, the projection direction of the standard parts may not be the desired direction. In addition, users often need to edit the flattening state. At this time, the connection view direction of the connector may be different from what is expected. In this case, it can be adjusted by commands in the view.

【What can be done】：

- ✓ Std Part Con View: The connection direction view of the connector. It can adjust the color hatching way of PIN, including Loop hatch and Fan hatch.
- ✓ Rotate view: It can adjust the projection direction of a standard part in the flat view to a desire direction.

【Example】：

1. You can continue to use the nailboard, use the "**Std Part Con View**" command, select the target connector, select the PIN filling method, and select the wiring view position.



Std Part Con View

Required

View #70892

3D Con View Std Part Con View1

Name Connector2_2

Point 679.752,695.511

Setting

● Ring hatch ○ Fan hatch

2. Select any connector and use the “**Rotate View”** command to rotate the connector to any target projection direction.

【Where is it】：

Nailboard >> View >> Std Part Con View

Nailboard >> View >> Rotate View

4.5.3.4 ★New Statistics Table Command

At present, seven tables are provided in the nailboard to help users to summarize the material and attribute information about harnesses. Types of tables are as follows:

- Harness BOM: Collects information about the materials used by the current harness.
- Wires Table: Collects information about the properties of the wires and cables used by the current harness.
- Protection Table: Collects information about the properties of the protection layer used by the current harness.
- Connectable Object Table: statistics of the wiring information of each connector, you can choose whether to ignore the unconnected PIN.
- Standard Part Table: statistics of the connectors, fixed parts, and guide parts used by the current harness.
- Content Table: Lists the harness objects at any point, you can choose whether to display the

hierarchy.

- Global Wiring Table: Consistent with the global wiring table in 3D, the actual 3D electrical connection information is collected.

【What can be done】:

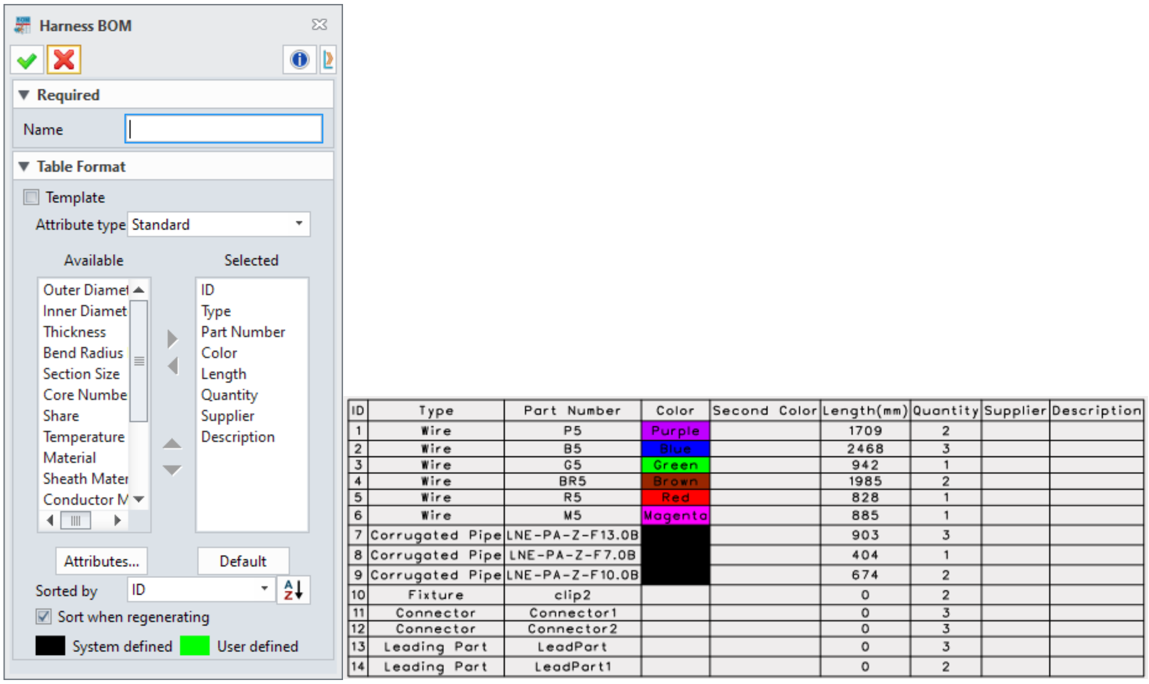
- ✓ Various tables can be added at any position in the nailboard.
- ✓ All table attributes are derived from the 3D model and can be refreshed along with changes to the 3D model.

【Note】:

- If there is no global wiring table in 3D, the nailboard will also fail to output it.

【Example】:

1. In any nailboard, select the target type of table as desired, and then place the table in the appropriate position.


Harness BOM
Required
Name
Table Format
Template
Available
Selected
Outer Diamet
Inner Diamet
Thickness
Bend Radius
Section Size
Core Numbe
Share
Temperature
Material
Sheath Mater
Conductor M
Attributes...
Default
System defined
User defined

☐ Template

Attribute type: Standard

Selected: ID, Type, Part Number, Color, Length, Quantity, Supplier, Description

Sorted by: ID

☑ Sort when regenerating

ID	Type	Part Number	Color	Second Color	Length(mm)	Quantity	Supplier	Description
1	Wire	P5	Purple		1709	2		
2	Wire	B5	Blue		2468	3		
3	Wire	G5	Green		942	1		
4	Wire	BR5	Brown		1985	2		
5	Wire	R5	Red		828	1		
6	Wire	M5	Magenta		885	1		
7	Corrugated Pipe	LNE-PA-Z-F13.0B	■		903	3		
8	Corrugated Pipe	LNE-PA-Z-F7.0B	■		404	1		
9	Corrugated Pipe	LNE-PA-Z-F10.0B	■		674	2		
10	Fixture	clip2			0	2		
11	Connector	Connector1			0	3		
12	Connector	Connector2			0	3		
13	Leading Part	LeadPart			0	3		
14	Leading Part	LeadPart1			0	2		

【Where is it】:

Nailboard >> BOM >> Harness BOM/Wires Table/Protection Table/Connectable Object

Table/Standard Part Table/Content Table/Global Wiring Table

4.5.3.5 ★New Dimension Function

For nailboards of which the layout has been adjusted, dimensions need to be marked. The Harness module of ZW3D 2025 provides two types of dimension methods, including manual dimension and automatic dimension.

- Harness Dimension: For manual dimension, you need to select the two points of harness dimension and the position that places the size.
- Harness Auto Dimension: For automatic dimension, you can select the point that needs to be labeled to dimension the two adjacent points.

【What can be done】:

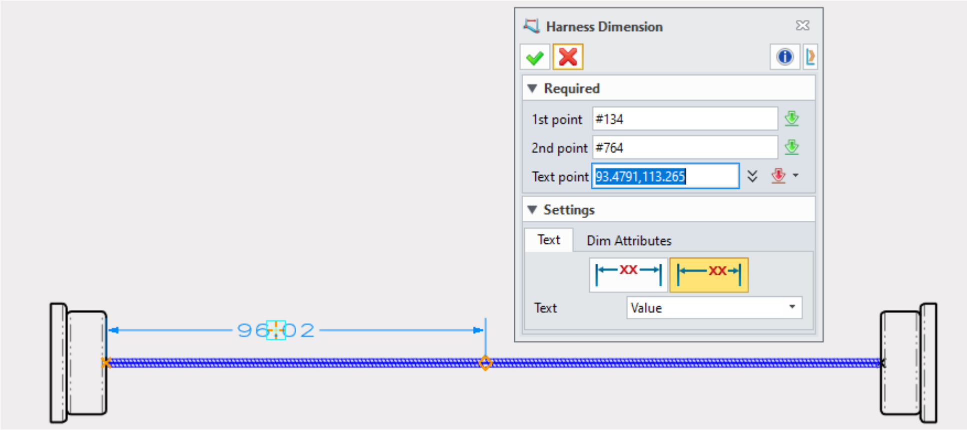
- ✓ Dimension the actual length of the harness in the nailboard, namely the length including the relaxation amount.

【Note】:

- Cannot dimension the distance between harnesses.

【Example】:

1. Open any nailboard, use the **"Harness Dimension"** command, select the two points to be dimensioned and the size placement position, and you can complete the line length annotation.
2. Open any nailboard, use the **"Harness Auto Dimension"** command to filter the points that need to be dimensioned, and after confirming, all harness lengths can be dimensioned.



1st point #134

2nd point #764

Text point 93.4791,113.265

Text Value

【Where is it】:

Nailboard >> Annotation >> Harness Dimension/ Harness Auto Dimension

4.5.3.6 New Harness Number Inquire Function

When the first and last position points of multiple harnesses are consistent, the nailboard will be displayed as a coincident line. If the harnesses are nested in multiple layers, only the outermost line of the harnesses will be displayed in the nailboard. In this case, you can run the " Harness Number " command to query the number of harnesses contained in this segment of the nailboard and quickly identify the information of harness quantity.

【What can be done】:

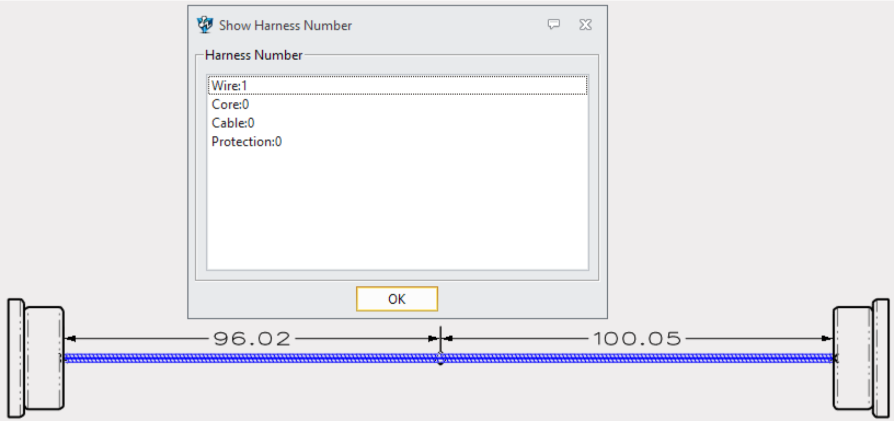
✓ Identify the number of harnesses contained in any segment of nailboard.

【Note】:

- Identify the specific name of the harness in the harness segment.

【Example】:

1. Open any nailboard, select any harness segment, and then click the "Show Harness Number" command to display the specific number of each type of harness.



Wire:1
Core:0
Cable:0
Protection:0

【Where is it】：

Nailboard >> Inquire >> Harness Number

4.6 Electrode Design

4.6.1 Electrode Processing Attribute Improvement（ZW3D 2025 SP）

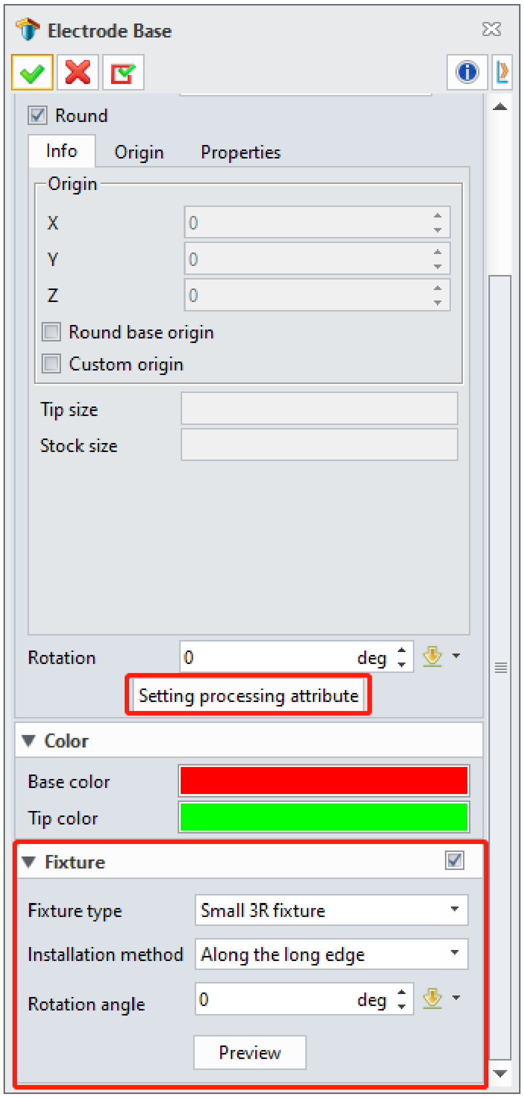
In the process of electrode design, it is necessary to endow the electrode with many processing attributes for the subsequent electrode processing and discharge procedures.

In the version of ZW3D 2025 SP, the columns of electrode processing attribute and fixture setting have been added.

【What can be done】：

Support electrode processing attribute setting and electrode fixture selection.

- ✓ Setting processing attribute: Including Base information、Process requirements、Explain、Discharge information setting.
- ✓ Fixture: Support the selection of fixture types, fixture installation method and fixture preview.


Electrode Base
Info
Origin
Properties
Setting processing attribute
Color
Preview

☑ Round

Origin

X 0

Y 0

Z 0

☐ Round base origin

☐ Custom origin

Tip size

Stock size

Rotation 0 deg

Base color 🟥

Tip color 🟩

Fixture ☑

Fixture type: Small 3R fixture

Installation method: Along the long edge

Rotation angle: 0 deg

【Where is it】：

Part/Assembly Environment >> Electrode >> Electrode Tools >> Electrode Base/Electrode Attribute

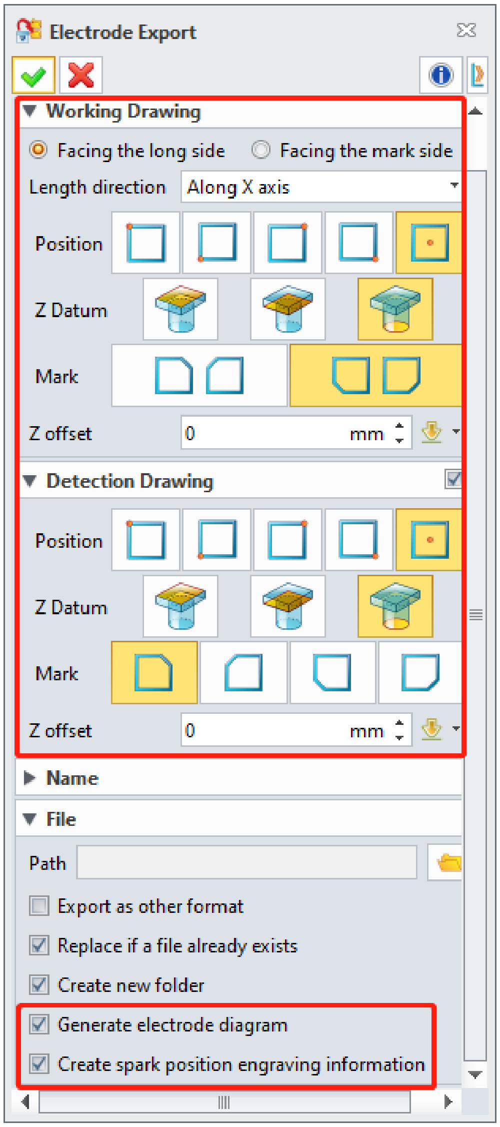
4.6.2 Strengthen the Electrode Export Function

When the electrode is led out to the next process, not only the 3D processing drawings of the electrode are needed, but also the 2D electrode diagram and detection drawings of the electrode are needed.

In the version of ZW3D 2025 SP, The Electrode Export function adds the derivation of electrode diagram, detection drawing file and spark position engraving information.

【What can be done】

- ✓ Optimize the export of working drawings: According to the requirements of different customers, the export of Working drawings includes the following two ways: "Facing the long side " and "Facing the mark side".
- ✓ Increase the export of 3D drawing files for electrode detection.
- ✓ Add the options of "generate electrode diagram" and "create spark position engraving information" to support users to export electrode diagram and electrode spark position engraving information.


Electrode Export

Working Drawing
● Facing the long side ○ Facing the mark side
Length direction: Along X axis
Position
Z Datum
Mark
Z offset: 0 mm

Detection Drawing ☑
Position
Z Datum
Mark
Z offset: 0 mm

Name

File
Path
☐ Export as other format
☑ Replace if a file already exists
☑ Create new folder
☑ Generate electrode diagram
☑ Create spark position engraving information

【Where is it】:

Part/Assembly Environment >> Electrode >> Electrode Tools >> Electrode Export

5 CAM

5.1 QM Module

5.1.1 Corner Finish Improvement

The Corner Finish operation is commonly used to machine corners formed between non tangent surfaces of parts. ZW3D 2025 has optimized the Corner Finish operation, making the detection range more accurate and reducing the occurrence of local missing and broken tool paths. The corner cleaning tool path is smoother at the corners.

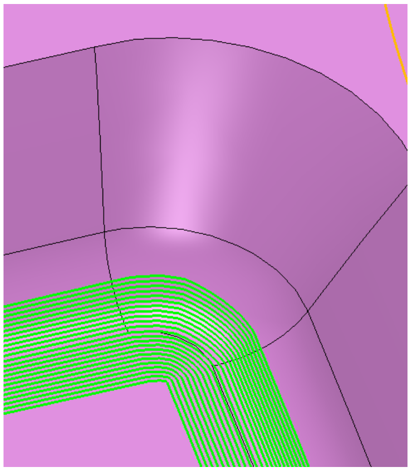
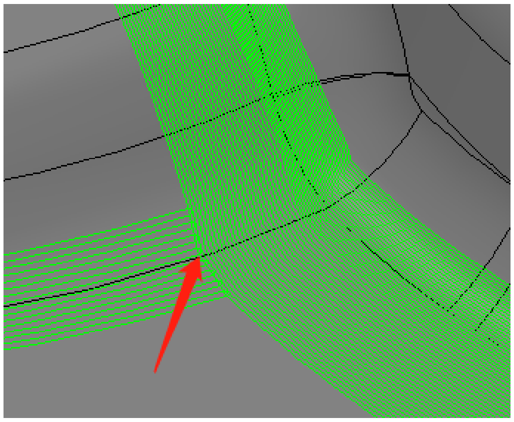
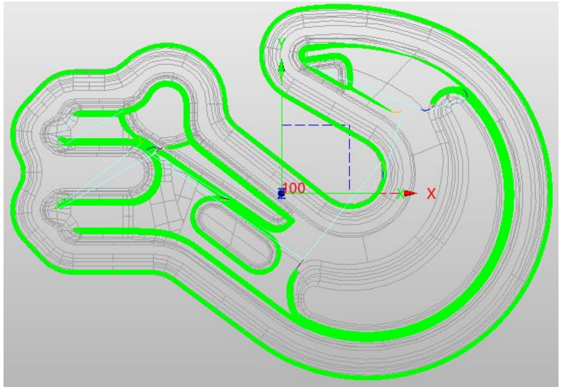
【What can be done】：

- ✓ Make the Corner Finish toolpath smoother.
- ✓ When using flat tools and cattle nose tools for processing, the missing and overlapping of toolpaths are reduced.
- ✓ Reduce the tool shift distance by adding any option in Steep area >> Steep Cut sequence.

【Notes】：

- Remove parameters in the Corner Finish Filter>>Angular.
- Remove parameters in the Path Setting>>Corner Control.
- Remove parameters Path Setting >>Cutting Control>>Cut Order in the Corner Finish operation.

【Example】：

Smoother toolpath corner	
The small overlap area of the toolpath makes it more efficient while ensuring complete machining	
Accurate corner finish detection, continuous toolpath, no missing toolpath	

【Where is it】:

3x Quick Ribbon>>Corner Finish

5.1.2 Pencil Cut Improvement

The Pencil Cut operation can create a single trajectory and multi-trajectory tool paths and remove corners between non tangent surfaces of parts. ZW3D 2025 has optimized the Pencil Cut operation, making the detection range more accurate and reducing the occurrence of local missing and disconnected tool paths.

【What can be done】:

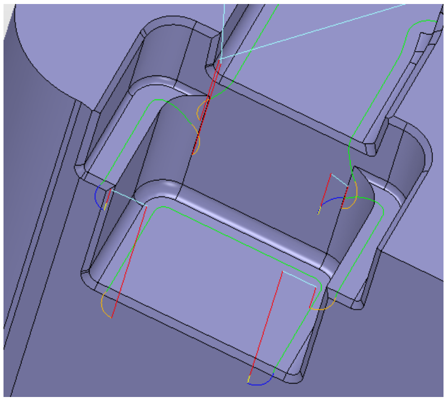
- ✓ New steep angle division function, support to set the cutting direction of the flat area and steep area respectively.
- ✓ New steep cutting direction setting, providing 3 ways including Any, High to Low, Low to High.

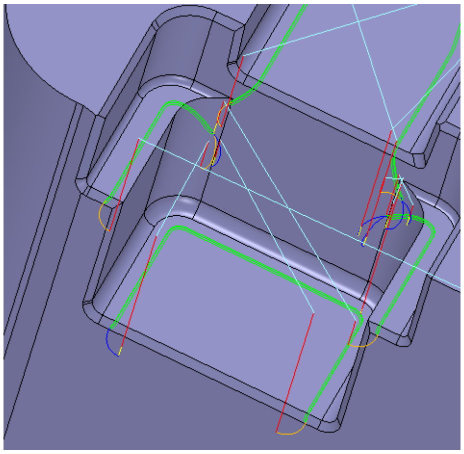
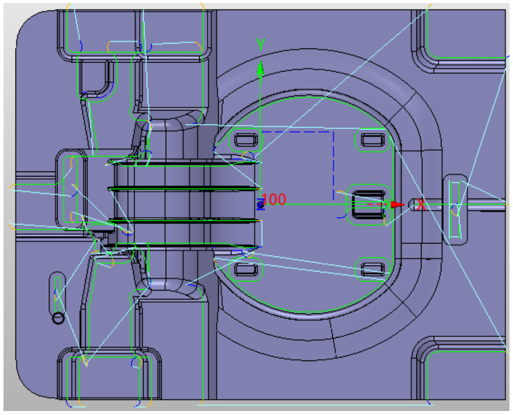
【Notes】:

- Remove parameters in Filter>>Angular
- Remove parameters in Path Setting>>Corner Control
- Modify parameters as Any, Conventional, and Climb milling in

  Path Setting>>Cutting Control>>Cut Direction
- When the cutting layer number is entered in the multi-paths cleaning process, the tool path with the corresponding layer number is created on both sides respectively.

【Example】:

Single path corner finish	

Multiple path corner finish	
Accurate corner finish detection, no missing toolpath or disconnection	

【Where is it】:

3x Quick Ribbon>>Pencil

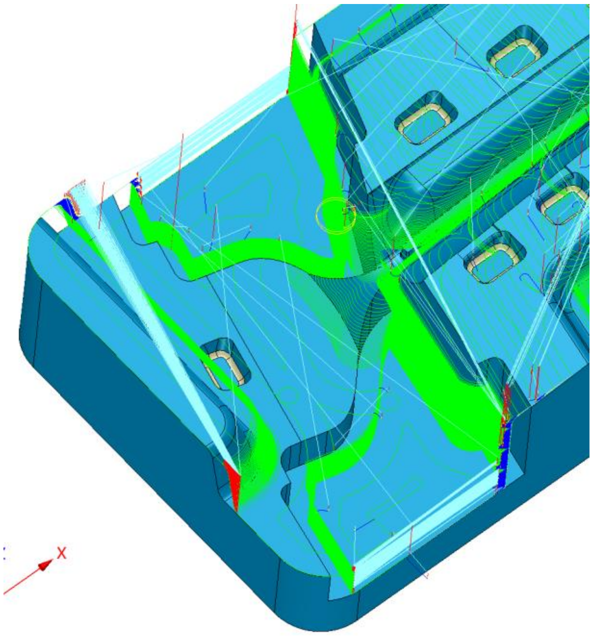
5.1.3 Rest Roughing Improvement

ZW3D 2025 has improved rest milling capability. Through the parameters of Blank Stock and Min Rest Thickness, the function can effectively improve the continuity of the Rest Rough milling tool path, reduce trivial tool paths, and improve machining efficiency.

【What can be done】:

- ✓ Set blank margin parameters to reduce tool lifting.
- ✓ Set the minimum residual thickness parameter to reduce trivial toolpaths.

【Example】：



【Where is it】：

3x Quick Ribbon>>Rough Smooth Flow/ Rough Offset 2D/ Rough Lace

5.1.4 Zlevel Undercut Improvement

ZW3D 2025 has optimized the Undercut function of the Zlevel operation, improved the efficiency calculations, and generated more complete and reliable tool paths.

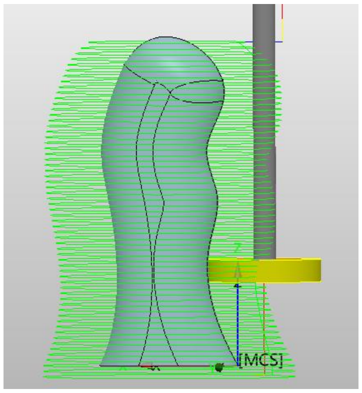
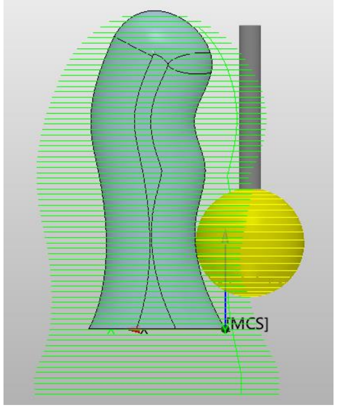
【What can be done】：

- ✓ The calculation efficiency of the tool path is 5 times higher than that of the previous version.
- ✓ Support the use of T-shaped and spherical tools to calculate the root cutting toolpath.

【Note】：

- Only horizontal arc and horizontal straight line are supported.

【Example】：

T-cutter processing, complete and reliable tool path	
Spherical tool processing, complete and reliable tool path	

【Where is it】:

3x Quick Ribbon>>ZLevel >>Path Setting>>Allow Undercutting

5.1.5 Innovation in Check Collision

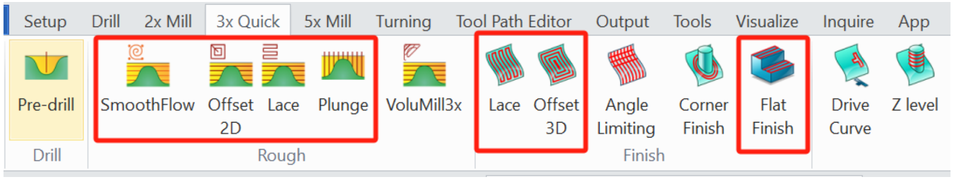
Tool and Holder collision is commonly used in machining scenarios with large height differences. When this function is activated, the system will automatically delete the tool path that collides with the clamping. In ZW3D 2025, the Check Collision function greatly improves calculation efficiency and accuracy.

【What can be done】:

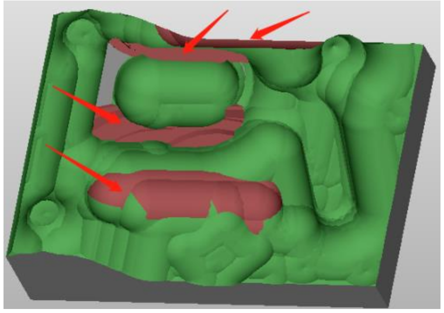
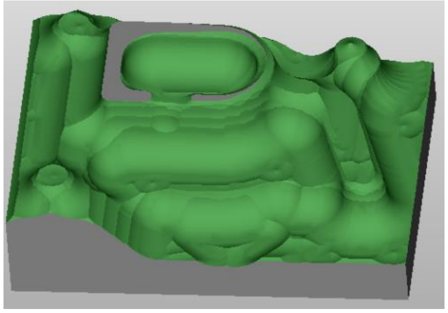
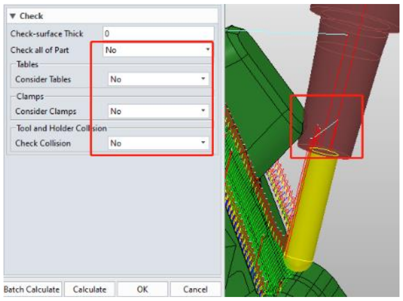
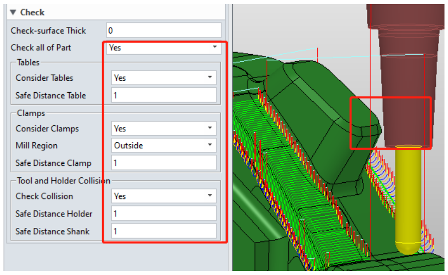
- ✓ The calculation efficiency of the toolpath is 2 times higher than that of the previous version.
- ✓ Support the setting of the holder gap and tool shank gap.
- ✓ The roughing process supports both check model and blank and model-only Settings.
- ✓ By default, the finishing operation only supports inspection models.

【Note】:

- The gripper collision function innovation applies only to the following operations:


Setup
Drill
2x Mill
3x Quick
5x Mill
Turning
Tool Path Editor
Output
Tools
Visualize
Inquire
App
Pre-drill
SmoothFlow
Offset 2D
Lace
Plunge
VoluMill3x
Lace
Offset 3D
Angle Limiting
Corner Finish
Flat Finish
Drive Curve
Z level
Drill
Rough
Finish

【Example】:

When collision check is off (collision overcut exists)	When collision check is on (no collision overcut)
	
 ▼ Check Check-surface Thick 0 Check all of Part No Tables Consider Tables No Clamps Consider Clamps No Tool and Holder Collision Check Collision No Batch Calculate Calculate OK Cancel	 ▼ Check Check-surface Thick 0 Check all of Part Yes Tables Consider Tables Yes Safe Distance Table 1 Clamps Consider Clamps Yes Mill Region Outside Safe Distance Clamp 1 Tool and Holder Collision Check Collision Yes Safe Distance Holder 1 Safe Distance Shank 1

【Where is it】:

3x Quick >> SmoothFlow / Offset 2D / Rough Lace / Plunge / Finish Lace / Flat Finish >>Limiting Parameters >> Check

5.2 2-Axis Module

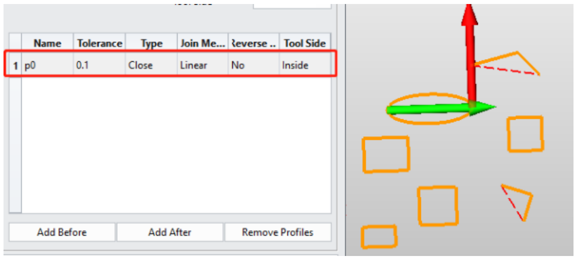
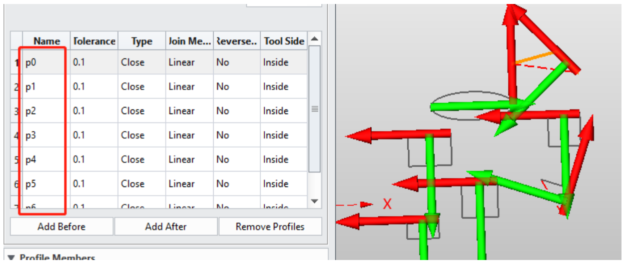
5.2.1 Added Auto Grouping for Profile Feature

In ZW3D 2025, profile features support the automatic grouping function, the system can group the objects selected by the box, and the starting edge and the processing side are displayed with arrows (The red arrow indicates the starting edge and the processing direction, and the green arrow indicates the tool side).

【What can be done】：

- ✓ When creating a profile feature, the user can identify the starting position of the tool path (The red arrow indicates the starting edge of the tool path).

【Example】：

When automatic grouping is off, all contours are grouped into one group	When automatic grouping is on, it is automatically grouped according to whether the contours are connected (connected into a group, grouped in turn).
	

	Name	Tolerance	Type	Join Me...	Reverse ..	Tool Side
1	p0	0.1	Close	Linear	No	Inside

	Name	Tolerance	Type	Join Me...	Reverse..	Tool Side
1	p0	0.1	Close	Linear	No	Inside
2	p1	0.1	Close	Linear	No	Inside
3	p2	0.1	Close	Linear	No	Inside
4	p3	0.1	Close	Linear	No	Inside
5	p4	0.1	Close	Linear	No	Inside
6	p5	0.1	Close	Linear	No	Inside
7	p6	0.1	Close	Linear	No	Inside

【Where is it】：

CAM Ribbon>> Profile Feature >>Auto Group

5.2.2 Optimize Cutting Start Point of 2x Operation

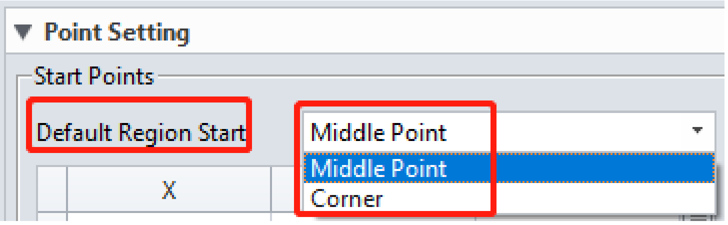
In the Profile, Ramp, Thread process of ZW3D 2025, the tool path starting point setting function has been optimized and the new Detection Range function has been added, which facilitates users to set the starting point of the processing area.

【What can be done】:

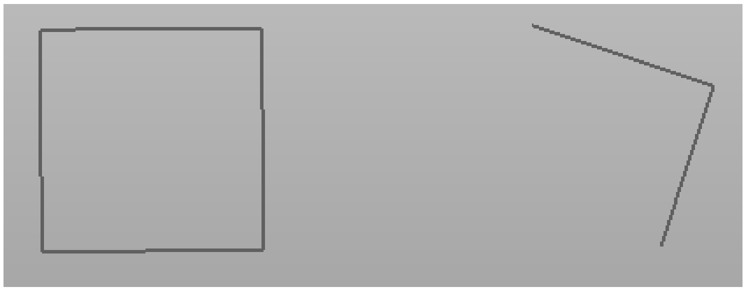
- ✓ Optimize the area start function, by which users can change the starting position of the cutting tool path.
- ✓ Add the Detection Range function, which supports setting the starting points of different closed profiles without affecting each other.

【Note】:

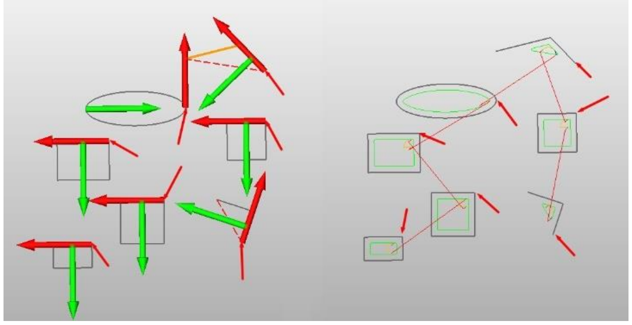
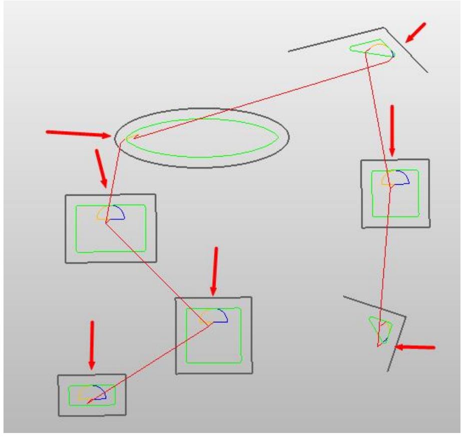
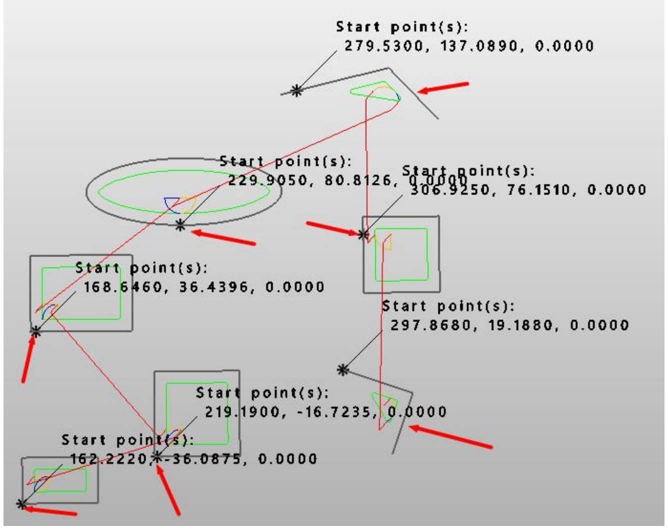
- Detection Range indicates the influence range of the setting of the processing starting point. If the setting is too large, the processing starting point of other regions will be affected, and the processing starting point of other regions will be the closest to the setting point. If other areas cannot be affected, the default area starting point is used as the processing starting point for this area.



- The set starting point is only effective for the geometrically closed area and has nothing to do with the open and closed Settings in the feature. When it is a geometrically open area, the starting point of processing is the default starting point of the region (figure left is the geometrically closed area, figure right is the geometrically open area).



【Example】:

Default Region Start=Corner (Start cutting at the beginning of the red arrow)	
Default Region Start=Middle Point (Start cutting at the profile center where the red arrow locates)	
The geometric closed area is the starting point of the set processing, and the geometric open area is the starting point of the default area.	 Start point(s): 279.5300, 137.0890, 0.0000 Start point(s): 229.9050, 80.8126, 0.0000 Start point(s): 306.9250, 76.1510, 0.0000 Start point(s): 168.6460, 36.4396, 0.0000 Start point(s): 297.8680, 19.1880, 0.0000 Start point(s): 219.1900, -16.7235, 0.0000 Start point(s): 162.2220, -36.0875, 0.0000

【Where is it】:

2x Mill Ribbon>> Profile Operation/ Ramp Operation/ Thread Operation >> Toolpath Setting >> Point Setting

5.2.3 Profilecut Add Extend and Reduce Function

In ZW3D 2025, the profile process can extend or shorten the two ends of the tool path to ensure that the parts are processed completely or avoid the key surface, and the specific parameters are adjusted according to the actual processing needs.

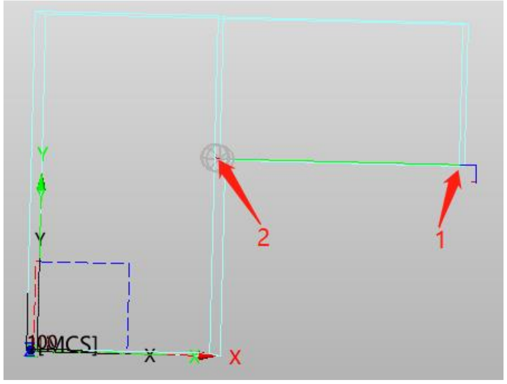
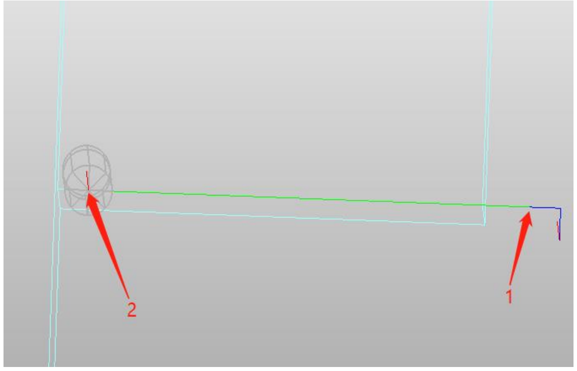
【What can be done】:

- ✓ Support inputting absolute value and tool diameter percentage to control tool track extension or shortening distance.

【Note】:

- This feature is only available for open profiles.

【Example】:

Before using the tool path extension and shortening function, the starting point of the cutting tool path 1 is just at the contour starting point, and the direct feed speed to the contour starting point will affect the processing quality if the feed speed is not appropriate; Retracting point 2 at the end of the profile line did not avoid the critical surface resulting in overcutting.	After using the tool path extension and shortening function, the cutting tool rail is extended to the starting point 1 to ensure that the cutting speed is fed close to the part before the cutting tool part; The position of the retracting tool is shortened to the retracting tool point 2 to ensure that the key surface is avoided, and no overcut is generated.
	

【Where is it】:

2x Mill Ribbon>> Profile Operation >> Limit Parameter >>Boundary>> Profile Starte/End Position

5.2.4 Profilecut Add Region Sequencing Function

In ZW3D 2025, the sorting function has been added in the two-axis profile operation, and users can select different sorting methods to generate the corresponding tool paths according to the processing needs, so that the tool paths are arranged more neatly, and the processing time is reduced.

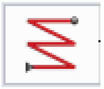
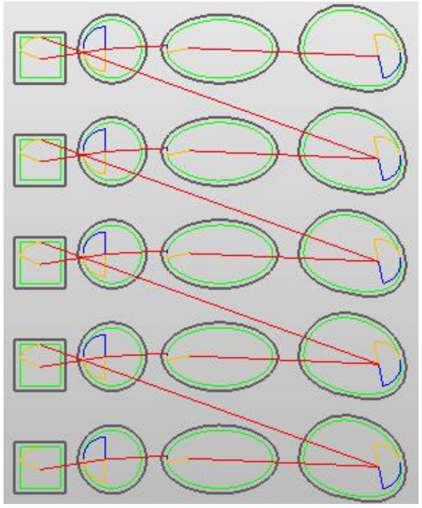
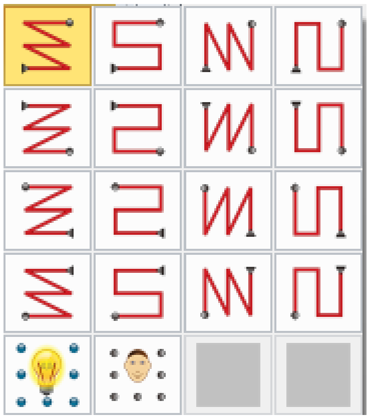
【What can be done】：

- ✓ A total of 18 sequencing modes are supported.

【Note】：

- The sorting of reference tools and reference processes is not supported.

【Example】：

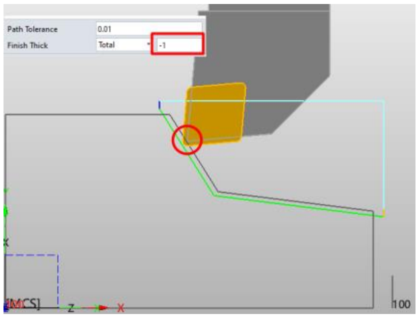
Case Sorting Method	Case Actual Toolpath	Function supports sorting methods
		

5.3 Turning & Drilling Modules

5.3.1 Turning Support Negative Thick

ZW3D 2025 supports setting negative margins in the turning operation, to satisfy users to eliminate machining margins caused by machine tool system errors.

【Example】:



Path Tolerance 0.01

Finish Thick Total -1

【Where is it】:

Turning >> End/Rough/Finish/Groove Operation >> Tolerances and Steps >> Tolerances and Allowances

5.3.2 Turning slot machining cutting strategy options adjustment

To prevent the user from using the turning groove machining process, the tool path generated by the two cutting strategies of "one-way" and "Zig" is the same when the "deep cutting" is not opened, and the misunderstanding that the two cutting strategies are repeated has been adjusted to this option.

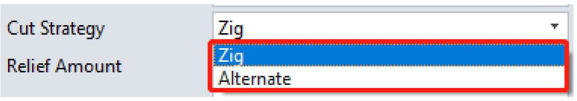
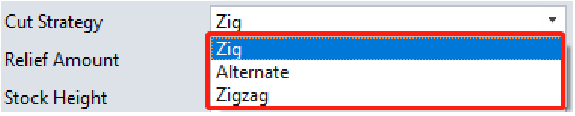
【What can be done】:

- ✓ You can only select Zig and Alternate as cut strategies when the Level Control is off.
- ✓ You can select Zig, Alternate, and Zigzag as cut strategies when the Level Control is enabled.

【Note】:

- Zigzag can be used when the Level Control is enabled.

【Example】:

Level Control is closed	Enable Level Control
 Cut Strategy Zig Relief Amount Zig Alternate	 Cut Strategy Zig Relief Amount Stock Height Zig Alternate Zigzag

【Where is it】:

Turning Ribbon>>Turn Grooving Operation Definition>>Toolpath Setting>>Cut Strategy

5.3.3 Add Circular Shrank for Turning Tool

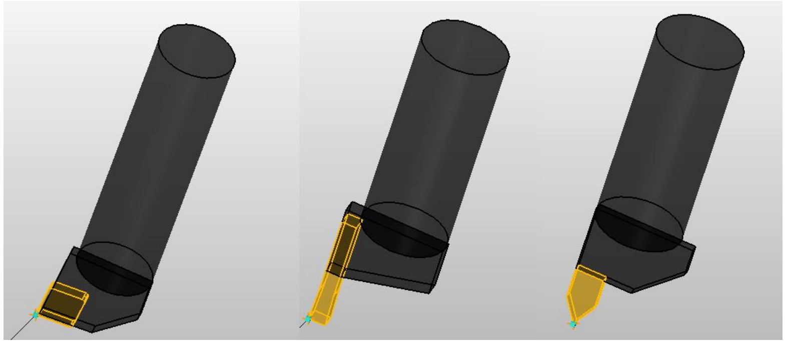
ZW3D 2025 has added a circular shrank to the definition of turning tools to meet the actual inner profile turning processing scenarios.

【What can be done】:

- ✓ Support creating round shanks.

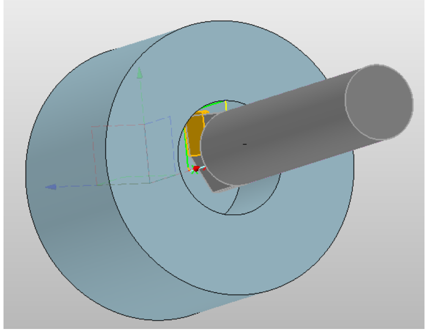
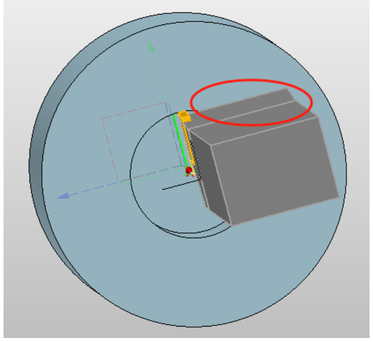
【Example】:

Round shank of different types of turning tools:



Comparison of different shank applications in special scenarios:

A circular shank machining a smaller inner circle (no overcut)	A square shank machining a smaller inner circle (overcut)

【Where is it】：

Setups >> Tool >> Turning >> Holder >> Shank Type

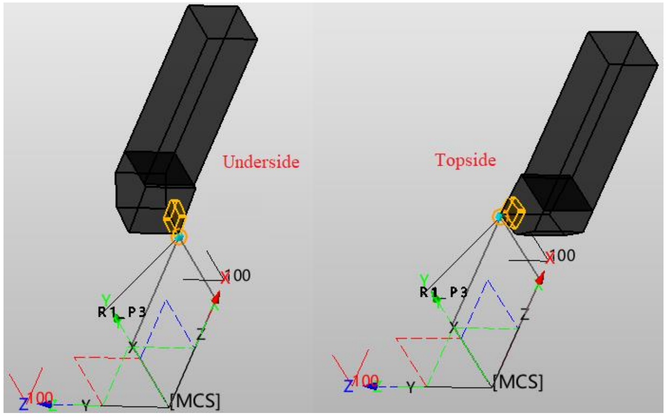
5.3.4 Add Insert Position for Turning Tool

In ZW3D 2025, a new installation position function has been added to the turning tool, which can change the installation position of the tool by setting the top side or the bottom side to better meet the actual operation scene.

【What can be done】：

- ✓ Support changing the tool position by setting the top side or underside.

【Example】：


Underside
100
R1_P3
X
Y
Z
100
[MCS]
Topside
100
R1_P3
X
Y
Z
100
[MCS]

【Where is it】：

CAM Ribbon>> Tool >> Turning tool >> Insert >> Insert position

5.3.5 Add Region Sequencing for Drill

ZW3D 2025 has added a sorting function in the drilling process, and users can choose different sorting methods to generate corresponding tool tracks, so that the tool tracks in different areas are arranged more neatly.

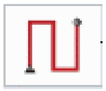
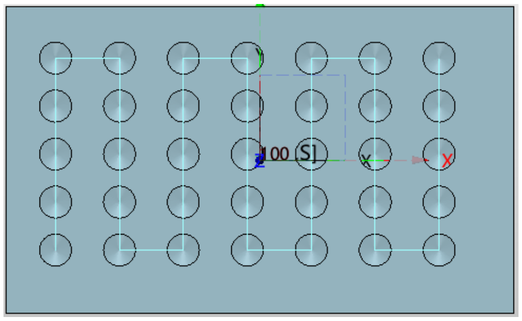
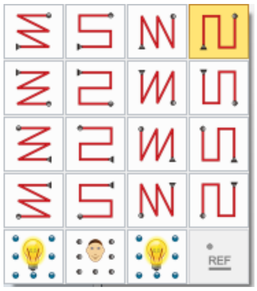
【What can be done】:

- ✓ A total of 19 sequencing modes are supported and the corresponding toolpath is generated according to the specified sorting mode.
- ✓ Support strategy function.

【Note】:

- The "Minimum Distance" and "Minimum Distance/ Axis" icons are the same, so be careful when using them.

【Example】:

Case Sorting Method	Case Actual Toolpath	Sorting Method Supported by the Function
		

【Where is it】:

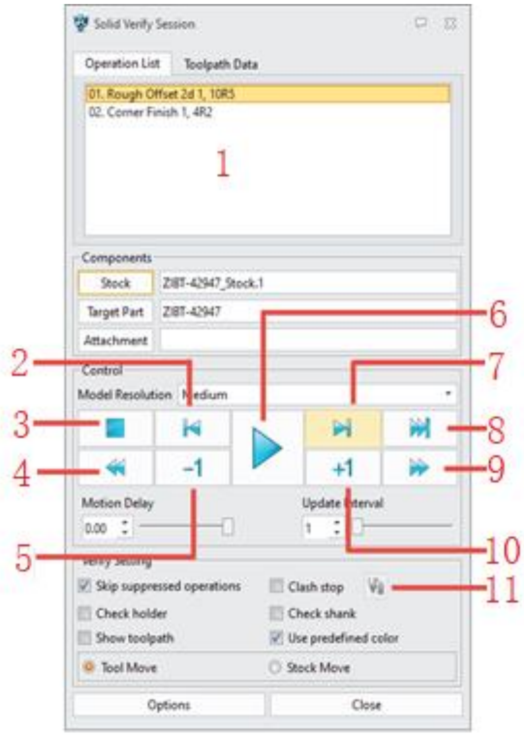
Drilling Ribbon>> Center/STD_Drill/Peck/Chip/Ream/Bore/Fine bore/Counterbore/Countersink/ Tap >> Toolpath Setting >>Region Sequencing

CAM Tree Table >> Strategy >> Hole Strategy manager >> Hole processing parameters >> Region Sequencing

5.4 Platform Matching Functionalities

5.4.1 ★Overall Innovation of Solidverify

Physical simulation is often used to check the rationality of programming and the safety of tool paths. ZW3D 2025 has made an overall innovation to the physical simulation function, redesigned the entire user interface, and optimized the user experience. A series of new process-by-process simulation operation functions have been added, and the efficiency of physical simulation has been improved.


Solid Verify Session
Operation List
Toolpath Data
1
Components
Control
-1
+1
2
3
4
5
6
7
8
9
10
11
Verify Setting
Options
Close

01. Rough Offset 2d 1, 10R5
02. Corner Finish 1, 4R2

Stock: ZIBT-42947_Stock.1
Target Part: ZIBT-42947
Attachment:
Model Resolution: Medium
Motion Delay: 0.00
Update Interval: 1

☑ Skip suppressed operations
☐ Clash stop
☐ Check holder
☐ Check shank
☐ Show toolpath
☑ Use predefined color
● Tool Move
○ Stock Move

1. Move forward and backward step by step by clicking the step in the list.
2. Step by step.
3. One-click to the beginning.
4. Back up by tool path type.
5. Retreat at tool point.
6. Start simulation.

7. Move operation in units of process.
8. One-click to the end.
9. Advance by tool path type.
10. Advance by tool position point.
11. Display collision information.

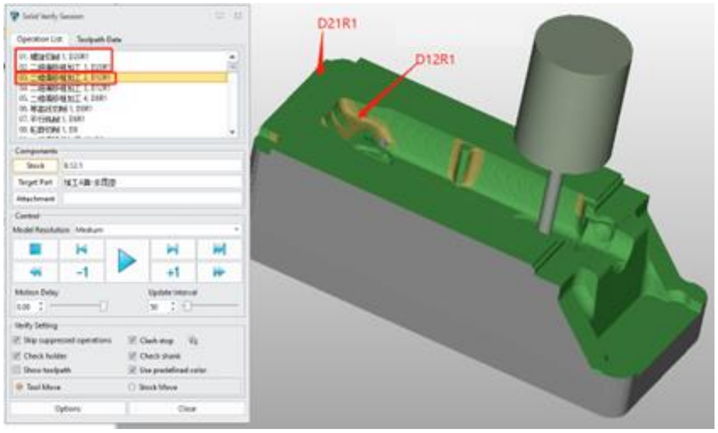
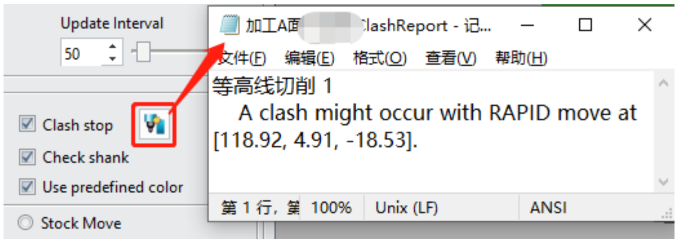
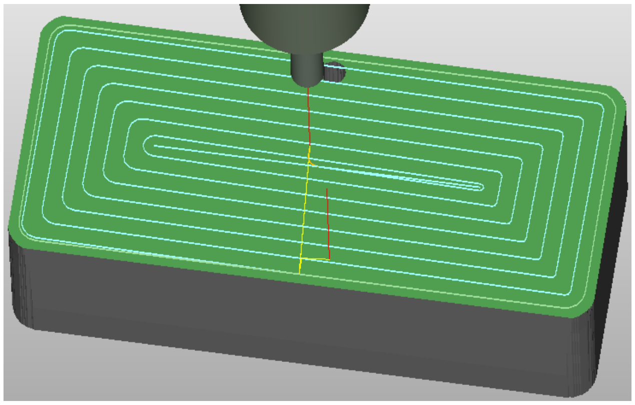
【What can be done】:

- ✓ The operation list controls the simulation forward and backward.
- ✓ When the simulation is rolled back to the previous process, the blank is rolled back accordingly.
- ✓ The color of the removal during simulation changes with the change of the tool.
- ✓ Support exporting collision list files.
- ✓ The tool path color of the physical simulation follows the settings in the operation.
- ✓ The collision detection of the tool holder and tool shank is carried out respectively.

【Note】:

- The simulated removal color does not support users setting and will be recycled.
- When entering the solid simulation, the billet model will be inspected and repaired, and the time will increase with the size of the model.
- Before overcut analysis and margin analysis, the part model and blank model will be inspected and repaired, and the time will increase with the size of the model.

【Example】:

The cut color changes with the tool	 D21R1 D12R1
Click to generate collision information	 Update Interval 50 ☑ Clash stop ☑ Check shank ☑ Use predefined color ○ Stock Move 加工A面[redacted]ClashReport - 记... 文件(F) 编辑(E) 格式(O) 查看(V) 帮助(H) 等高线切削 1 A clash might occur with RAPID move at [118.92, 4.91, -18.53]. 第 1 行，第 100% Unix (LF) ANSI
The tool path color of the physical simulation follows the settings in the process (the default cutting tool path color is green, here it is set to sky blue).	

5.4.2 Verify Add Tool Path Filling Display Function

The new tool path filling display function has been added to the middle line frame of ZW3D 2025, which can fill and display the planar tool path, so that users can quickly find the position of missing machining.

【What can be done】:

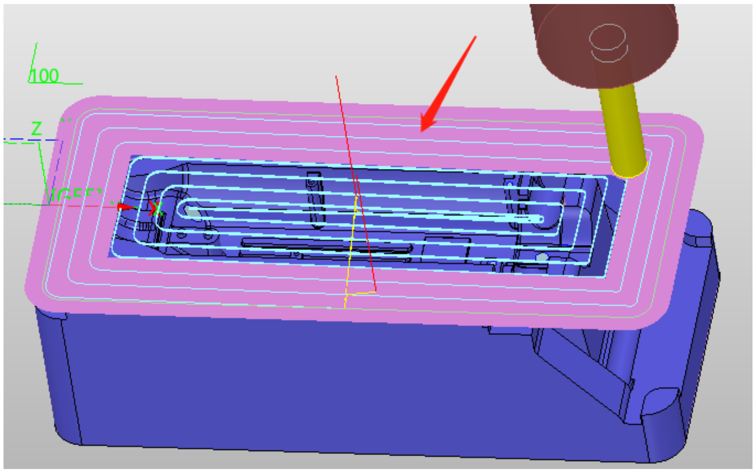
- ✓ Often used in two-axis machining scenarios, allowing users to quickly identify non-machining region.

【Note】:

- The fill effect only applies to the flat tool path.

【Example】:

The Wire frame simulation fills the simulated tool path (pink is the filling display in the figure).

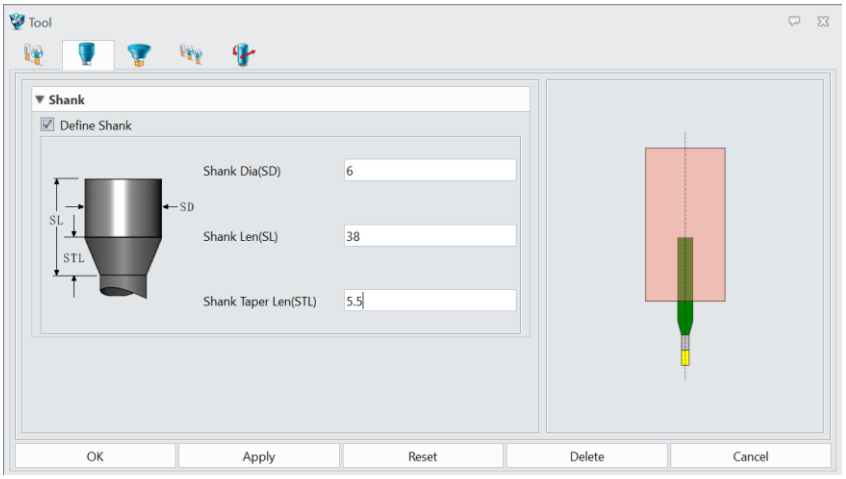


【Where is it】:

Output Ribbon>>Toolpath Verify>>Toolpath Expand

5.4.3 Adding shank to milling tools

ZW3D 2025 supports the addition of shank parameters in the definition of the milling cutter, and users can fully define the tool with a small diameter (diameter less than ϕ6mm). At the same time, on the clamping page, new tool offset length and extension length parameters, users can set the tool length.


Tool
SL
STL
SD
OK
Apply
Reset
Delete
Cancel

Shank

☑ Define Shank

Shank Dia(SD) 6

Shank Len(SL) 38

Shank Taper Len(STL) 5.5

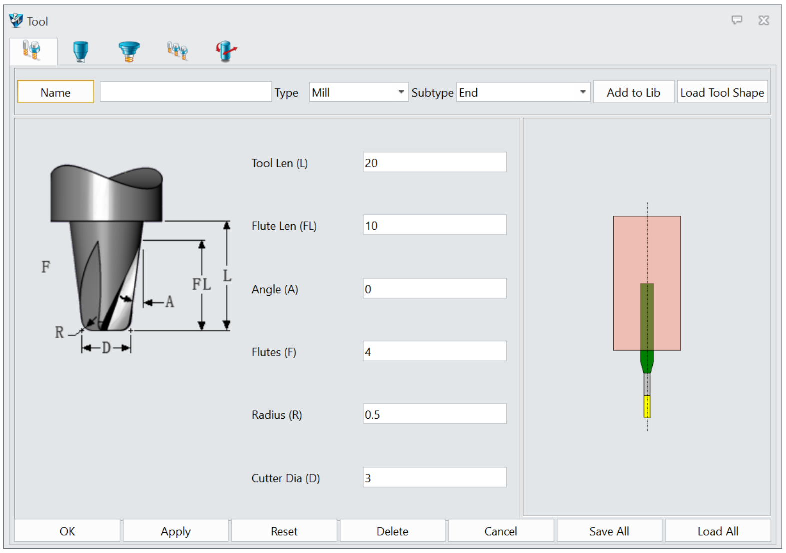
【What can be done】:

- ✓ Complete definition of the smaller diameter milling cutter.
- ✓ Set insertion tool length.
- ✓ Output tool loading length in shop documentation.

【Note】:

- Under normal circumstances, if tool diameter is less than ϕ6mm, it is necessary to define the tool shank.

【Example】:


Tool
Name
Add to Lib
Load Tool Shape
F
R
D
A
FL
L
OK
Apply
Reset
Delete
Cancel
Save All
Load All

Type Mill

Subtype End

Tool Len (L) 20

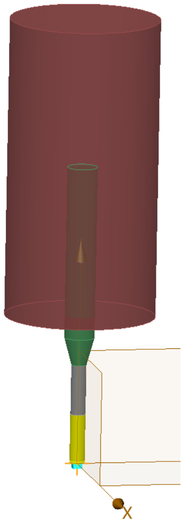
Flute Len (FL) 10

Angle (A) 0

Flutes (F) 4

Radius (R) 0.5

Cutter Dia (D) 3

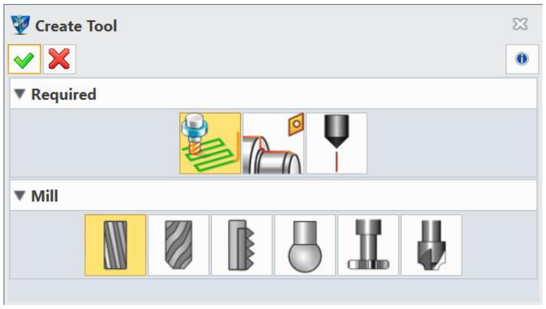


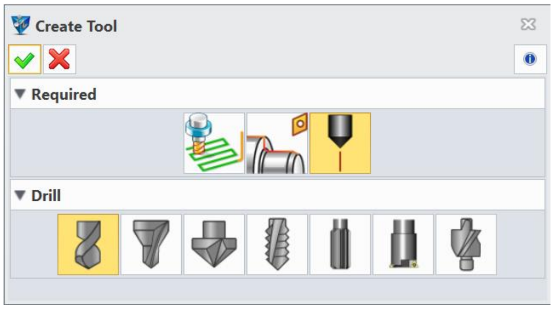
【Where is it】:

Setups >> Tool >>Tool Shank Page/Holder Page

5.4.4 Create Tool Interface Improvement

In ZW3D 2025, the Create tool entrance interface has added a variety of tool entrances, so that users can directly create the required tools. The icon is enlarged to make it easier for users to click. At the same time, when creating a tool, it supports the display of a three-dimensional model of the tool, so that users can intuitively judge the size of the tool.

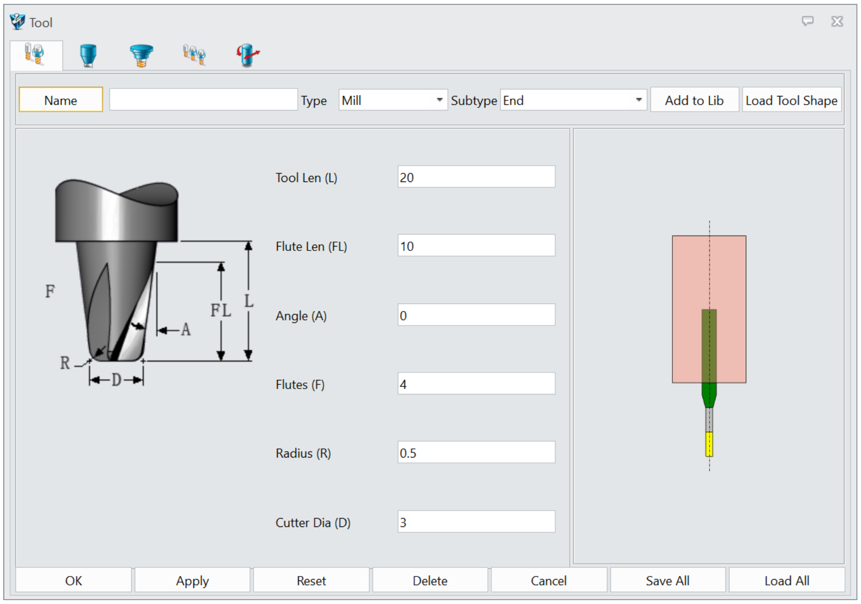




【What can be done】:

- ✓ Quickly select a tool to create.
- ✓ Directly display the 3D model.

【Example】:


Tool

Name

Type: Mill

Subtype: End

Tool Len (L): 20

Flute Len (FL): 10

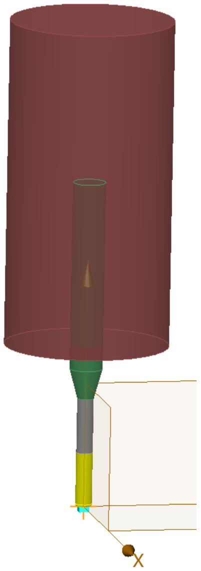
Angle (A): 0

Flutes (F): 4

Radius (R): 0.5

Cutter Dia (D): 3

Add to Lib
Load Tool Shape
F
FL
L
A
R
D
OK
Apply
Reset
Delete
Cancel
Save All
Load All



【Where is it】:

Setups >> Tool

5.4.5 T-Slot Tool Improvement

The transition diameter and transition length parameters of the original T-slot tools are removed, so that the parameters of the T-slot tools are closer to the real tools.

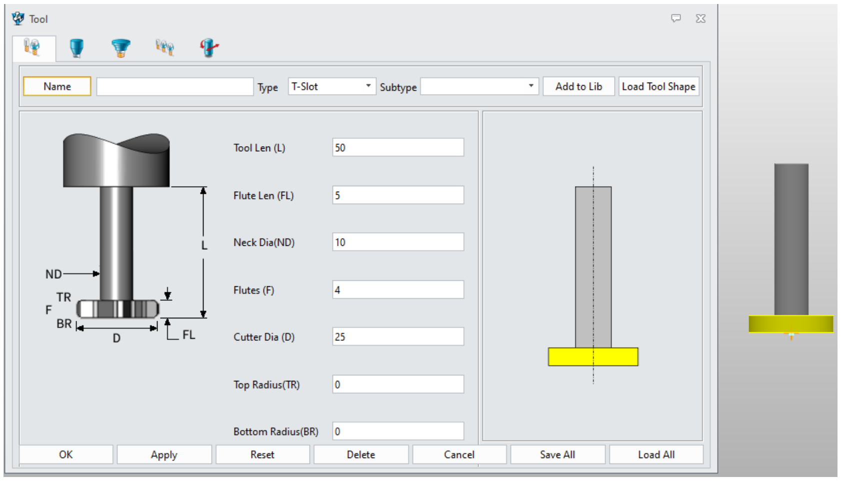
【What can be done】:

✓ Create a tool as required.

【Note】:

- The top radius and bottom radius cannot be greater than half of the blade length.

【Example】:


Tool
Name
Add to Lib
Load Tool Shape
ND
TR
F
BR
D
L
FL
OK
Apply
Reset
Delete
Cancel
Save All
Load All

Type: T-Slot
Subtype:
Tool Len (L): 50
Flute Len (FL): 5
Neck Dia(ND): 10
Flutes (F): 4
Cutter Dia (D): 25
Top Radius(TR): 0
Bottom Radius(BR): 0

【Where is it】:

Setups >>Tool >>T-slot Tool >> Shape Page

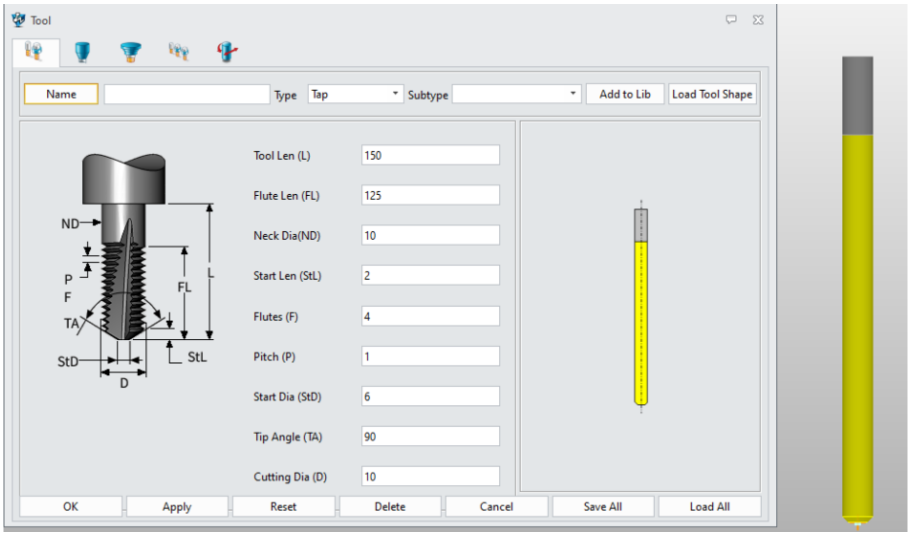
5.4.6 Tap Tool Improvement

The logical relationship between the tapping tool parameters has optimized, and the default tool parameters are adjusted when creating the tapping tool, which is closer to the actual application scenario.

【What can be done】:

- ✓ Create a tool as required.

【Example】:


Tool
Name
Add to Lib
Load Tool Shape
ND
P
F
TA
StD
D
FL
L
StL
OK
Apply
Reset
Delete
Cancel
Save All
Load All

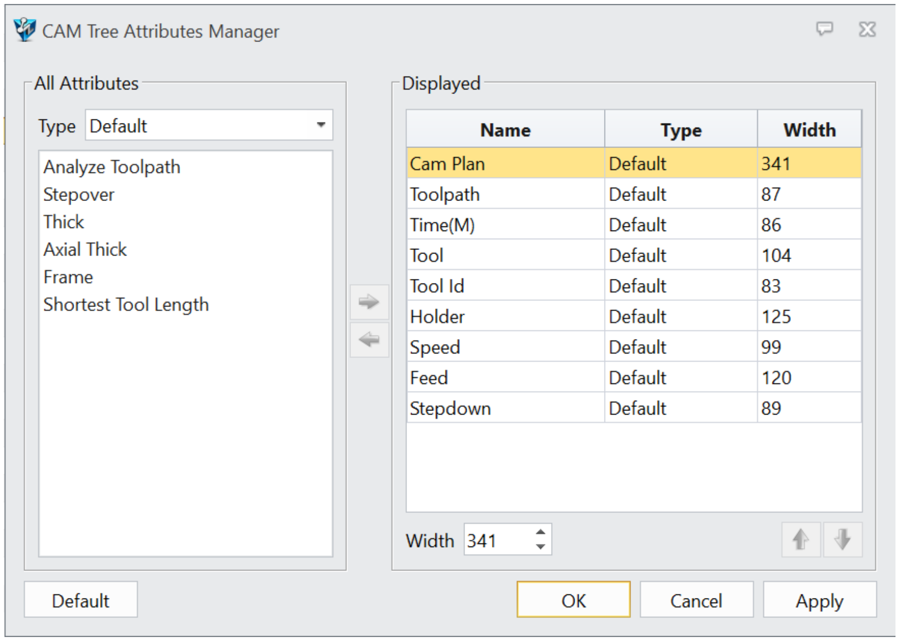
Type: Tap
Subtype:
Tool Len (L): 150
Flute Len (FL): 125
Neck Dia(ND): 10
Start Len (StL): 2
Flutes (F): 4
Pitch (P): 1
Start Dia (StD): 6
Tip Angle (TA): 90
Cutting Dia (D): 10

【Where is it】:

Setups>>Tool >>Thread >> Shape Page

5.4.7 CAM Tree Manager Improvement

In this release, the CAM Manager has been improved overall, with more process information displayed through tree table controls. Distinguish between the process state and the tool path state to alert the user that the process parameters and tool path have been modified. Optimize the refresh efficiency of the CAM manager and improve the efficiency of hiding and displaying tool tracks. At the same time supports keyboard control, users can select items through the keyboard's Up and Down keys, Left and Right buttons to expand and fold nodes, space bar to control the display and hiding of items.



CAM Tree Attributes Manager

All Attributes

Type: Default

- Analyze Toolpath
- Stepover
- Thick
- Axial Thick
- Frame
- Shortest Tool Length

Displayed

Name	Type	Width
Cam Plan	Default	341
Toolpath	Default	87
Time(M)	Default	86
Tool	Default	104
Tool Id	Default	83
Holder	Default	125
Speed	Default	99
Feed	Default	120
Stepdown	Default	89

Width: 341

Operation status type (as figure shows)	**Meaning**
✓	The tool path was successfully calculated
!	1. Operation newly created, not calculated the tool path 2. The operation parameters were modified after the tool

	path was successfully calculated, but the tool path was not recalculated.
Tool status (as figure shows)	**Meaning**
✓	The tool path was successfully calculated
✂	The tool paths are trimmed
🔒	The tool paths are locked
✗	Failed to be calculated the tool paths
❓	The tool path is not calculated

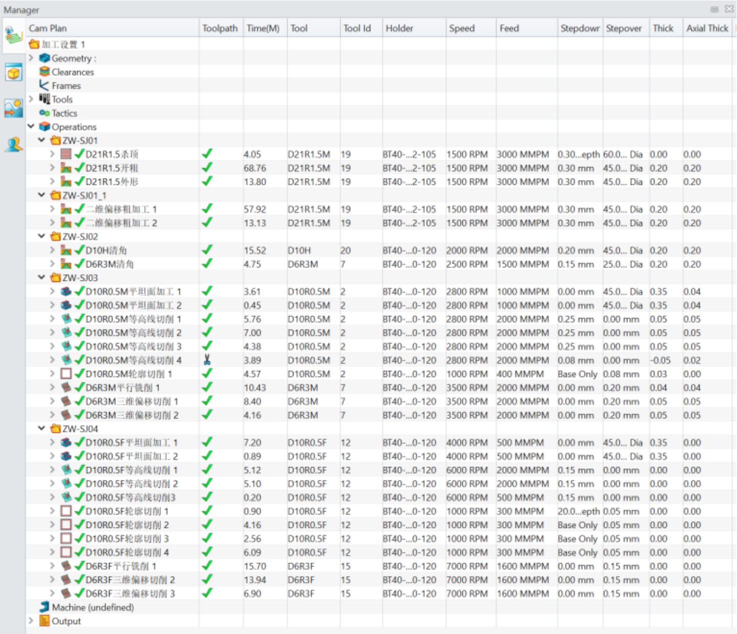
【What can be done】:

- The operation parameters of the process can be directly viewed in CAM management.
- Use the up and down keys of the keyboard to select items in CAM management.
- Expand and collapse items in CAM management from the left and right buttons of the keyboard.
- After modifying the process parameters, the process status changes to! But the state of the blade is unchanged.
- The display and hiding efficiency of the tool path have been significantly improved.

【Note】:

- If you cannot unhighlight an entry by clicking on a blank space in CAM Manager, you can unhighlight it by pressing Esc.

【Example】:



Manager

Cam Plan	Toolpath	Time(M)	Tool	Tool Id	Holder	Speed	Feed	Stepdowr	Stepover	Thick	Axial Thick
加工设置 1											
Geometry :											
Clearances											
Frames											
Tools											
Tactics											
Operations											
ZW-SJ01											
✓D21R1.5杀顶	✓	4.05	D21R1.5M	19	BT40-...2-105	1500 RPM	3000 MMPM	0.30...epth	60.0... Dia	0.00	0.00
✓D21R1.5开粗	✓	68.76	D21R1.5M	19	BT40-...2-105	1500 RPM	3000 MMPM	0.30 mm	45.0... Dia	0.20	0.20
✓D21R1.5外形	✓	13.80	D21R1.5M	19	BT40-...2-105	1500 RPM	3000 MMPM	0.30 mm	45.0... Dia	0.20	0.20
ZW-SJ01_1											
✓二维偏移粗加工 1	✓	57.92	D21R1.5M	19	BT40-...2-105	1500 RPM	3000 MMPM	0.30 mm	45.0... Dia	0.20	0.20
✓二维偏移粗加工 2	✓	13.13	D21R1.5M	19	BT40-...2-105	1500 RPM	3000 MMPM	0.30 mm	45.0... Dia	0.20	0.20
ZW-SJ02											
✓D10H清角	✓	15.52	D10H	20	BT40-...0-120	2000 RPM	2000 MMPM	0.20 mm	45.0... Dia	0.20	0.20
✓D6R3M清角	✓	4.75	D6R3M	7	BT40-...0-120	2500 RPM	1500 MMPM	0.15 mm	25.0... Dia	0.20	0.20
ZW-SJ03											
✓D10R0.5M平坦面加工 1	✓	3.61	D10R0.5M	2	BT40-...0-120	2800 RPM	1000 MMPM	0.00 mm	45.0... Dia	0.35	0.04
✓D10R0.5M平坦面加工 2	✓	0.45	D10R0.5M	2	BT40-...0-120	2800 RPM	1000 MMPM	0.00 mm	45.0... Dia	0.35	0.04
✓D10R0.5M等高线切削 1	✓	5.76	D10R0.5M	2	BT40-...0-120	2800 RPM	2000 MMPM	0.25 mm	0.00 mm	0.05	0.05
✓D10R0.5M等高线切削 2	✓	7.00	D10R0.5M	2	BT40-...0-120	2800 RPM	2000 MMPM	0.25 mm	0.00 mm	0.05	0.05
✓D10R0.5M等高线切削 3	✓	4.38	D10R0.5M	2	BT40-...0-120	2800 RPM	2000 MMPM	0.25 mm	0.00 mm	0.05	0.05
✓D10R0.5M等高线切削 4	✂	3.89	D10R0.5M	2	BT40-...0-120	2800 RPM	2000 MMPM	0.08 mm	0.00 mm	-0.05	0.02
✓D10R0.5M轮廓切削 1	✓	4.57	D10R0.5M	2	BT40-...0-120	1000 RPM	400 MMPM	Base Only	0.08 mm	0.03	0.00
✓D6R3M平行铣削 1	✓	10.43	D6R3M	7	BT40-...0-120	3500 RPM	2000 MMPM	0.00 mm	0.20 mm	0.04	0.04
✓D6R3M三维偏移切削 1	✓	8.40	D6R3M	7	BT40-...0-120	3500 RPM	2000 MMPM	0.00 mm	0.20 mm	0.05	0.05
✓D6R3M三维偏移切削 2	✓	4.16	D6R3M	7	BT40-...0-120	3500 RPM	2000 MMPM	0.00 mm	0.20 mm	0.05	0.05
ZW-SJ04											
✓D10R0.5F平坦面加工 1	✓	7.20	D10R0.5F	12	BT40-...0-120	4000 RPM	500 MMPM	0.00 mm	45.0... Dia	0.35	0.00
✓D10R0.5F平坦面加工 2	✓	0.89	D10R0.5F	12	BT40-...0-120	4000 RPM	500 MMPM	0.00 mm	45.0... Dia	0.35	0.00
✓D10R0.5F等高线切削 1	✓	5.12	D10R0.5F	12	BT40-...0-120	6000 RPM	2000 MMPM	0.15 mm	0.00 mm	0.00	0.00
✓D10R0.5F等高线切削 2	✓	5.10	D10R0.5F	12	BT40-...0-120	6000 RPM	2000 MMPM	0.15 mm	0.00 mm	0.00	0.00
✓D10R0.5F等高线切削3	✓	0.20	D10R0.5F	12	BT40-...0-120	6000 RPM	500 MMPM	0.15 mm	0.00 mm	0.00	0.00
✓D10R0.5F轮廓切削 1	✓	0.90	D10R0.5F	12	BT40-...0-120	1000 RPM	300 MMPM	20.0...epth	0.05 mm	0.00	0.00
✓D10R0.5F轮廓切削 2	✓	4.16	D10R0.5F	12	BT40-...0-120	1000 RPM	300 MMPM	Base Only	0.05 mm	0.00	0.00
✓D10R0.5F轮廓切削 3	✓	2.56	D10R0.5F	12	BT40-...0-120	1000 RPM	300 MMPM	Base Only	0.05 mm	0.00	0.00
✓D10R0.5F轮廓切削 4	✓	6.09	D10R0.5F	12	BT40-...0-120	1000 RPM	300 MMPM	Base Only	0.05 mm	0.00	0.00
✓D6R3F平行铣削 1	✓	15.70	D6R3F	15	BT40-...0-120	7000 RPM	1600 MMPM	0.00 mm	0.15 mm	0.00	0.00
✓D6R3F三维偏移切削 2	✓	13.94	D6R3F	15	BT40-...0-120	7000 RPM	1600 MMPM	0.00 mm	0.15 mm	0.00	0.00
✓D6R3F三维偏移切削 3	✓	6.90	D6R3F	15	BT40-...0-120	7000 RPM	1600 MMPM	0.00 mm	0.15 mm	0.00	0.00
Machine (undefined)											
Output											

【Where is it】:

CAM Manger >> Header Right-click >> Attribute Manager

5.4.8 Add Tool Path Divide Function

The tool path splitting function is often used to split the tool path with a long operation time or large height drop, which is conducive to replacing the tool or changing the clamping length of the tool in the actual operation and reducing the programming time of the user.

【What can be done】:

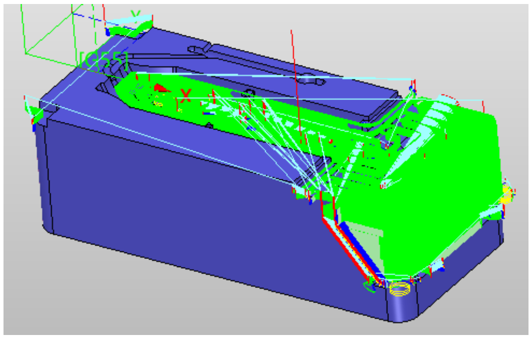
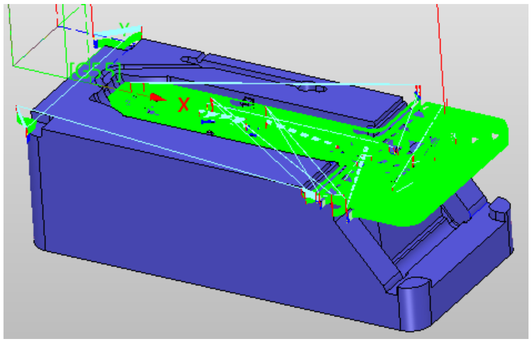
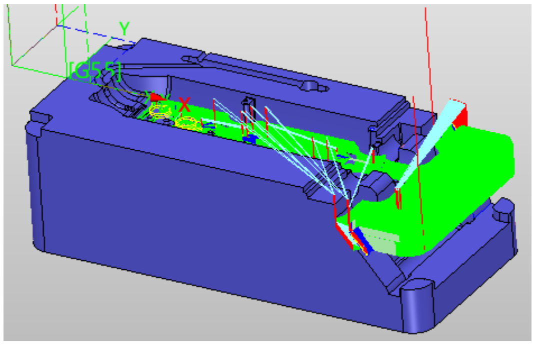
- ✓ Divide the tool path by the length of the tool path.
- ✓ Divide the tool track according to processing time.
- ✓ Divide the tool path according to the tool path segment.

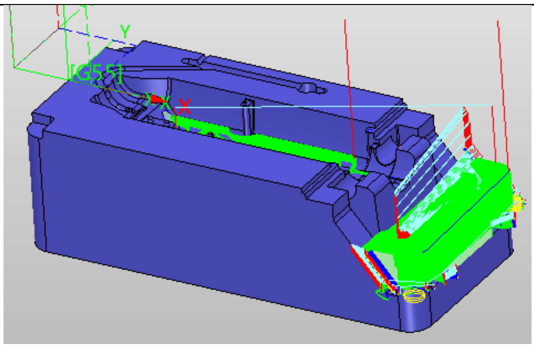
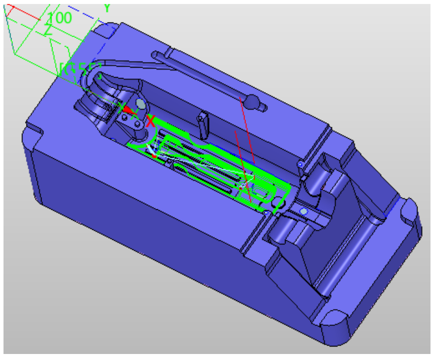
【Notes】:

- The tool path is divided according to the cutting tool path time, and the tool path will produce a new advance and retreat tool path after the tool path is divided, so the processing time on the CAM tree table will be greater than the set division time, and even more than the set time error.
- When using this feature, there is a delay of a few seconds before the blade track split progress

bar pops up.

【Example】：

Original toolpath	
Toolpath after being divided	 

【Where is it】:

Toolpath Editor >>Toolpath Divide

5.4.9 Innovation of Tool Path Trim Function

ZW3D 2025 has carried out an overall innovation to the tool track trimming function, and users can choose to cut the inside or outside of the cutting edge of the tool track, supporting the preservation of the original operation.

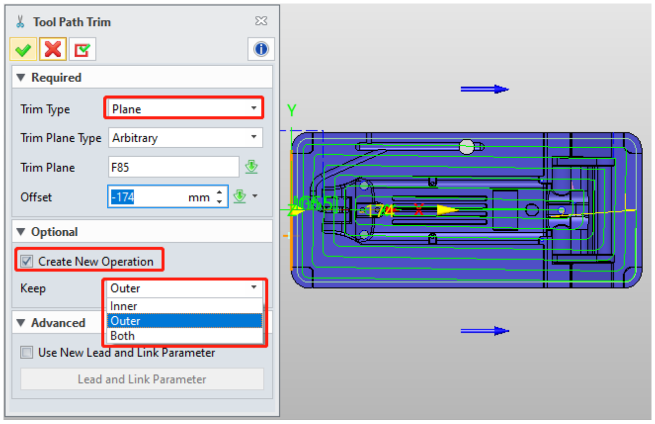
【What can be done】:

- ✓ Support select plane to trim toolpath.
- ✓ Support whether remain the original operation.
- ✓ Support remaining 3 trimming methods including Outer, Inner, and Both.

【Note】:

- Enable the Create New Operation function to preserve both modes.

【Example】:



Tool Path Trim

Required

Trim Type: Plane

Trim Plane Type: Arbitrary

Trim Plane: F85

Offset: -174 mm

Optional

☑ Create New Operation

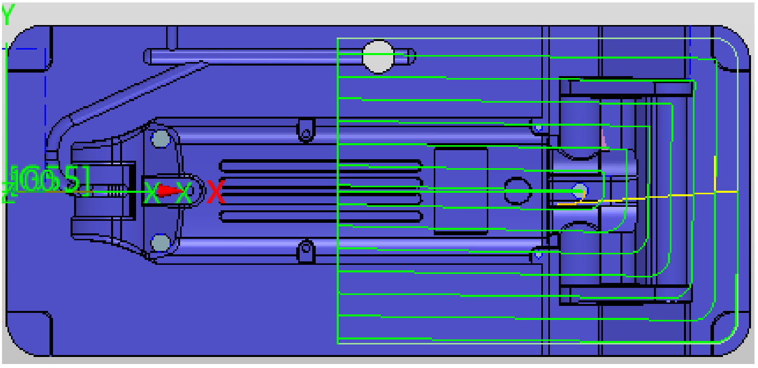
Keep: Outer

Advanced

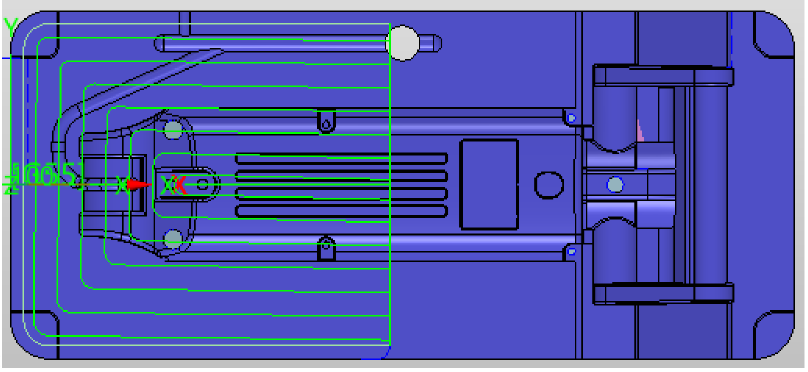
☐ Use New Lead and Link Parameter

Inner
Outer
Both
Lead and Link Parameter
Y
-174

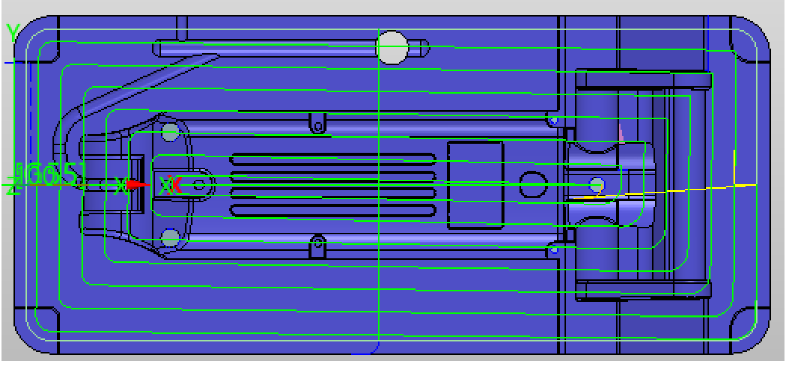
Outer side:



Inner side:



Both:



【Where is it】:

Toolpath Editor >> Toolpath Divide

5.4.10 Add Method to Create Stock

In the stock adding function of ZW3D 2025, two new billet creation methods have been added, respectively, part outline and part convex hull. The part outline is the drawing of the blank from the projected outline of the user-selected geometric object in the specified direction. The convex hull of the part is the blank drawn after simplifying the projected contour of the user-selected geometric object in the specified direction.

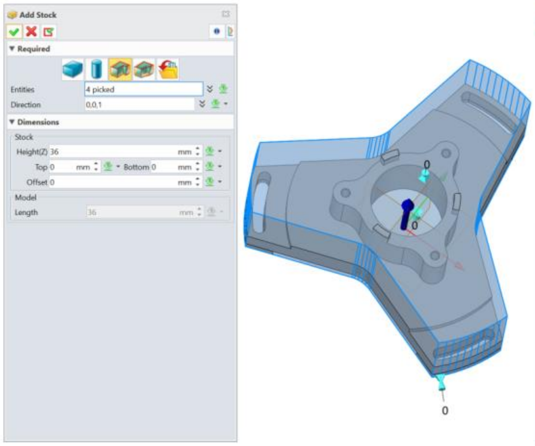
【What can be done】:

- ✓ Select a geometric object, specify the direction, and stretch the blank with its projected outline.
- ✓ Select a geometric object, specify the direction, and stretch the blank with the convex hull shape of its projected profile.

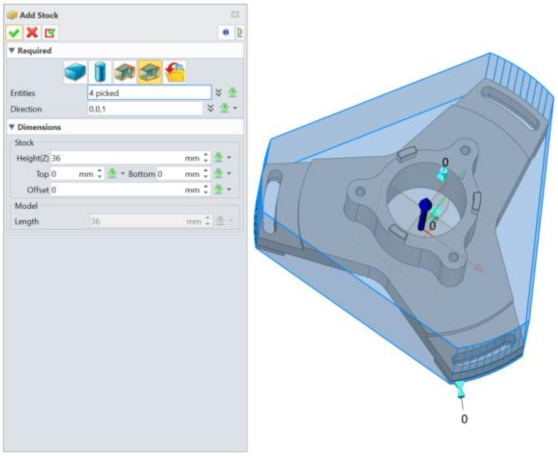
【Note】:

- The efficiency of drawing blank decreases with the complexity of the projected profile.
- The arc outline on the part is stretched by dividing it into several line segments.

【Example】:



Entities 4 picked
Direction 0,0,1
Height(Z) 36 mm
Top 0 mm
Bottom 0 mm
Offset 0 mm
Length 36 mm



Entities 4 picked
Direction 0,0,1
Height(Z) 36 mm
Top 0 mm
Bottom 0 mm
Offset 0 mm
Length 36 mm

【Where is it】:

Setups>>Add Stock>>Part Profile/ Part Convex Hull

6 API

6.1 ★API System Upgrade and Interface Expansion

6.1.1 Old Interface Iterative Upgrade

The new interface adopts the naming form of Zw+ module + function, which is more unified in structure. The new interface replaces the old id with a handle, which improves stability. The interface parameters are more standardized and more convenient to use. Increase the relationship between the point and the entity to ensure the accuracy of the regeneration.

【What can be done】:

- ✓ The old interface searches for the new interface
- ✓ The new interface can be used independently.
- ✓ The new interface provides greater ease of use and stability.

【Where is it】:

See the API Interface Conversion List and ApiHelpDoc.chm

6.1.2 New Interface Batch Expansion

Some new interfaces have been added according to the platform functions and customer requirements, and now the interface coverage of the platform functions is higher. By supplementing the interfaces required by customers, the degree of support for machinery, 3C, mold, robot, and other industries is higher.

【What can be done】:

- ✓ Platform features higher coverage.
- ✓ Support is higher in specific industries.

【Where is it】:

See Interface and Platform Function Comparison Table (to be released) and ApiHelpDoc.chm

6.1.3 Help Documentation Optimization

The interface supporting the help document has been updated, in addition to retaining the relevant content of the old interface, the mechanism and use of the new interface have been explained in detail. By referring to the help document, customers can quickly understand the relevant functions of ZWAPI. Help documents provide the ability to quickly search for the required interface, while online documents can quickly read comments with the help of online translation.

【What can be done】：

- ✓ Quickly understand interface functions.
- ✓ Find the desired interface.

【Where is it】：

ApiHelpDoc.chm and Online help documents (to be released)

6.1.4 Enrich Example Code

The new interface complements the accompanying code examples, including simple examples for the interface level and engineering examples with some visualization. You can paste sample code directly into a development template for rapid engineering development.

【What can be done】：

- ✓ Developers get familiar with the code quickly.
- ✓ Quick reuse of simple examples.

【Where is it】：

See Code and engineering examples (not yet available) and ApiHelpDoc.chm

6.2 Tcmd Editor

Given the problems of high difficulty in Tcmd file development and abstract keywords, this paper provides a tool for interactive Tcmd file editing, which can quickly edit the required Tcmd file by clicking based on the template file. Help documentation is also provided to quickly learn how to use the Tcmd editor.

【What can be done】:

- ✓ Template-developed Tcmd files.
- ✓ The relative concretization of keywords.

【Where is it】:

Tcmd Editor Introduction.pdf

6.3 VS Developing Wizard

Provides a one-click installation template, without the cumbersome environment configuration can be directly code development, combined with code examples and the Tcmd editor can complete efficient development.

【What can be done】:

- ✓ One-click installation development templates.
- ✓ Windows and Linux are supported.

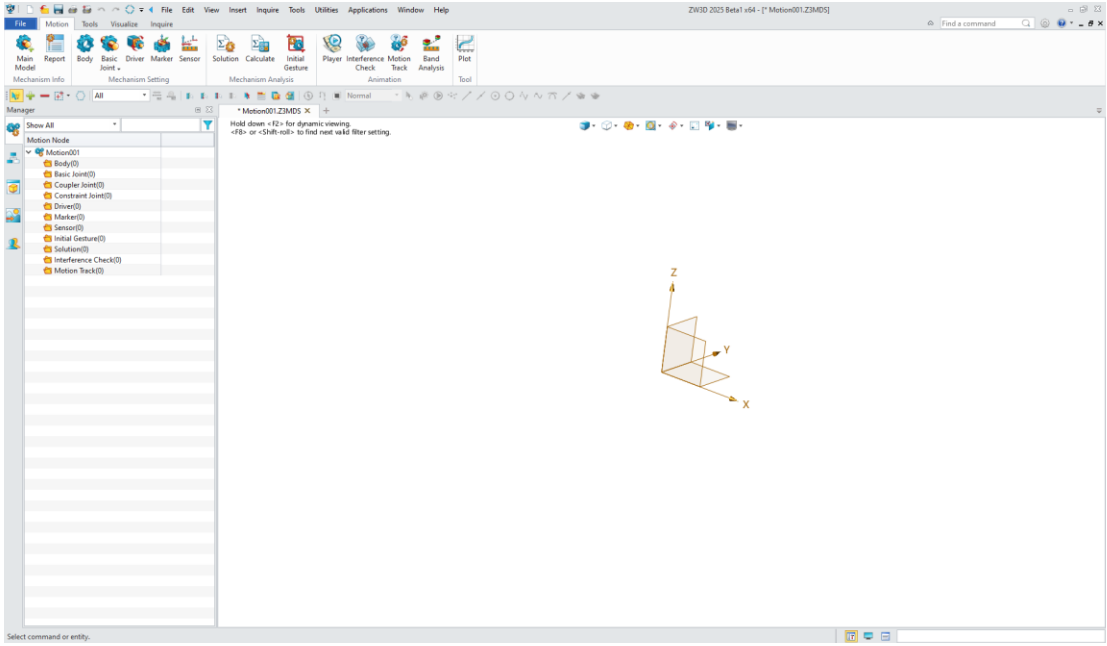
【Where is it】:

Secondary development wizard template

7 Simulation

7.1 ★Kinematics Simulation

Kinematics simulation has been added to ZW3D 2025, which can realize motion simulation based on the part file or assembly file. The analysis is based on motion objects such as body, joint, and driver. Analysis functions such as interference, motion track, and animation measurement can be defined in the analysis process. It also supports the export of analysis results, including motion data export, motion track export, motion video export, etc.



ZW3D 2025 Beta1 x64 - [* Motion001.Z3MDS]

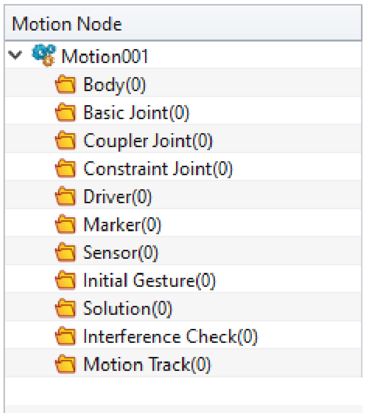
Hold down <F2> for dynamic viewing.
<F8> or <Shift-roll> to find next valid filter setting.

Select command or entity.

File
Edit
View
Insert
Inquire
Tools
Utilities
Applications
Window
Help
Motion
Tools
Visualize
Inquire
Main Model
Report
Body
Basic Joint
Driver
Marker
Sensor
Solution
Calculate
Initial Gesture
Player
Interference Check
Motion Track
Band Analysis
Plot
Mechanism Info
Mechanism Setting
Mechanism Analysis
Animation
Tool
Manager
Show All
Motion Node
Motion001
Body(0)
Basic Joint(0)
Coupler Joint(0)
Constraint Joint(0)
Driver(0)
Marker(0)
Sensor(0)
Initial Gesture(0)
Solution(0)
Interference Check(0)
Motion Track(0)
* Motion001.Z3MDS
Z
Y
X

7.1.1 Motion Object Manager

All objects which are created in the Motion Simulation environment are recorded in the motion object manager, including body, joint, driver, marker, sensor, solution, initial gesture, interference, motion track, etc. Users can perform various operations on motion objects.



Motion Node
Motion001
Body(0)
Basic Joint(0)
Coupler Joint(0)
Constraint Joint(0)
Driver(0)
Marker(0)
Sensor(0)
Initial Gesture(0)
Solution(0)
Interference Check(0)
Motion Track(0)

1. Edit

   Modify the defined motion objects by the Edit command.

2. Unblank/Blank

   Objects such as body, joint, driver generated in the Motion Simulation environment can be displayed or hidden. For specific behavior, refer to the Unblank/Blank operations of entities.

3. Deactivate/Activate

   The objects created in the motion simulation environment are in the active state by default. When solving the solution, the objects in the active state will be included in the solution solving, and the inactive objects will not be considered.

4. Deactivate All/Activate All

   If an object does not need to participate in solving the solution of the simulation, the “Deactivate All” command can completely disable the object.

5. Rename

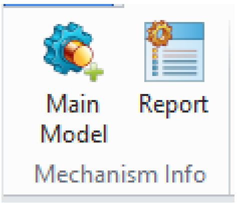
   Modify the name of the generated simulation object by the Rename command.

6. Delete

   Delete the generated simulation objects directly.

7.1.2 Mechanism Info

The information on the motion simulation model includes the main model and mechanism information in ZW3D. The former is the source of a model in motion simulation, while the latter is the information summary of the whole defined motion mechanism.



【What can be done】:

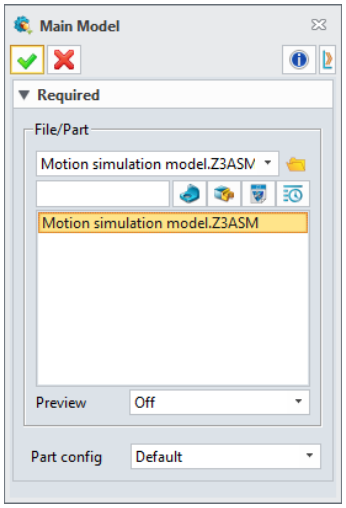
- ✓ Based on the assembly or part, it is added to the motion simulation file as the main model to carry out the motion simulation analysis
- ✓ Quickly preview the relationship between all objects in the entire motion simulation file and form a mechanism report that can be exported.

【Note】

- After the main model is imported, modifying the configuration of the file is not supported.

【Example】:

1. Use the “Main Model” command to select the created model for simulation.



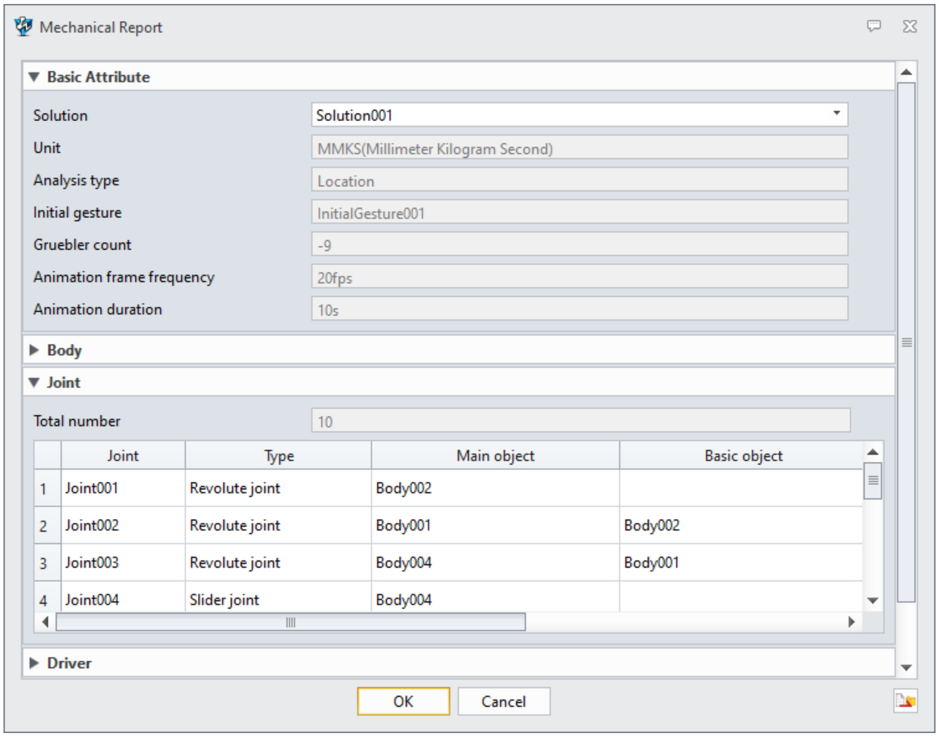
File/Part: Motion simulation model.Z3ASM

Motion simulation model.Z3ASM

Preview: Off

Part config: Default

2. When the motion mechanism is built, if fails to solve the solution, a report command can be used to quickly sort out the relationship between the motion objects and judge the correctness of the joint construction, convenient for modifying the motion mechanism.



Mechanical Report

Basic Attribute

Solution	Solution001
Unit	MMKS(Millimeter Kilogram Second)
Analysis type	Location
Initial gesture	InitialGesture001
Gruebler count	-9
Animation frame frequency	20fps
Animation duration	10s

Body

Joint

Total number 10

	Joint	Type	Main object	Basic object
1	Joint001	Revolute joint	Body002	
2	Joint002	Revolute joint	Body001	Body002
3	Joint003	Revolute joint	Body004	Body001
4	Joint004	Slider joint	Body004	

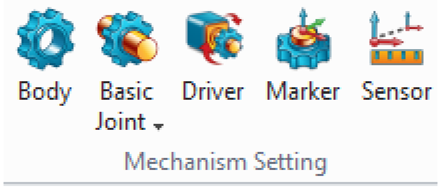
Driver

【Where is it】：

Motion >> Mechanism Info

7.1.3 Mechanism Setting

Mechanism setting is the basic construction of the whole simulation object, including defining the simulated body, creating joints between the motion bodies, and defining the simulation power source driver, defining the simulation analysis point marker and sensor. Only after the mechanism is set up, the motion simulation can be solved.



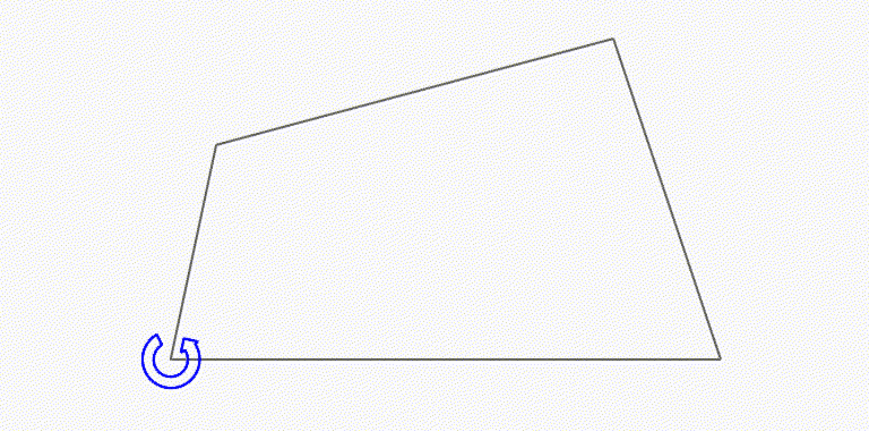
【What can be done】：

- ✓ Motion simulation based on part or assembly files is supported. The points, curves, shapes, components, and other objects can be defined as motion bodies.

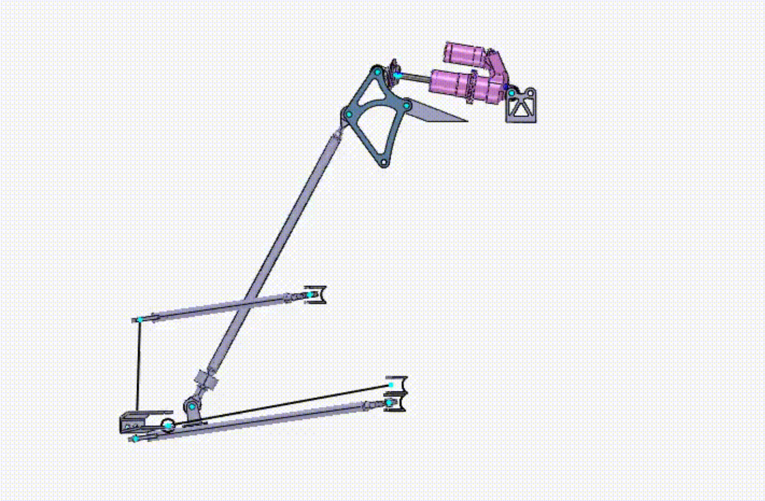
- ✓ Support the construction of common motion joints in kinematics simulation, and is classified into basic joint, constraint joint, and coupler joint.
- ✓ Supports the creation of various types of driver functions. The external data can be fit to be the drive rule for function.
- ✓ Read the motion parameters of any point of the motion body is supported.
- ✓ Read the change of motion parameters between any two motion objects is supported.

【Example】：

1. Verifying the correctness of the four-link mechanism preliminarily by the motion simulation of the point-and-line model.



2. Verifying whether the chassis suspension design meets the requirements by testing the motion performance of the chassis in the car chassis design process.



7.1.4 Mechanism Analysis

After finishing the mechanism setting, users can proceed to the mechanism analysis process. In the mechanism analysis, users can set the solution type, solution duration and initial conditions of the mechanism through the solution, and the initial gesture functions. In addition, the object which is involved in the solution can be set by the solution function.

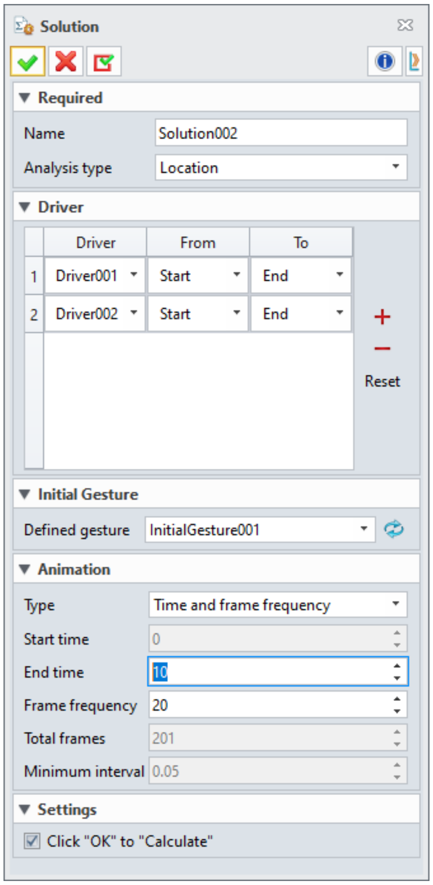
During the simulation, the initial position of the simulation can be adjusted through the initial gesture function. The gesture relationship of all motion bodies at the time of definition is recorded in the initial gesture.

【What can be done】：

- ✓ Support the definition of two types of motion analysis: kinematics analysis and position analysis.
- ✓ Support selecting different objects to participate in the simulation solution.
- ✓ Support adjusting the initial state of the motion model and carry out simulation analysis with different initial states.

【Example】：

1. The car suspension motion simulation includes two different driving speeds, which can be realized by selecting different drivers to participate in the calculation.



Solution

Required

Name: Solution002

Analysis type: Location

Driver

	Driver	From	To
1	Driver001	Start	End
2	Driver002	Start	End

Initial Gesture

Defined gesture: InitialGesture001

Animation

Type: Time and frame frequency

Start time: 0

End time: 10

Frame frequency: 20

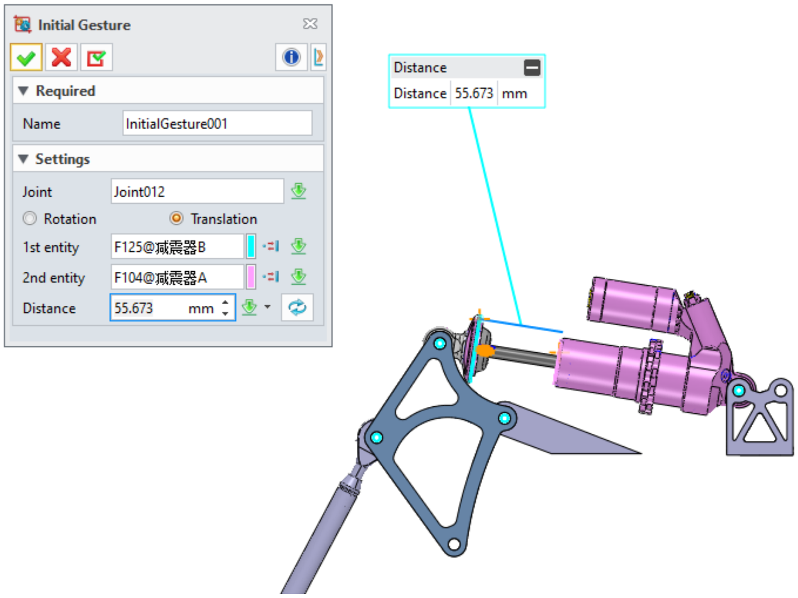
Total frames: 201

Minimum interval: 0.05

Settings

☑ Click "OK" to "Calculate"

2. Adjust the initial position of the shock absorber.



Initial Gesture

Required

Name: InitialGesture001

Settings

Joint: Joint012

○ Rotation ● Translation

1st entity: F125@减震器B

2nd entity: F104@减震器A

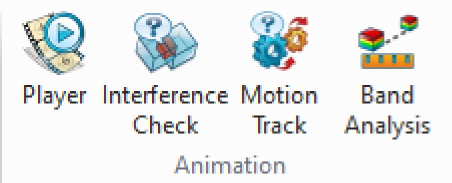
Distance: 55.673 mm

【Where is it】：

Motion >> Mechanism Analysis

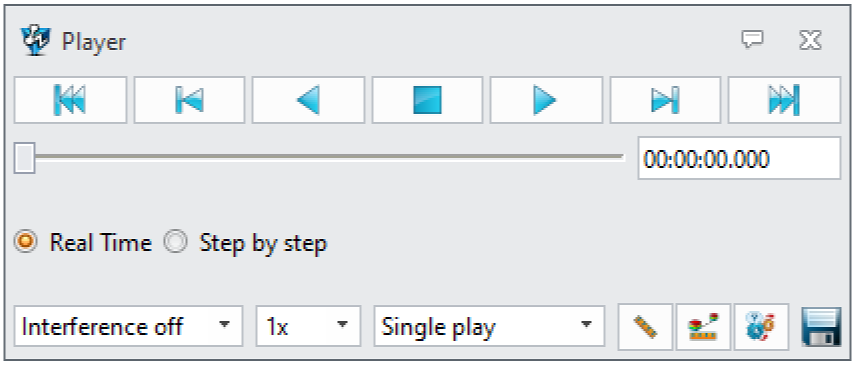
7.1.5 Animation

After finishing the solution calculation, users can view the motion process through the player. It can also set an animation analysis item to check whether the mechanism meets the design requirements.



7.1.5.1 Player

Through the player, the motion effect of the motion mechanism that has been solved can be demonstrated. It provides multiple playback modes, and it can control whether animation analysis items take effect during playback.



00:00:00.000

● Real Time ○ Step by step

Interference off

1x

Single play

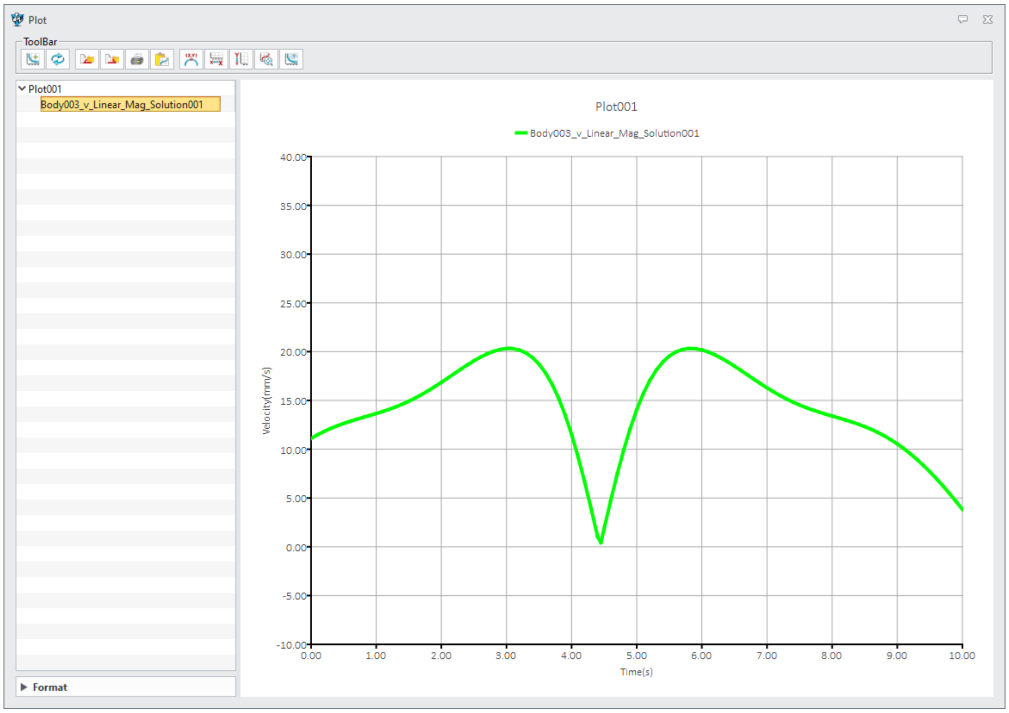
For the introduction of interference check, motion track, band analysis, and animation measurement, please refer to the Assembly Animation section.

【Where is it】:

Motion >> Animation

7.1.6 Tool

After finishing the motion simulation analysis, visual analysis of the solved motion data can be carried out through the ZW3D plot tool, it can also carry out the data image, data analysis, and other operations.

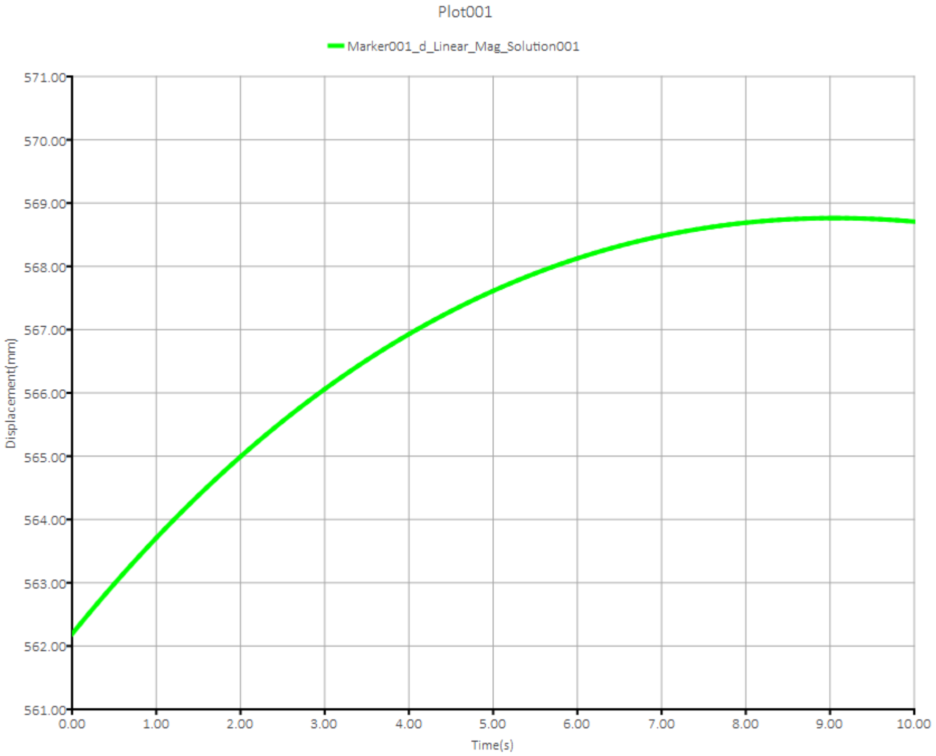

Plot
ToolBar
Plot001
Body003_v_Linear_Mag_Solution001
Format
Plot001
Body003_v_Linear_Mag_Solution001
Velocity(mm/s)
40.00
35.00
30.00
25.00
20.00
15.00
10.00
5.00
0.00
-5.00
-10.00
0.00
1.00
2.00
3.00
4.00
5.00
6.00
7.00
8.00
9.00
10.00
Time(s)

【What can be done】：

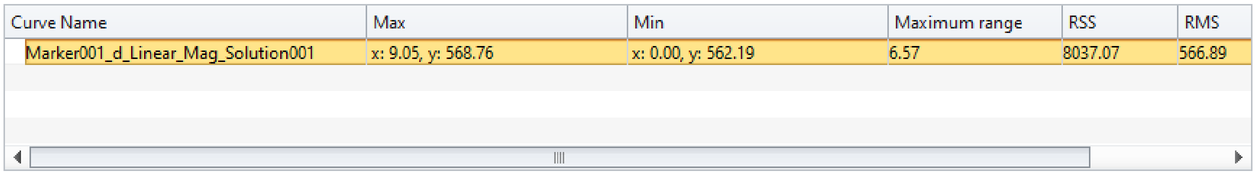
- ✓ Check the changes of motion parameters in all directions on each motion object.
- ✓ The display format and data form of each curve can be modified.
- ✓ Multiple curves can be placed in the same chart and compared to see how the data changes.
- ✓ Quadratic calculations can be performed based on the existing motion data.
- ✓ Data analysis can be performed on the obtained data results to get the maximum value, minimum value, arithmetic average value, and other parameters.
- ✓ The obtained motion data can be exported.

【Example】：

1. Check the movement displacement change of the car transverse arm.



2. Analyze and obtain the maximum value and minimum value of movement displacement of the car suspension transverse arm.



Curve Name	Max	Min	Maximum range	RSS	RMS
Marker001_d_Linear_Mag_Solution001	x: 9.05, y: 568.76	x: 0.00, y: 562.19	6.57	8037.07	566.89

【Where is it】:

Motion>>Tool>>Plot

7.2 Structural Simulation

7.2.1 Analysis Types

7.2.1.1 Added Harmonic Fatigue Analysis and Random Vibration Fatigue Analysis

Because structures are subject to harmonic vibration or random vibration, it is often necessary to pay attention to their fatigue life under these working conditions. This type of problem is relatively common in vibrating machinery and electronic components. The new version provides harmonic fatigue and random vibration fatigue analysis.

【What can be done】:

- ✓ Users can predict the extent of damage and remaining life of components subjected to repeated cyclic loading at a given operating frequency.
- ✓ Users can predict the degree of damage and remaining life of working components in a random vibration environment.

【Note】:

- Fatigue analysis mode does not support finite element analysis results imported in the form of mesh files.

【Example】:

1. Please refer to the Fatigue Analysis of bases (Constant amplitude) in the tutorial document.

【Where is it】:

Part/Assembly Environment >> Simulation >> New Structure >> Fatigue Analysis >> Harmonic/ Random Vibration Fatigue Analysis

7.2.1.2 Added Drop Test Analysis

For electronic components or household appliances, it is necessary to analyze the stress under drop conditions during transportation or use and observe their deformation, stress, and strain. The new version provides the Drop Test module.

【What can be done】:

- ✓ Can automatically create a virtual wall, which is the ground where it falls.
- ✓ The initial conditions of the structure under falling conditions can be given by the falling height or initial velocity.
- ✓ It can observe the performance of objects in free fall motion and the degree of damage, deformation, and functional performance on rigid or flexible planes.

【Note】:

- For drop tests, users need to define drop test analysis options before solving to determine the parameters and methods of solving.
- Drop test supports linear elastic materials and plastic materials.

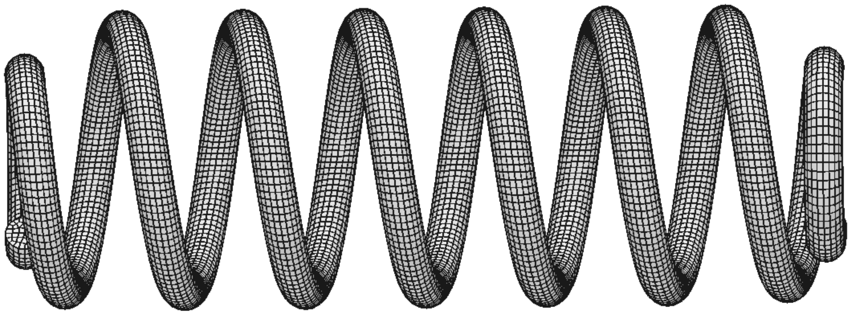
【Where is it】:

Part/Assembly Environment >> Simulation >> New Structural >> Drop Test

7.2.2 Improvement of Pre- and Post-Processing Function

7.2.2.1 Added Mesh Translation, Rotation and Sweep

The new version provides more ability to manually create meshes. The desired mesh can be directly created through mesh translation, rotation, and sweeping in mesh editing. The swept mesh is mapped to another surface by drawing a 2D mesh first on one surface to generating a volume mesh.



Sweep Mesh

【What can be done】:

- ✓ Use mesh translation to translate nodes or elements along a custom direction.

- ✓ Use mesh rotation to rotate nodes or elements around the normal of a custom plane.
- ✓ Use mesh sweep to create a 3D mesh by sweeping the selected 2D mesh source plane toward the target plane.

【Note】:

- The edited mesh will lose its relationship with the geometry.

【Where is it】:

Part/Assembly Environment >> Simulation >> Ribbon >> Mesh Edit >> Translate, Rotate, Sweep

7.2.2.2 Added Manual Element Creation

The new version provides a way to manually create meshes. Element nodes can be selected to create meshes, and currently creating surface and solid meshes is supported.

【What can be done】:

- ✓ Supports manual creation of surface elements and solid elements.

【Note】:

- The manually created mesh is independent, and its material properties need to be reset.

【Where is it】:

Part/Assembly Environment >> Simulation >> Ribbon >> Mesh Editing >> Create Elements

7.2.2.3 Added Finding Mesh Edges and Surfaces and Extracting Mesh Edges Functions

The new version supports the functions of finding mesh open edges, open faces, and extracting all edge lines of the mesh. This function can be used to view the boundary lines of the mesh to observe their connection relationships and extract all edge lines of the mesh.

【What can be done】:

- ✓ Searching for surfaces can search for 3D mesh surface meshes and generate 2D mesh.
- ✓ Find edges to find 2D/3D mesh boundary edges and generate 1D mesh.

- ✓ Extracting edges can extract 2D/3D mesh boundary edges separately to generate 1D mesh.

【Note】:

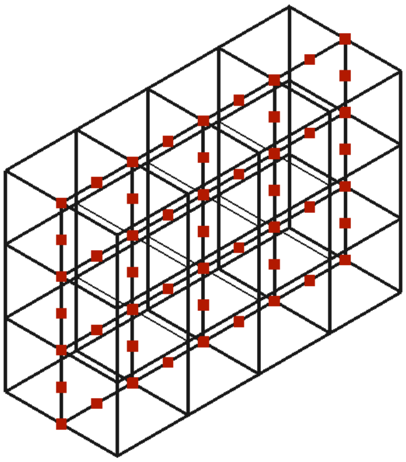
- The generated mesh will be stored in the mesh component.

【Where is it】:

Part/Assembly Environment >> Simulation >> Ribbon >> Mesh Edit >> Find Edges/Find Faces/Extract Edges

7.2.2.4 Added Merging Mesh Nodes Function

The new version supports merging mesh nodes at adjacent locations, which can quickly remove small-sized elements, remove low-quality elements, fill surfaces, etc.



【What can be done】:

- ✓ Selected mesh nodes can be merged within the tolerance range and new nodes can be generated.

【Where is it】:

Part/Assembly Environment >> Simulation >> Ribbon >> Tools >> Merge Nodes

7.2.2.5 The Efficiency of Importing/Exporting Large BDF Files has been Improved Significantly in the New Version

In the new version, the import and export efficiency of large BDF files has been improved by orders of magnitude.

【Example】:

1. In the new version, the BDF export time of the 3 million element model is reduced to a few minutes.

7.2.2.6 Added the Function of Automatically Creating Beams（ZW3D 2025 SP）

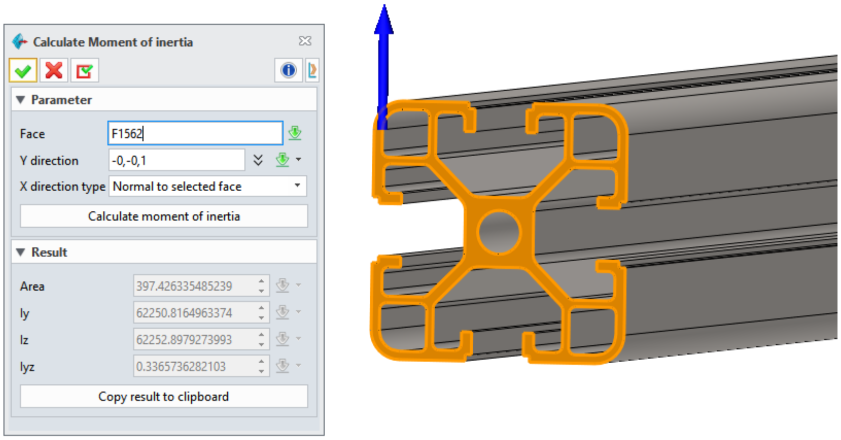
Extracting the centerline means simplifying the 3D solid structure into a line representing its geometric center in simulation. It can reduce the amount of calculation and complexity in simulation and is usually used for structures with large slenderness. Additionally, section information of the model can be identified during the extraction process, and the cross-sectional attributes of the beam elements are automatically assigned during analysis creation.

➔ Where

Simulation >> New Structure >> Automatically Create Beam

7.2.2.7 Added the Function of Automatically Calculating Beam Section Parameters（ZW3D 2025 SP）

The new version supports quickly calculating the beam section properties (such as area, moment of inertia) by selecting the cross-sectional geometry of the beam. Users can obtain accurate beam properties without manually deriving complex formulas.



Calculate Moment of inertia

Parameter

Face F1562

Y direction -0,-0,1

X direction type Normal to selected face

Result

Area 397.426335485239

Iy 62250.8164963374

Iz 62252.8979273993

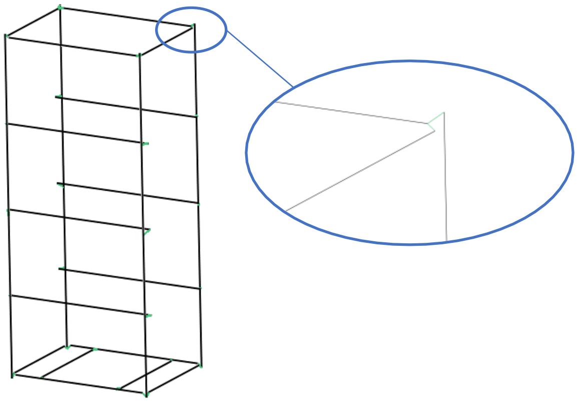
Iyz 0.3365736282103

➔ Where

Simulation >> New Structure >> Calculate Moment of Inertia

7.2.2.8 Added the Function of Automatically Creating Beam Connections（ZW3D 2025 SP）

The automatic beam connection function can automatically recognize the connections between adjacent beams in a wireframe model and quickly generate connection constraints based on defined connection rules. It can reduce manual definition errors and improve modeling efficiency and accuracy for complex framework structures.

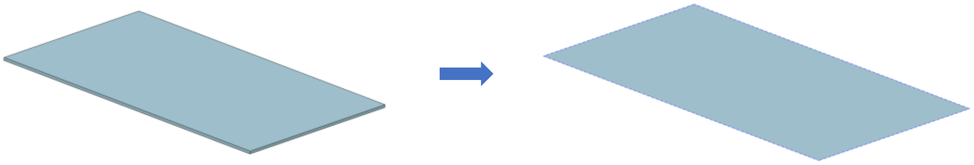


➔ Where

Simulation >> New Structure >> Auto Joint Connection

7.2.2.9 Added Mid-Surface Creation by Surface Offset（ZW3D 2025 SP）

For thin-shell structures, simplifying a 3D structure with small thickness into a 2D mid-surface reduces the number of nodes and elements to be calculated, enhancing computational efficiency. The new version supports the selection of geometric faces and the automatic generation of mid-surfaces through a surface offset algorithm. During the creation of the analysis task, shell element thickness attributes are automatically assigned.



➔ **Where**

Simulation >> New Structure >> Mid-Surface by Offset

7.2.2.10 Added Automatic Mid-surface Creation Function（ZW3D 2025 SP）

The new version supports automatic mid-surface creation based on surface offset, eliminating the need for manual face selection. By defining angle tolerance and thickness range, it automatically identifies geometric face pairs and generates mid-surface. During the creation of the analysis task, shell element thickness attributes are automatically assigned.

➔ **Where**

Simulation >> New Structure >> Automatically Mid-surfaces

7.2.2.11 Added Batch Mid-Surface Creation for Assemblies（ZW3D 2025 SP）

The new version offers a mid-surface extraction manager for assemblies, which can automatically recognize mid-surface faces based on defined angle tolerance and thickness range. Users can monitor extraction progress and stop the extraction process.

➔ **Where**

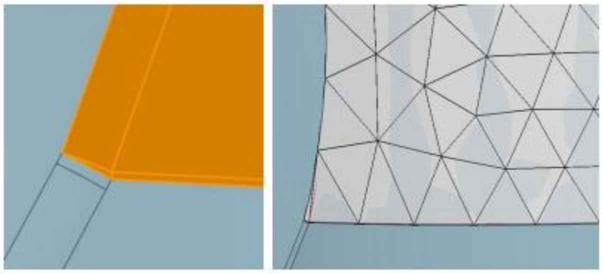
Assembly Environment >> Simulation >> New Structure >> Batch Mid-surfaces

7.2.2.12 Added the Function of Inheriting ZW3D Material Parameters（ZW3D 2025 SP）

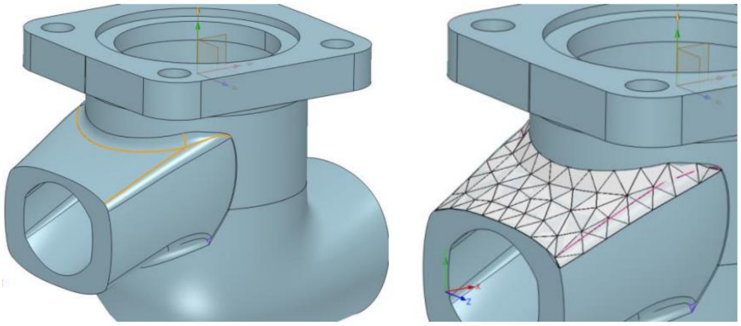
The new version allows users to directly inherit the parameters of the materials set in the ZW3D modeling environment when creating simulation tasks, such as density, elastic modulus, Poisson's ratio, specific heat capacity, etc.

7.2.2.13 Added Virtual Topology Functions for Merging Faces and Suppressing Edges（ZW3D 2025 SP）

The new version optimizes the virtual topology technology. By merging faces and suppressing edges based on virtual topology, high-quality mesh can be obtained without modifying the real geometry.



2D mesh results after virtual topology merge faces



2D mesh results after virtual topology suppress edge

➔ Where

Simulation >> New Structure>> Mesh >> Merge Face

Simulation >> New Structure >> Mesh >> Suppress Edge

7.2.2.14 Added Mesh Renumbering Function（ZW3D 2025 SP）

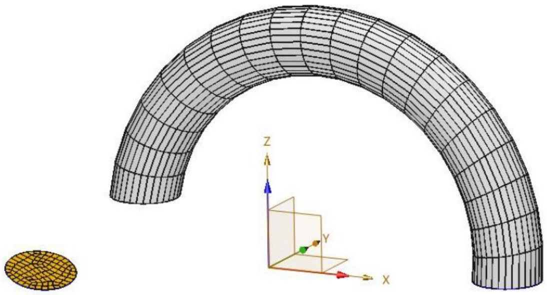
The new version supports renumbering nodes or elements in the mesh with non-continuous numbering.

➔ Where

Simulation >> New Structure >> Mesh >> Renumber

7.2.2.15 Added Mesh Extrude Function（ZW3D 2025 SP）

The new version introduces a mesh extrusion tool, which can extrude an existing 2D mesh along a specified direction to generate a regularly shaped 3D mesh.

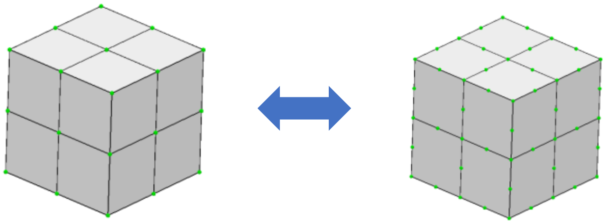


➔ Where

Simulation >> New Structure >> Mesh >> 3D Extrude

7.2.2.16 Added the Function of Changing the Element Order（ZW3D 2025 SP）

The new version adds the ability to change element order, such as switching from first-order elements to second-order elements. Users can flexibly adjust element order based on different analysis needs.



➔ Where

Simulation >> New Structure >> Mesh >> Elements order change

7.2.2.17 Added the Function of XY Plot based on the Maximum and Minimum Values of Each Step（ZW3D 2025 SP）

The new version optimizes the XY plot function, supporting the output of curve plots based on the maximum and minimum values per step.

7.2.2.18 Add Result Calculation Function（ZW3D 2025 SP）

The new version supports arithmetic operations on contour plots and outputs a new contour plots with the computed results.

➔ Where

ZBF: Simulation >> Simulation Tree >> Results >> Result Function

7.2.2.19 Optimized Result Output（ZW3D 2025 SP）

The new version optimizes the solver's use of storage space and greatly improves the efficiency of writing result files.

7.2.2.20 Added the Function of Importing and Exporting inp, zdf, and zbf files（ZW3D 2025 SP）

The new version supports the import and export of Abaqus solver input files (.INP), ZW3D simulation solver input files (.ZDF) and ZW3D structural simulation result files (.ZBF).

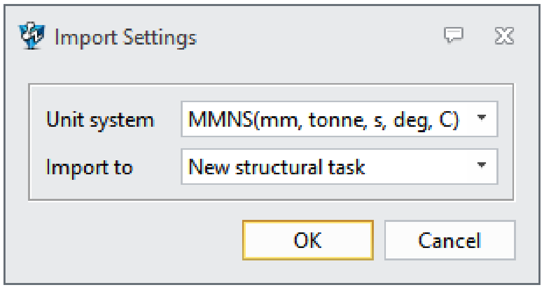
➔ Where

ZBF: Simulation >> Simulation Tree >> Results >> Import/Export Result Files

ZDF, INP: Simulation >> Tools >> Input/Output

7.2.2.21 Optimized Unit Selection for Simulation File Import（ZW3D 2025 SP）

When importing solver files, the new version supports the unit combination of millimeter-length and ton-mass, improving data import flexibility.



Unit system MMNS(mm, tonne, s, deg, C)

Import to New structural task

➔ Where

Simulation >> Tools >> Input >> Units

7.2.2.22 Added the Unit Switching Function for Material Library（ZW3D 2025 SP）

In the new version’s material library, users can directly switch material parameter units without converting material properties into a fixed unit system before inputting them.

➔ Where

Simulation >> Material Library

7.2.3 Improvement Of Calculation Functions

7.2.3.1 Added Algorithm for Incompatible Hexahedral Elements

In the new version, when modeling with first-order hexahedral elements, the incompatible element algorithm will be automatically applied. This algorithm resolves the self-locking issue of first-order hexahedral elements, providing more accurate results in geometric nonlinearity.

【What can be done】：

- ✓ When using first-order hexahedrons for simulation analysis, the non-coordinated element algorithm will be automatically used.

【Note】：

- This hexahedral element algorithm has high requirements for mesh quality.

【Example】：

1. The first-order non-coordinated element calculates the bending problem. When the number of elements is similar, the calculation accuracy can be like that of the second-order element, but the calculation overhead is greatly reduced.

7.2.3.2 Added Load Stiffness Calculation for Pressure, Line Loads, and Centrifugal Force

The new version supports the calculation of load stiffness for pressure, line loads, and centrifugal forces. In geometrically nonlinear analysis, it is essential to consider the changes in force direction and magnitude due to the deformation of the structure and the stiffness contributions resulting from it for more accurate results.

【What can be done】:

✓ Turn on the geometric nonlinearity switch when related loads are involved in the calculation. Turning on follower loads will consider the impact of the spatial position of these loads on the stiffness and external loads, making the calculation more accurate.

【Note】:

- Follower Loads needs to be turned on to consider deformation-related effects.

【Example】:

1. The problem of large deformation in cantilever beam bending. Considering the deformation effects of pressure and line load, the beam bending will be more accurate.
2. Centrifugal force is suitable for rotation problems that consider large deformations.

【Where is it】:

Part/Assembly Environment >> Simulation >> Task Options >> Nonlinear Basis >> Follower Loads

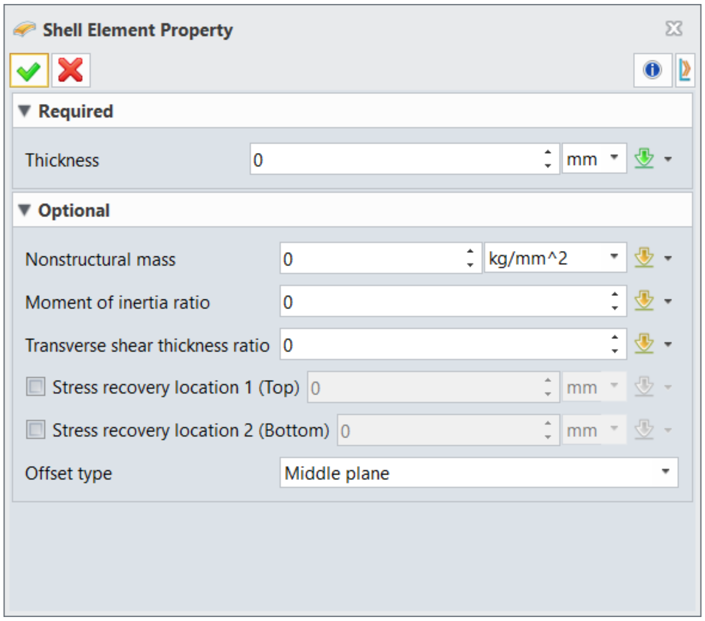
7.2.3.3 Added Offset Functionality for Beams and Shells

The new version supports the offset functionality for beams and shells. When the neutral axis or neutral plane of a beam or shell is at a certain distance from the selected element line or face, it can be achieved by setting the offset amount without the need to redraw element lines or faces.

【Where is it】：

Part/Assembly Environment >> Simulation >> Simulation Tree >> Beam Properties >> Section Offset

Part/Assembly Environment >> Simulation >> Simulation Tree >> Shell Properties >> Offset Values



Shell Element Property

▼ Required

Thickness: 0 mm

▼ Optional

Nonstructural mass: 0 kg/mm^2

Moment of inertia ratio: 0

Transverse shear thickness ratio: 0

☐ Stress recovery location 1 (Top): 0 mm

☐ Stress recovery location 2 (Bottom): 0 mm

Offset type: Middle plane

7.2.3.4 Added Cable Elements

The new version supports cable elements, which can be considered as rod elements that can apply pre-stress. Initial force, initial strain, or initial relaxation can be used to set initial conditions for cable elements.

【What can be done】：

- ✓ Its initial force, initial strain, or initial relaxation can be set.

【Where is it】：

Part/Assembly Environment >> Simulation >> Simulation Tree >> Cable Properties

7.2.3.5 Added Orthotropic and Isotropic Materials

The new version introduces orthotropic and isotropic material models in the material library. These models are used to simulate materials with significantly different properties in different directions. Both materials can be assigned to solid or shell elements, and material direction needs to be specified.

【What can be done】:

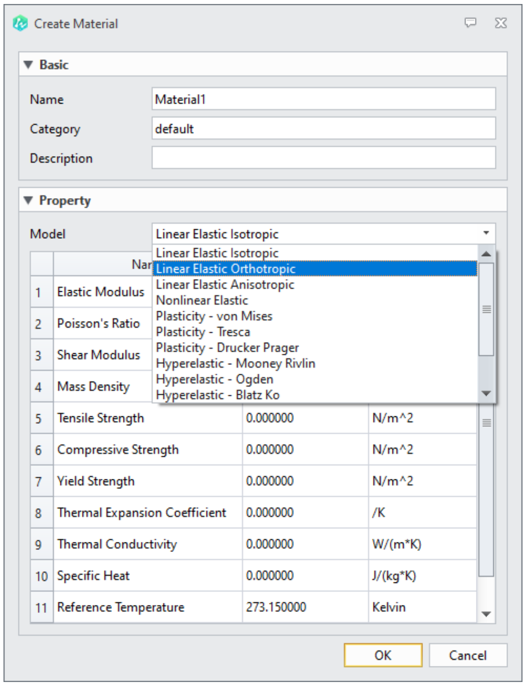
- ✓ Three parameters can be used to define the elastic modulus of orthotropic materials in the X, Y, and Z directions respectively.
- ✓ Six parameters can be used to define the shear modulus and Poisson's ratio of orthotropic materials in the XY, YZ, and XZ directions respectively.
- ✓ 33 parameters can be used to define the elastic modulus, shear model and Poisson's ratio of fully anisotropic materials.

【Note】:

- The orthotropic and fully anisotropic material models can only be assigned to solid or shell elements, and the material direction needs to be specified.

【Where is it】:

Part/Assembly Environment >> Simulation >> Ribbon >> Material Library >> Linear Elastic Orthotropy/ Linear Elastic Anisotropy


Create Material
Basic
Property
Linear Elastic Isotropic
Linear Elastic Orthotropic
Linear Elastic Anisotropic
Nonlinear Elastic
Plasticity - von Mises
Plasticity - Tresca
Plasticity - Drucker Prager
Hyperelastic - Mooney Rivlin
Hyperelastic - Ogden
Hyperelastic - Blatz Ko
OK
Cancel

Name: Material1
Category: default
Description:
Model: Linear Elastic Isotropic

	Nar		
1	Elastic Modulus		
2	Poisson's Ratio		
3	Shear Modulus		
4	Mass Density		
5	Tensile Strength	0.000000	N/m^2
6	Compressive Strength	0.000000	N/m^2
7	Yield Strength	0.000000	N/m^2
8	Thermal Expansion Coefficient	0.000000	/K
9	Thermal Conductivity	0.000000	W/(m*K)
10	Specific Heat	0.000000	J/(kg*K)
11	Reference Temperature	273.150000	Kelvin

7.2.3.6 Added Thermal Contact Resistance Functionality

In the new version, steady-state and transient thermal analyses support the setting of thermal contact resistance between surfaces. This is used to simulate changes in thermal conductivity due to the presence of gaps between components, resulting in discontinuous temperature distributions.

【What can be done】：

- ✓ Define contact thermal resistance by setting the thermal conductivity coefficient.

【Note】：

- Contact thermal resistance can be used only in steady-state thermal analysis and transient thermal analysis.

【Where is it】：

Part/Assembly Environment >> Simulation >> Contact >> Heat Transfer

7.2.3.7 Added Kinematic Rigid Elements for Large Deformations

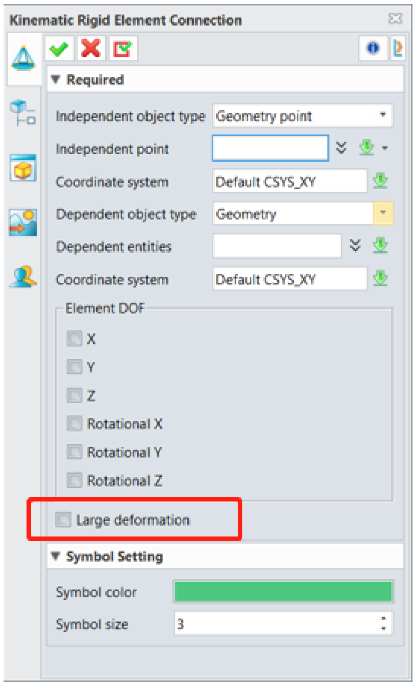
In the new version, kinematic rigid elements for large deformations have been added. These elements use finite rotation theory to accurately simulate the motion of rigid elements in large deformation analyses.

【What can be done】：

- ✓ Under the nonlinear analysis type, check the large deformation under the dynamic rigid element function to turn it on.

【Where is it】：

Part/Assembly Environment >> Simulation >> Connections >> Dynamic Rigid Elements >> Large Deformations



Kinematic Rigid Element Connection

▼ Required

Independent object type: Geometry point
Independent point:
Coordinate system: Default CSYS_XY
Dependent object type: Geometry
Dependent entities:
Coordinate system: Default CSYS_XY

Element DOF
☐ X
☐ Y
☐ Z
☐ Rotational X
☐ Rotational Y
☐ Rotational Z

☐ Large deformation

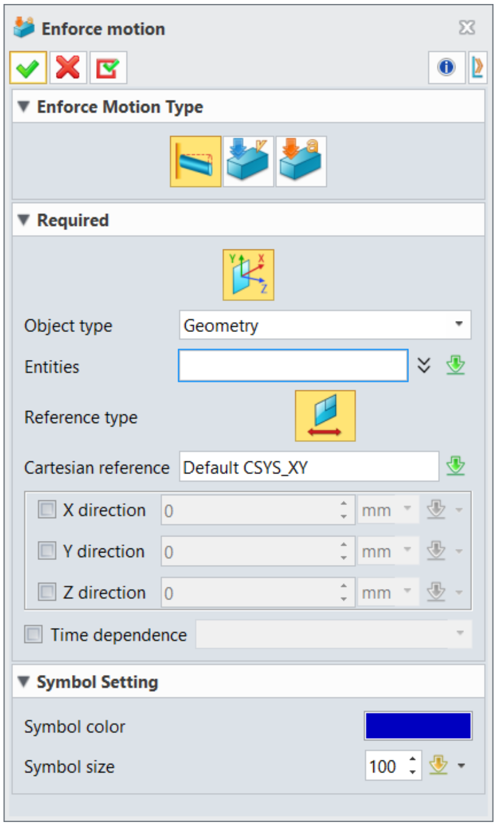
▼ Symbol Setting

Symbol color: 🟩
Symbol size: 3

7.2.3.8 Added Enforced Motion

In the new version, enforced motion is supported in linear statics, nonlinear statics, linear dynamics, and nonlinear dynamics. Corresponding displacements, velocities, and accelerations can be specified.

【Example】：



Enforce motion

Enforce Motion Type

Required

Object type: Geometry

Entities:

Reference type

Cartesian reference: Default CSYS_XY

☐ X direction: 0 mm

☐ Y direction: 0 mm

☐ Z direction: 0 mm

☐ Time dependence

Symbol Setting

Symbol color

Symbol size: 100

【Where is it】:

Part/Assembly Environment >> Simulation >> Simulation Tree >> Mechanical Loads >> Forced Motion

7.2.3.9 Improved Large Sliding Frictional Contact

In the new version, both the convergence and accuracy of large sliding friction contact have been further improved.

【Where is it】:

Part/Assembly Environment >> Simulation >> Contact

7.2.3.10 Optimized the Computing Power of Large Volume Models

The new version has optimized the memory consumption in the calculation of large-volume constraint equations. And through optimization matrix technology, the computational efficiency of large-volume models has been significantly improved.

【Example】:

1. In the previous version, 600,000 constraint equations required about 60GB of memory, but in the current version, it only requires about 10GB.
2. In the previous version, a linear solution with 4 million degrees of freedom took 3600s, but in the current version, it only takes about 700s.

7.2.3.11 Added Timoshenko Beam Element（ZW3D 2025 SP）

The new version introduces the Timoshenko beam element, which considers the effects of shear deformation. Users can obtain more accurate stress and deformation results of beam structures.

➔ **Where**

Simulation Tree >> New Structure >> Beam Element Property

7.2.3.12 Optimize Contact Function（ZW3D 2025 SP）

The new version’s contact function allows users to define more contact parameters, such as ramp interference, which prevents the solver from experiencing sudden excessive interference at the initial contact stage, which may cause the solver to fail to converge. Users can also define contact penalty factors to balance calculation accuracy and solver stability.

➔ Where

Simulation >> New Structure >> Contact

7.2.3.13 Optimize Harmonic Analysis Algorithm（ZW3D 2025 SP）

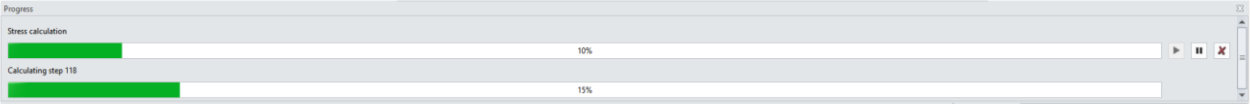
The new version optimizes the harmonic response analysis solver algorithm, improving computational efficiency dozens of times for large-scale cases. Additionally, the harmonic response analysis now supports the residual vector method for force loading, which can correct modal truncation errors for higher analysis accuracy.

7.2.4 Others

7.2.4.1 Added the Overall Progress Bar

In addition to the step-by-step progress bar, the new version also displays an overall progress bar for assessing the estimated overall progress.

【Example】:



7.2.4.2 Added Real-Time Convergence Curve and Energy Curve

The new version will display the calculated convergence residuals in real time for nonlinear statics and nonlinear dynamics. Users can evaluate the convergence situation based on this. Explicit dynamics analysis (drop test) will display the kinetic energy curve. If the kinetic energy fluctuation is too large, it is generally the time increment selected is too large and needs to be adjusted to make it smaller than the stable time.

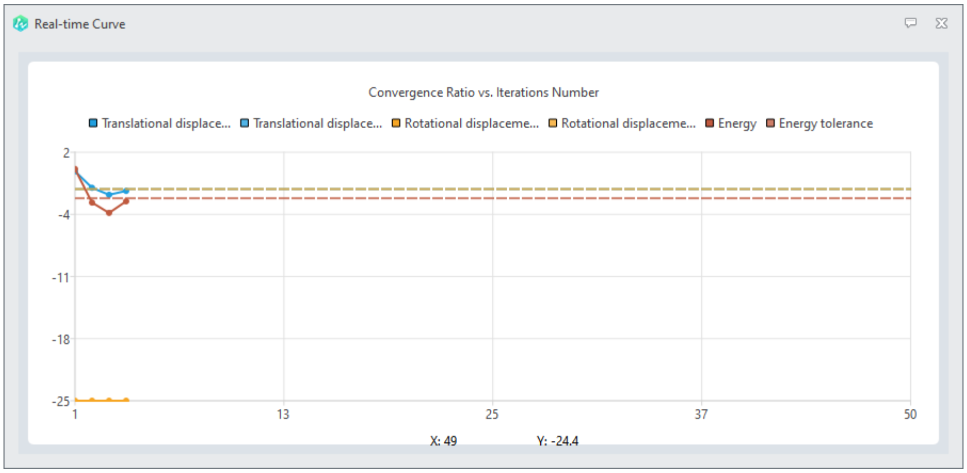
【What can be done】:

- ✓ The iteration steps and convergence tolerance of nonlinear analysis can be plotted as a curve graph.
- ✓ During calculation, a real-time updated convergence rate vs. number of iterations graph will appear on the interface. The horizontal axis is the number of iterations of the algorithm, and the vertical axis is the rate of solution convergence in each iteration.

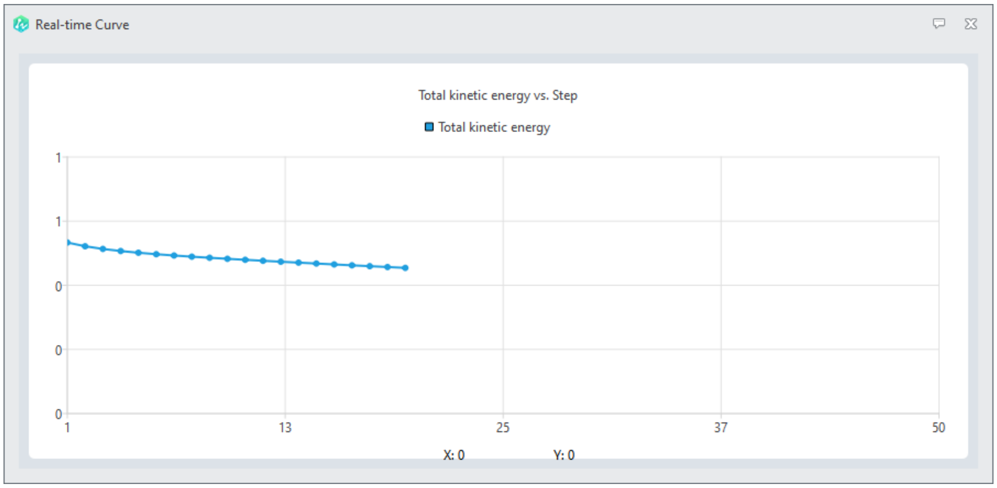
【Note】:

- This function is supported in the Nonlinear Statics, Nonlinear Dynamics, and Drop Test modules.

【Example】:



Convergence Curve

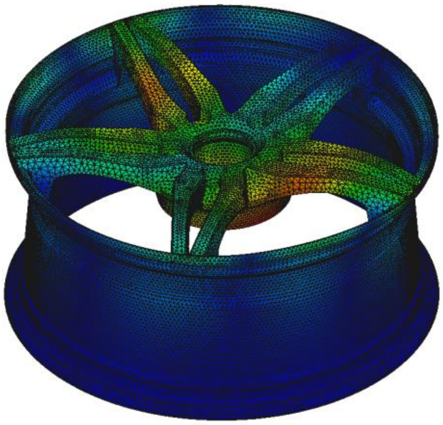


Energy Curve

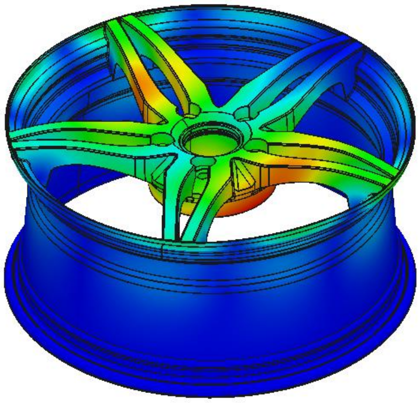
7.2.4.3 Added Feature Line Display and Wireframe Display

The new version extends the display methods of mesh and post-processing contour plots. Currently, there are five display methods for users to choose from: Shaded Elements and Mesh Lines, Shaded Elements and Feature Lines, Shaded Elements, Wireframe Elements Skin Only, and Wireframe Elements.

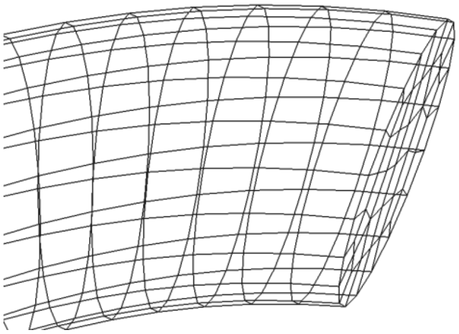
【Example】：



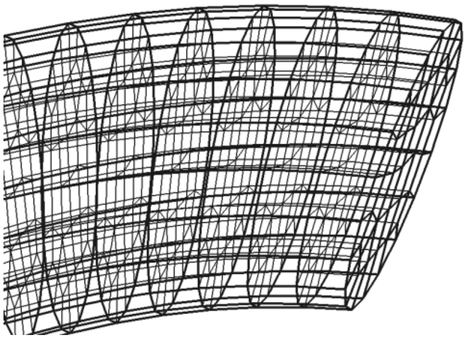
Shaded Elements and Mesh Lines



Shaded Elements and Feature Lines



Wireframe Elements Skin Only



Wireframe Elements

【Where is it】:

Part/Assembly Environment >> Simulation >> Shortcut Toolbar>> Shaded Elements and Mesh Lines >> Shaded Elements and Feature Lines >> Wireframe Elements Skin Only>> Wireframe Elements

7.3 ★Low Frequency Electromagnetic Simulation

7.3.1 ★Analysis Types

New low-frequency electromagnetic simulation analysis module has been added. The ZW3D low-frequency electromagnetic simulation plugin adopts the high-precision finite element (FEM) solving algorithm, supporting solving types, including two-dimensional and three-dimensional static magnetic fields, transient magnetic fields, electrostatic fields, and DC conducted electric fields. It can help users complete the electromagnetic field distribution, performance parameter simulation, and optimization analysis of multiple types of motors, transformers, actuators, sensors, wireless charging, and other electromechanical and power equipment.

Applicable scenarios include various types of motors (permanent magnet motors, induction motors, linear motors, etc.), transformers, reactors, lightning arresters, insulation sleeves, actuators, electromagnetic sensors, and wireless charging equipment.

7.3.1.1 Analysis of Electrostatic

The electrostatic field solver is used to calculate the two-dimensional or three-dimensional electrostatic field of static charge distribution and applied potential. It can be used to solve quantities such as potential, electric field, energy, surface charge density, and capacitance matrix.

【What can be done】：

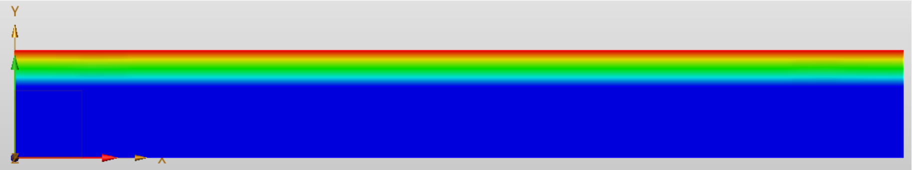
- ✓ Support various excitation conditions such as voltage excitation, charge excitation, floating conductor, charge density, etc.
- ✓ Support viewing of post-processing field map distribution results for electric fields, electric displacement vectors, and electric potential.
- ✓ Support solving in 2D XY coordinate system, 2D RZ cylindrical coordinate system, and 3D Cartesian coordinate system.

【Note】：

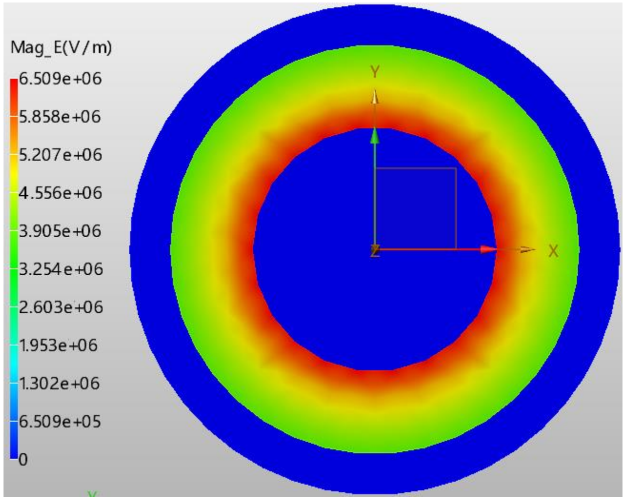
- When simulating 2D electrostatic fields, attention should be paid to the coordinate system defined by the material when setting up the material.
- When simulating 2D electrostatic fields, attention should be paid to defining the priority of materials when dividing the grid, to achieve automatic cutting of overlapping areas of materials.
- When simulating 3D electrostatic fields, it is necessary to pay attention to the use of compatible pairs during the meshing process to achieve compatible meshing.

【Example】：

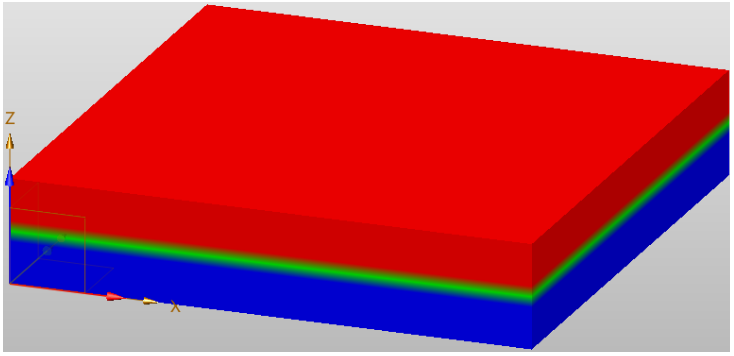
1. Electric potential analysis of 2D flat plate capacitors.



2. Analysis of 2D coaxial electric field intensity amplitude.



3. Electric potential analysis of 3D flat plate capacitors.



【Where is it】：

Part/Assembly Environment >> Simulation >> ZwMetas 2D >> Electrostatic (Cartesian, XY) / Electrostatic (Cylindrical about Z)

7.3.1.2 Analysis of DC Conduction

The DC conduction solver can calculate the steady-state two-dimensional or three-dimensional electric field in a conductor under current source excitation or external potential, and the quantity solved is the potential; The electric field and current density can be automatically calculated based on the potential, and the resistance matrix can be derived.

【What can be done】:

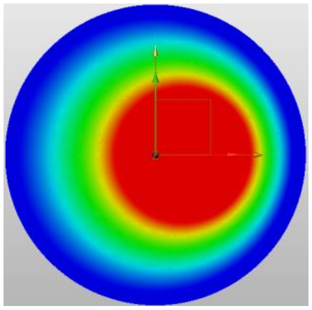
- ✓ Support voltage excitation and current excitation.
- ✓ Support viewing of post-processing field map distribution results for electric fields, currents, and electric potential.
- ✓ Support solving in 2D XY coordinate system, 2D RZ cylindrical coordinate system, and 3D Cartesian coordinate system.

【Note】:

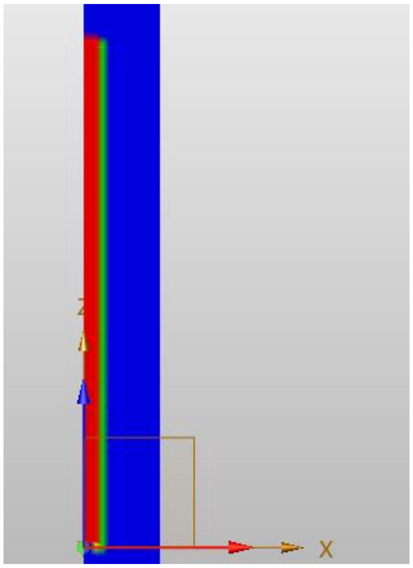
- When simulating 2D DC conduction, attention should be paid to the coordinate system defined by the material when setting up the material. When dividing the grid, attention should be paid to defining the priority of materials to achieve automatic cutting of overlapping areas of materials.
- When simulating 3D DC conduction, it is necessary to pay attention to the use of compatible pairs during the meshing process to achieve compatible meshing.

【Example】:

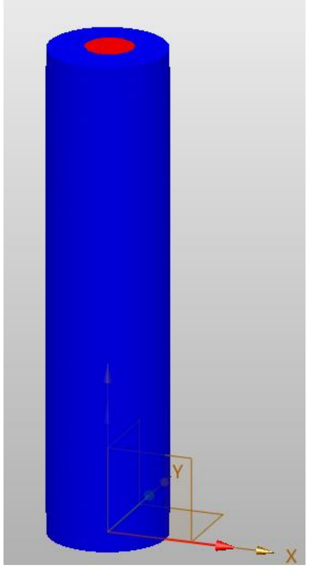
1. Electric potential analysis of 2D eccentric coaxial line under voltage excitation.



2. Electric potential analysis of solenoid in 2D cylindrical coordinate system.



3. DC Conduction analysis of 3D Coaxial electric potential under voltage excitation



【Where is it】:

Part/Assembly Environment >> Simulation >> ZwMetas 2D >> DC Conduction (Cartesian, XY)/ DC Conduction (Cylindrical about Z)

7.3.1.3 Analysis of Magnetostatic

The Magnetostatic field solver is used to solve two-dimensional or three-dimensional static magnetic field distribution problems. The source of the static magnetic field can be the DC current in the conductor, the static external magnetic field represented by boundary conditions or the permanent magnet. Through sharing, parameters such as magnetic flux density and vector magnetic potential in space can be calculated.

【What can be done】:

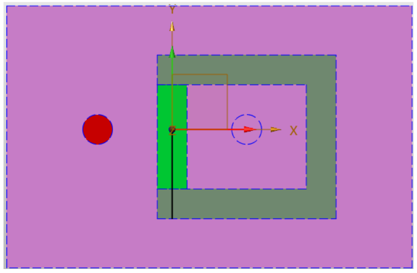
- ✓ Support current excitation, surface current density excitation, and permanent magnet excitation.
- ✓ Support viewing of post-processing field map distribution results for magnetic vector contour lines and magnetic flux density.
- ✓ Support solving in 2D XY coordinate system, 2D RZ cylindrical coordinate system, and 3D Cartesian coordinate system.

【Note】:

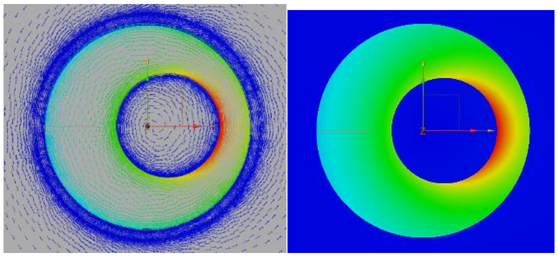
- When simulating 2D Magnetostatic, attention should be paid to the coordinate system defined by the material when setting up the material. When dividing the grid, it is necessary to pay attention to defining the priority of materials to achieve automatic cutting of overlapping areas of materials.
- When simulating 3D Magnetostatic, it is necessary to pay attention to the use of compatible pairs during the meshing process to achieve compatible meshing.

【Example】:

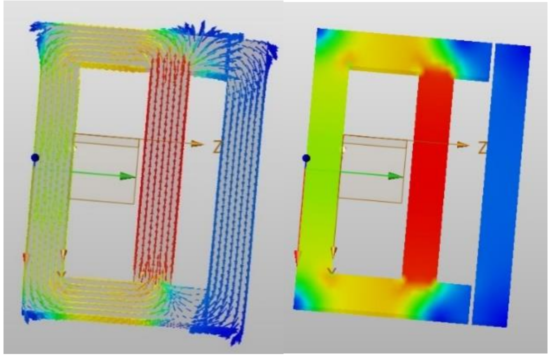
1. 2D Magnetostatic analysis under permanent magnet excitation.



2. Vector and amplitude analysis of spatial magnetic flux density under current excitation on a 2D eccentric coaxial line.



3. Magnetostatic analysis of permanent magnets and magnetic coils in a 3D coordinate system



【Where is it】:

Part/Assembly Environment >> Simulation >> ZwMetas 2D >> Magnetostatic (Cartesian, XY)/ Magnetostatic (Cylindrical about Z)

7.3.1.4 Analysis of Transient

The transient magnetic field solver is used to calculate the two-dimensional time-domain magnetic field. The source of a magnetic field can be time-varying currents and voltages that move or do not move, permanent magnets and/or coils that move or do not move, and external coupling circuits that move or do not move. It can be used to solve physical quantities such as force, torque, back electromotive force, energy, winding flux, and winding-induced electromotive force.

【What can be done】:

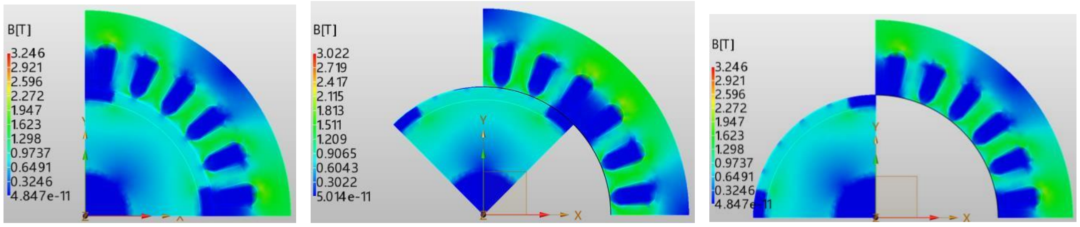
- ✓ Support coil voltage excitation, coil current, and permanent magnet excitation.
- ✓ Support the calculation of eddy current effects, Y-connections, and loss materials.
- ✓ Support viewing of post-processing field map distribution results for magnetic vector contour lines and magnetic induction intensity.
- ✓ Support 2D XY coordinate system and 2D RZ cylindrical coordinate system solution.

【Note】:

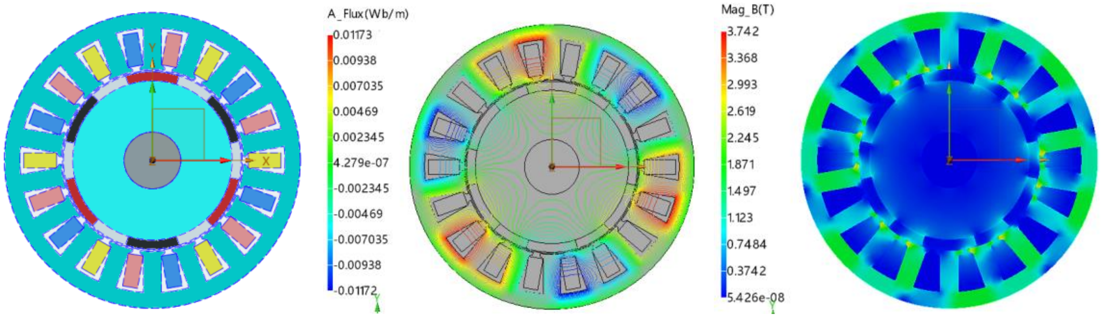
- When simulating 2D Transient, attention should be paid to the coordinate system defined by the material when setting up the material.
- When simulating 2D Transient, it is necessary to pay attention to defining the priority of materials when dividing the grid, to achieve automatic cropping of material overlapping areas.
- When simulating 2D Transient, attention should be paid to setting reasonable solution time steps and total solution time to ensure that calculations can be completed under current computing resources.
- When simulating 2D Transient, it is necessary to choose whether to save the calculation field for each step based on actual needs, to avoid causing the result file to be too large and saving a pile of useless data.

【Example】:

1. 2D Transient simulation of motor based on periodic model.


B[T]
3.246
2.921
2.596
2.272
1.947
1.623
1.298
0.9737
0.6491
0.3246
4.847e-11
B[T]
3.022
2.719
2.417
2.115
1.813
1.511
1.209
0.9065
0.6043
0.3022
5.014e-11
X
B[T]
3.246
2.921
2.596
2.272
1.947
1.623
1.298
0.9737
0.6491
0.3246
4.847e-11
X

2. 2D Transient simulation of a full model permanent magnet synchronous motor.


X
A_Flux(Wb/m)
0.01173
0.00938
0.007035
0.00469
0.002345
4.279e-07
-0.002345
-0.00469
-0.007035
-0.00938
-0.01172
Mag_B(T)
3.742
3.368
2.993
2.619
2.245
1.871
1.497
1.123
0.7484
0.3742
5.426e-08

【Where is it】:

Simulation >> ZwMetas 2D >> Transient (Cartesian, XY) / Transient (Cylindrical about Z)

7.3.2 Analysis Functions

7.3.2.1 Function of Motor Library

The software has a built-in motor template library module, allowing users to quickly generate corresponding motor models through simple parameter definitions without the need for manual modeling processes.

【What can be done】：

- ✓ Support parameterized one-click generation of motor models.
- ✓ Support the generation of over 100 types of motor templates.

【Where is it】：

Simulation >> ZwMetas 2D >> Transient >> Motor Design

7.3.2.2 Function of Skew Model Calculation

Support the skew model calculation function, by which users can define continuous and segmented skew models.

【What can be done】：

- ✓ Support continuous and segmented skew model settings and calculations.

【Where is it】：

Simulation >> ZwMetas 2D >> Transient >> Design Settings

7.3.2.3 Function of Loss Analysis

Support loss analysis and calculation function, which can calculate iron loss, eddy current loss, and copper loss.

【Where is it】：

Simulation >> ZwMetas 2D >> Transient >> Excitations >> Set Core Loss

7.3.2.4 Lamination Material

Support Lamination and LitzWire composition more accurately describe the parameters of electromagnetic materials.

【Where is it】：

Simulation >> ZwMetas 2D >> Transient >> Material >> Composition

7.3.2.5 Lossy Material

Support iron loss models, including electrical steel and power ferrite models. Provide more accurate description of the loss characteristics of electromagnetic materials.

【Where is it】：

Simulation >> ZwMetas 2D >> Transient >> Material >> Core Loss Model

7.3.3 ★Mesh

7.3.3.1 Function of Auto Trim

Support the automatic geometric cropping function. During the modeling process, there is no need to perform special geometric Boolean operations on overlapping geometric models. The software can automatically perform geometric materials based on material priority.

【What can be done】：

- ✓ The processing of geometric overlapping module is realized automatically.
- ✓ The user does not need to perform manual Boolean processing of model overlap.

【Where is it】：

Simulation >> ZwMetas 2D >> Mesh >> Auto Trim Setting