# ZW3D WHAT'S NEW

*V 2026*

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# ZW3D™ V2026 What's New

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# ZW3D 2026 Highlight Improvement

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

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

ZW3D 2026 focuses on enhancing the core modeling capabilities such as fillet, sweep and plastic engineering features; improving assembly constraint efficiency and usability to enable smarter assembly design; Further improving the drawing efficiency and annotation usability of drawings sheet, support 2D / 3D linkage update; delivering enriched industry-specific toolkits ,including harness, mold & die, sheet

metal, and motion simulation, among others; realizing more efficient and convenient CAM programming, and supporting efficient finishing strategies;and offering enhanced PDM capabilities to strengthen collaboration and data management. Additionally, optimized user experience and performance upgrades accelerate product design cycles while boosting design efficiency, empowering enterprises to achieve cost reduction and efficiency improvement.

✓ **Smarter Design, Enhanced Efficiency.** ZW3D 2026 provides more automated and intelligent tools in assembly design and engineering drafting to help users complete product design more efficiently. For example: the assembly module supports intelligent constraint type inference and batch copying of components with constraints;drawing sheets offer enhanced automatic drawing generation and dimensioning capabilities; assembly constraints can be automatically converted to kinematic pairs with one click; and rule-based automatic routing is supported.

✓ **Optimized user experience to make it easier to use.** ZW3D 2026 leverages process simplification and enhancement of parts and assembly design, industry toolset, drawing sheet, and CAM machining design to complete more complex product designs in more simplified steps. For example, the workflow of common functions such as drafting, stiffeners, and assembly constraints is simplified, core modeling capabilities such as fillets and sweeps are enhanced, the ease of use of drawing views and annotations is improved, and the interactive experience is optimized.

✓ **Enhance multi-party collaboration and strengthen data management.** ZW3D 2026 offers deep integration with ZWTeammate, and by utilizing its real-time collaborative design and data management capabilities, it further enhances the efficiency of corporate data governance, promotes synergy between design and manufacturing, optimizes resource allocation and reduces costs, and facilitates more efficient, collaborative, and secure work in the innovation, research, and design processes of enterprises.

# Top Enhancements

ZW3D 2026 focuses on enhancing the capabilities of the software and improving ease of use, while providing more powerful innovative features. Items marked with "★" in the text are the top enhancements for this version.

| | |
|---|---|
| **Basic:** | ★1.4.2 Optimized Config Table |
| | ★1.5.6 New Part Clearance Check |
| | ★1.5.7 Optimized Section |
| | ★1.6.1 Mass Properties Optimization |
| | ★1.8.1 Graphic area micro-panel interactive application |
| **Shape Design:** | ★3.2.2 Fillet Optimized |
| | ★3.2.4 “Var Sweep” Improvement |
| | ★3.2.5 “Draft” Improvement |
| | ★3.2.6 Thread adds standard profile |
| | ★3.2.7 “Cylindrical Bend” Improvement |
| | ★3.2.13 Slot Feature |
| | ★3.2.14 New Mounting Boss |
| | ★3.2.15 New Snap Hook |
| | ★3.2.16 New Snap Hook Groove |

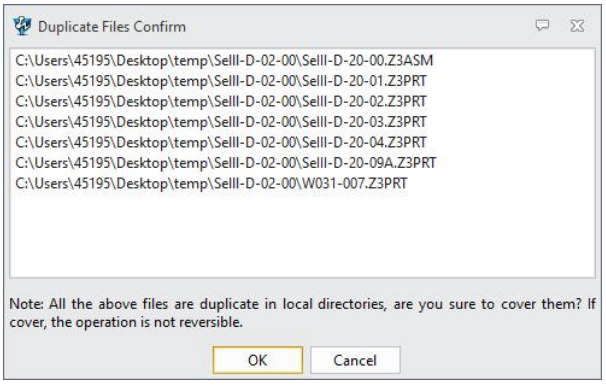
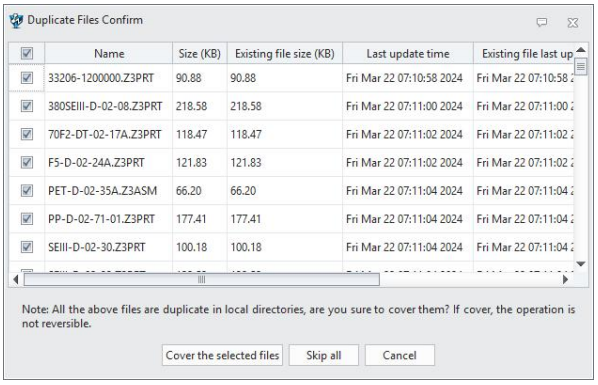
| | |
|---|---|
| **Assembly Design** | ★3.3.1 Assembly Constraints |
| | ★3.3.2 Copy with Constraint |
| | ★3.3.3 Assembly mirror optimization |
| | ★3.3.9 Smart Fastener Optimization |
| **Drawing Sheet** | ★3.4.1 2D/3D Support For Association Updates |
| | ★3.4.2 Optimization of projection efficiency |
| | ★3.4.4 Annotation Optimization |
| **Sheet Metal Design** | ★4.1.1 Convert to sheet metal Improvement |
| **Structure** | ★4.2.2 Function of Reusing and Replacing profiles |
| **Wiring Harness Design** | ★4.4.2 Path and Route |
| **ECAD** | ★4.5.2 Compare IDF |
| | ★4.5.3 IDX Import and Export |
| **Simulation** | ★7.1.1 Convert |
| | 7.4★ Fluid Simulation |
| **PDM** | 8.5★ Check-in |
| | 8.9★ Embedded Browser |

# 1 Basic

## 1.1 File Management

### 1.1.1 Optimization of Operations when Packing Duplicate Files

When performing file packaging, if files with the same name exist in the target path, the system will display file size and last update time for comparison, and support three operations: Cover the selected files, Skip all , and Cancel.

| ZW3D2025 | ZW3D2026 |
| --- | --- |
|  |  |





**【What users can do】**

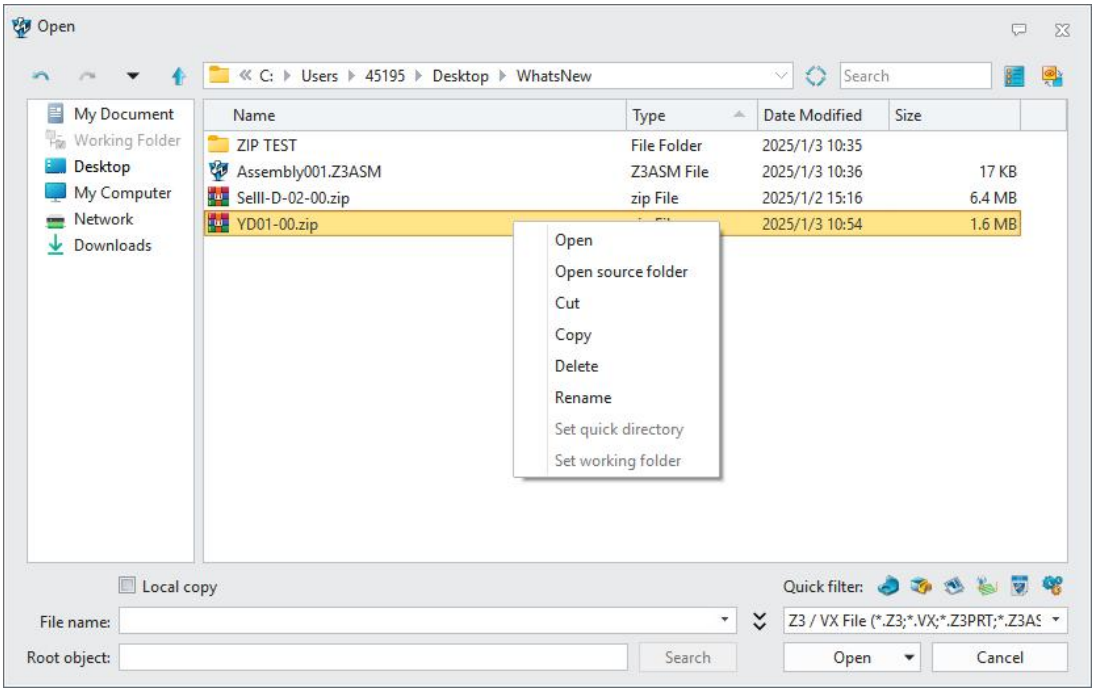
✓ When packaging files, you can choose whether to overwrite files with the same name, supporting either overwriting all or selective overwriting.

**【Where is it】**

File >> Pack

### 1.1.2 Support Direct Opening of Files within ZIP

In the "Open" command panel, you can directly open a ZIP file without unzipping it, and open the files in it in read-only mode.




**【What users can do】**

✓ In ZW3D file manager, you can double-click or right-click to open ZIP files and then open the files inside

**【Notes】**

- The unzipped ZIP folder can only be used as a temporary folder.
- The model files opened in this way are all in read-only mode and will not be automatically saved locally. Please save the files you need to keep to your working folder by using the "Save As" function.
- Multipart compressed ZIP files or ZIP files with compression passwords are not supported.

**【Where is it】**

File >> **Open**

## 1.2 Clipboard

Clipboard can be saved with a file and pasted into a specified file.

**【What users can do】**

✓ Select the sketch/drawing outline/3D solid and copy it to the clipboard of one object file to save with the file.

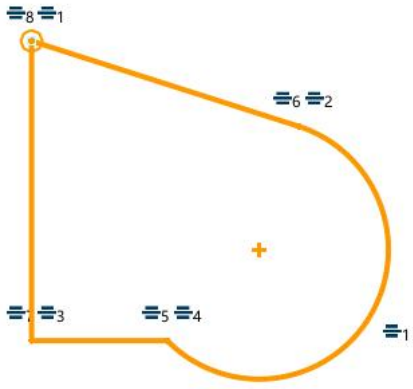
✓ Paste the clipboard into the drawing/sketch of the specified file to achieve rapid reuse.

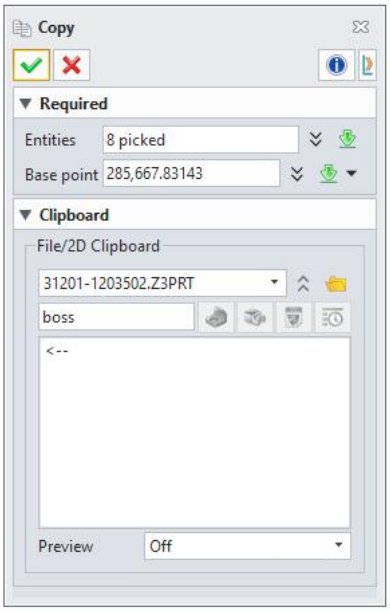
**【Example】**

1) Select the 2D contour that needs to be frequently reused and initiate the copy command using Ctrl+C.

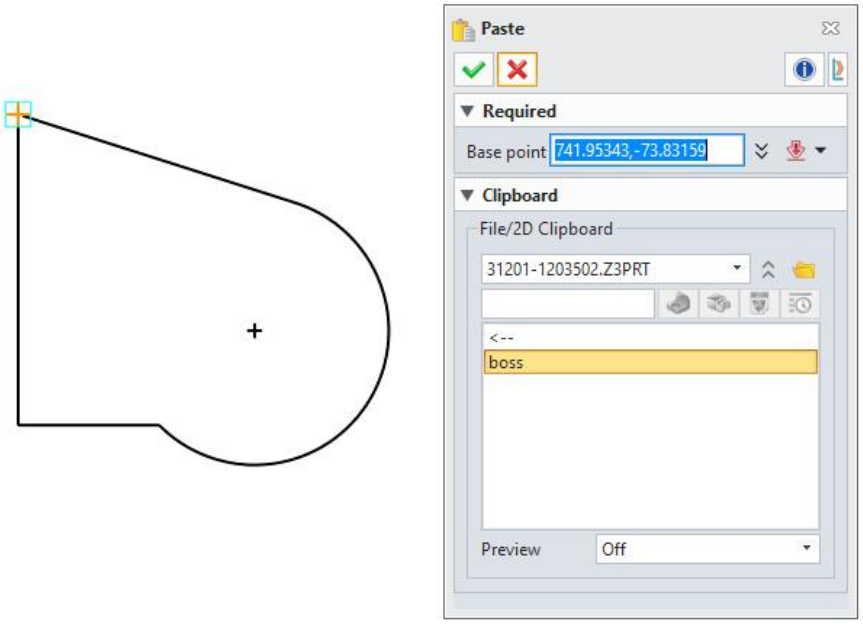
2) Specify the base point location and the file to be used as the clipboard.

3) Enter a name that has not been used by other clipboards in the file, and then you can create the corresponding clipboard in the specified file.




4) Use Ctrl+V to bring up a clipboard in the sketch/drawing environment, select the clipboard under the specified file to paste the copied 2D contour.




**【Where is it】**

Part/Sketch/Drawing environment >> Edit >> **Copy（Ctrl+C）**

Sketch/Drawing environment >> Edit >> **Paste（Ctrl+V）**

## 1.3 Optimized Datum Plane

The datum function is improved, and the datum axis and external geometric objects can be selected as references, which can create the target datum more quickly and accurately.

**【What users can do】**

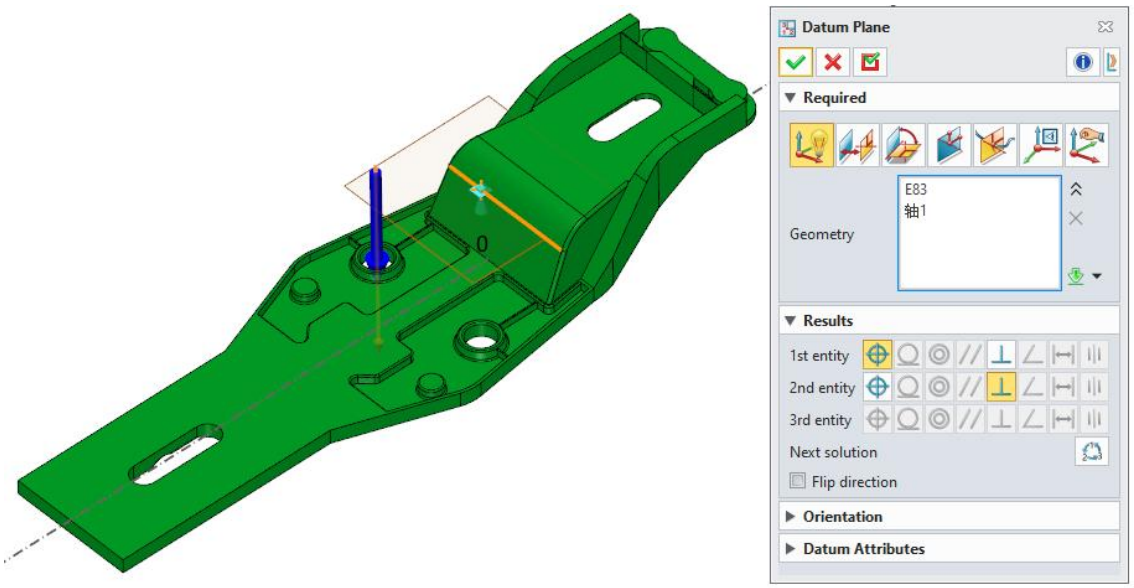
- ✓ Select axis (including the axes of the sketch) to create a plane
- ✓ Select external geometric object
- ✓ Displays the outline of the reference plane

**【Example】**

1) In the design process, it is necessary to create a datum plane that coincides with the straight line and is perpendicular to the axis line.

2) Select the straight line and the axis as reference objects at the same time, and set the

corresponding constraint relationships.




**【Notes】**

- It can't create a datum plane when only one axis is selected, because the origin of the plane should be defined.

**【Where is it】**

Part Environment >> Shape >> Datum >> **Datum Plane**

## 1.4 Tools

### 1.4.1 User Folder Optimized

By optimizing the user directory structure, normalizing the usage logic, and increasing the restrictions on modifying rules, this new ZW3D version can manage user files and data more effectively and reduce the risk of software crashes or abnormal operation.

**【What users can do】**

✓ If the user folder causes the program to be unable to start, the user folder will be automatically backed up and the initial user folder will be used to try to start the program.

✓ The files of the temporary folder are deleted when the program is closed.

✓ The Output folder is not allowed to affect the running of the program, which improves the stability of the program.

**【Example】**

1) The user directory of the industry module is adjusted to the user folder manager. For example, the Routing user directory is changed to the user folder.

**【Notes】**

- The old version of the user folder cannot be directly used for the new version of the user folder.

**【Where is it】**

Part/Assembly Environment >> Tools >> Utilities >> **User Folder Manager**

### 1.4.2 ★Config Table Optimized

In the process of parts and assembly design, the optimized configuration table function realizes the correct linkage between the drawing object and the part, which is very conducive to the design of multiple configurations.

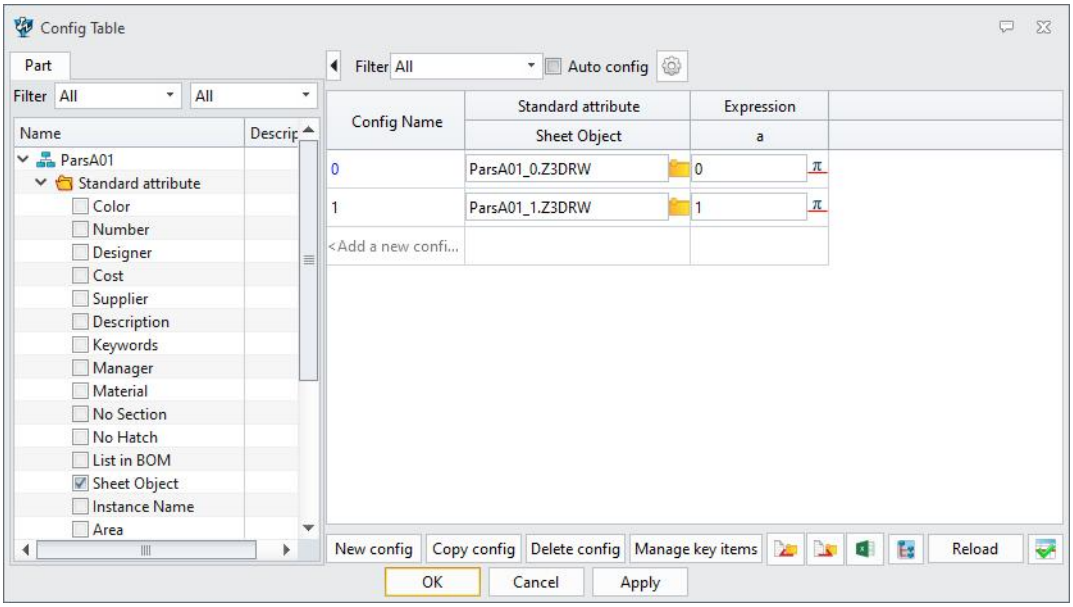
**【What users can do】**

✓ When the name of the configuration is modified, the view configuration of the sheet is automatically associated

✓ Editing parts in the drawing view will automatically recognize the corresponding configuration

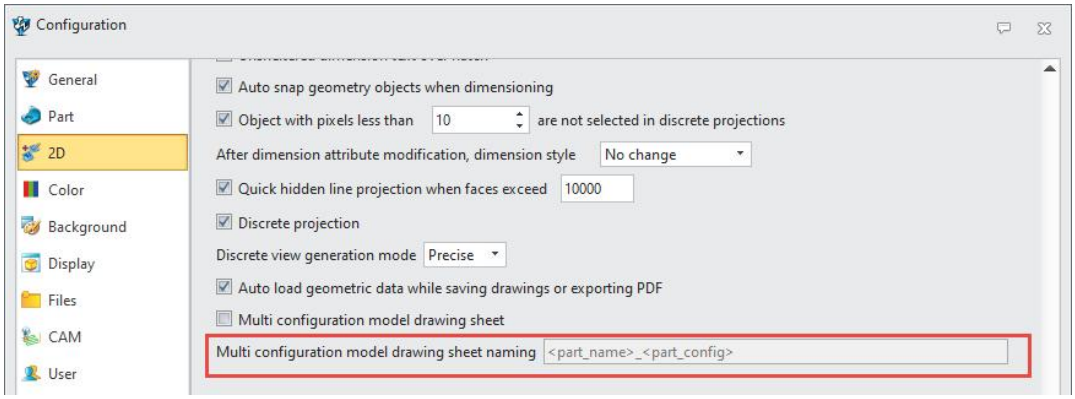
**【Example】**

In the process of part design, different configurations correspond to different drawing objects, and the mutual switching between different drawings and parts can be automatically changed to the corresponding configuration of the part.




**【Notes】**

- Users need to check "Multi configuration model drawing sheet naming" in configuration - 2D - Drawing sheet, so as to enable this function.




**【Where is it】**

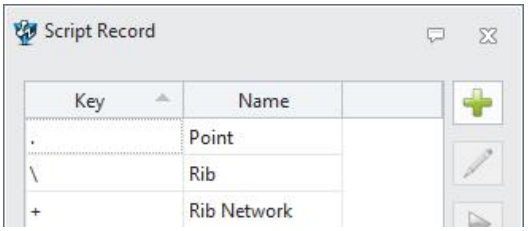
Part/Assembly Environment >> Tools >> Insert >> **Config Table**

## 1.4.3 Script Optimized

Script record supports more keyboard special characters, providing users with more shortcut setting options and improving design efficiency.

**【What users can do】**

- ✓ Such as: script record “-”,“+”,“/”,“*”,“.” ,“,”,“.”,“;”,“’”,“\”,“[” ,“]”and so on.




**【Example】**

In the design process, if the rib command is frequently used. The user can make the script record of "/" first, and then use the rib function by the key of "/".

**【Notes】**

- It needs to be used under the English input method.

**【Where is it】**

Part/Assembly/Sketch Environment >> Utilities >> **Script Record**

## 1.5 Inquire

### 1.5.1 Measurement Upgrade

Inquire function is optimized to cover more measurement scenarios. The measurement capability of the sketch environment remains the same as that of the part and assembly environment. At the same time, the measurement ability of the surface and the transformation ability of the results of the angle measurement are strengthened.

**【What users can do】**

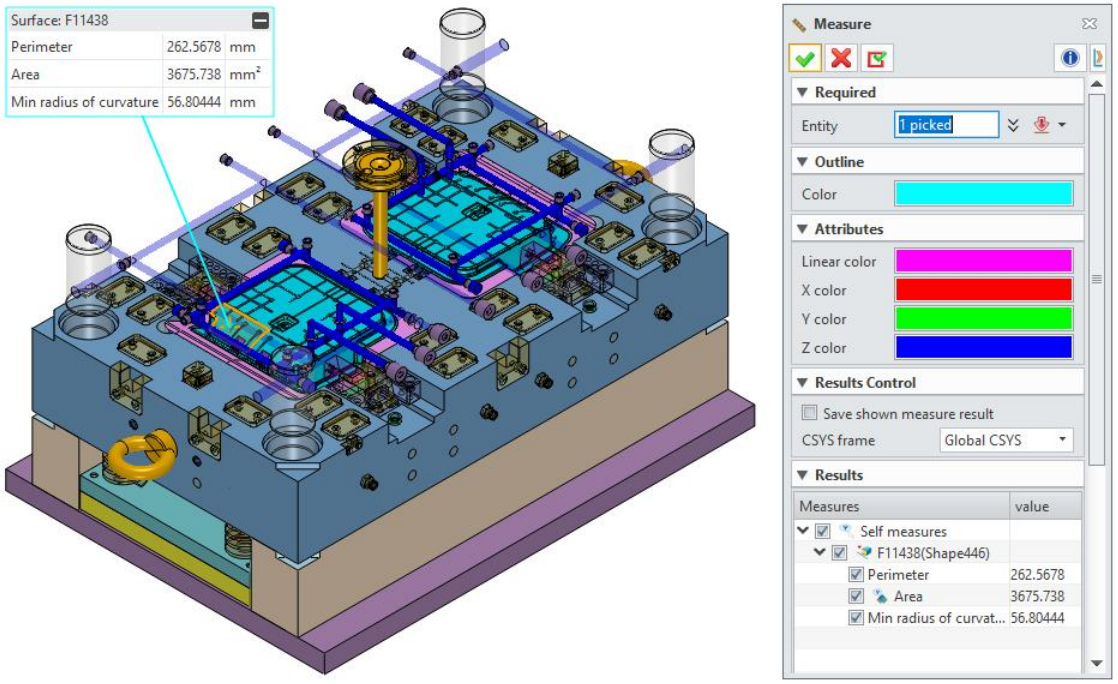
- ✓ Measure distances between two faces in the sketch environment.
- ✓ Measure the minimum radius of curvature of the surface
- ✓ Angle measurement supports projection angle

**【Example】**

1) When sketching, measure the distance value between the key surfaces and draw the sketch based on the distance value.

2) When the part is being designed, measure the projection angle of the angle of the edge of the surface to the XY plane.

3) When designing a surface, measure the minimum radius of curvature of the designed surface.




**【Where is it】**

Part/Assembly Environment >> Inquire >> Measure >> **Measure**

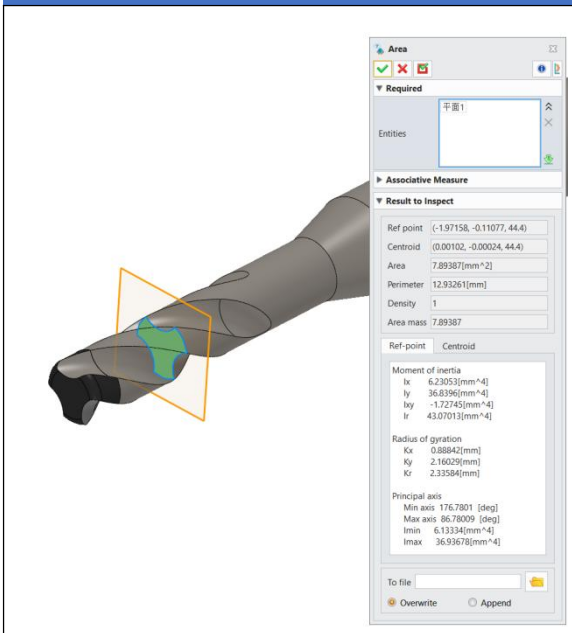
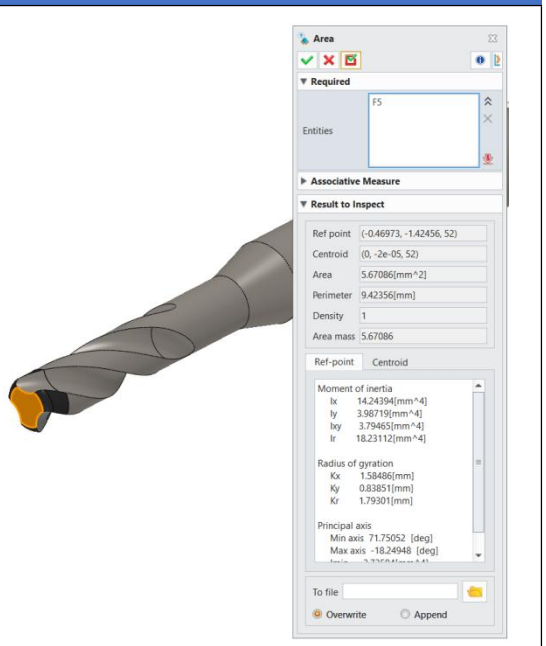
Part/Assembly Environment >> Inquire >> Measure >> **Measure**

Sketcher Environment >> Inquire >> **Measure**

## 1.5.2 Area Measurement Tool Improvement

Area measurement is improved and now supports measuring information such as area and moment of inertia by picking up datum plane or planes. When picking up datum plane, all shapes' section will be included and measured; when picking a plane, multiple coplanar objects can be picked for

measurement.

| Datum | Plane |
| --- | --- |
|  |  |

**【What users can do】**

✓ Inquire the area, centroid, moment of inertia, and other information of the section taken from a datum plane.

✓ Select multiple coplanar planes and measure the area, centroid, moment of inertia, and other information of the selected bounded plane.

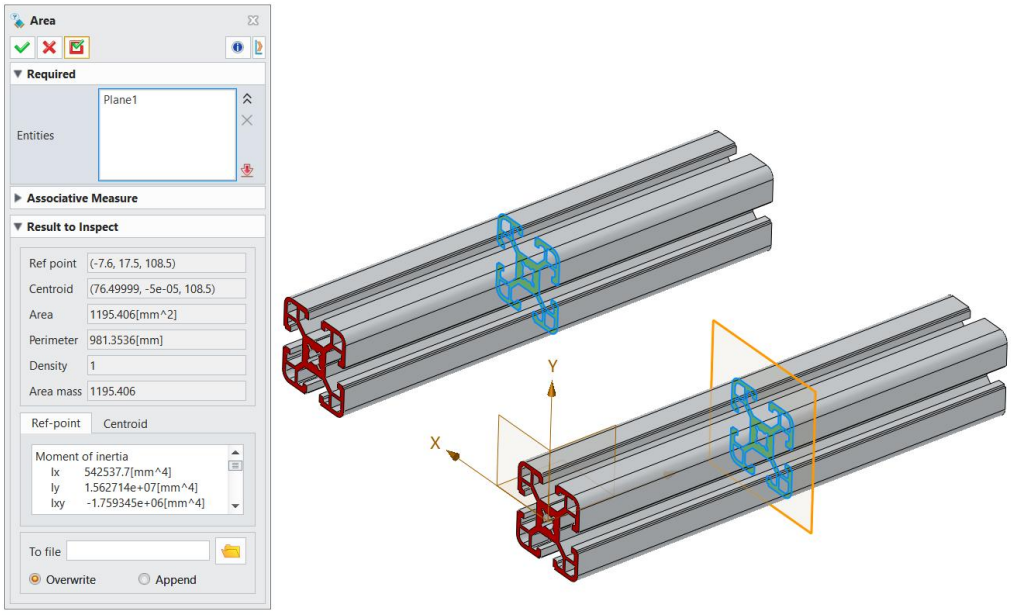
**【Notes】**

- The function doesn't support mixing the input of area measure tool such as datum/planes/curves.
- When multiple faces are selected and measured, they must be coplanar.
- The function doesn't support picking up 3D surfaces for area measurement. It is recommended to use "3D Surface" function to measure nonplanar area.

**【Example】**

Select the reference plane, measure the sum of the cross-sectional areas of all shapes intercepted

by the reference plane.




**【Where is it】**

Part/Assembly >> Inquire >> **Area**

## 1.5.3 Stock Size Measurement

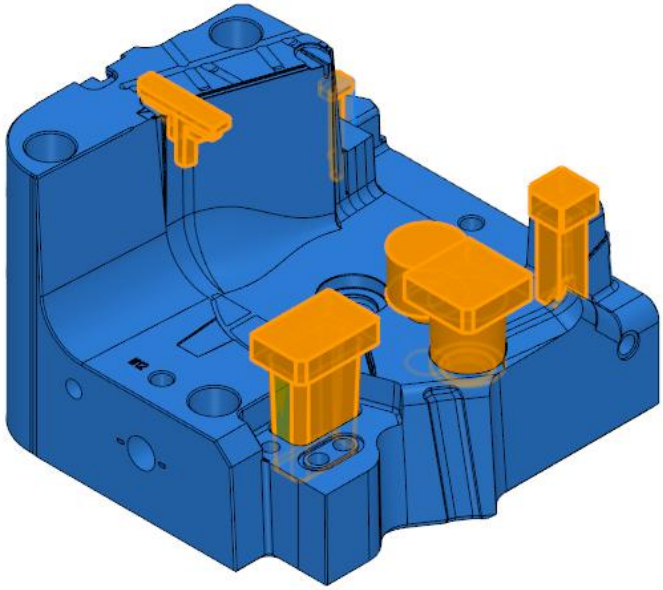
The new stock size function can help the user easily view the basic information of the stock. By selecting the target shape and surface, the stock size data can be quickly obtained.

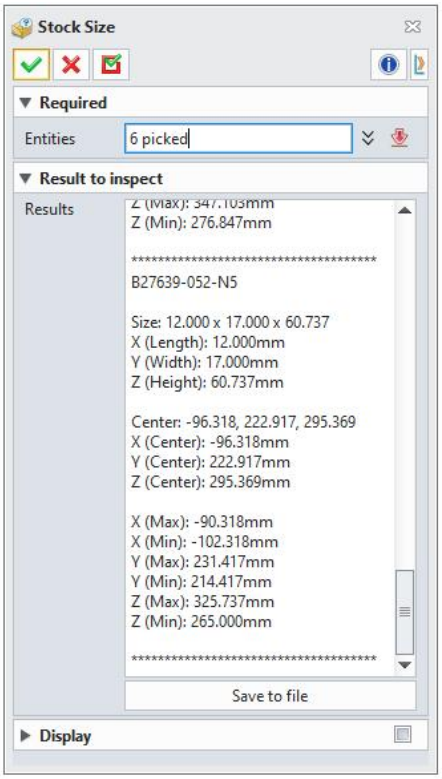
**【What users can do】**

- ✓ Inquire the length, width, height ,center coordinates,and maximum/minimum values along the X, Y, Z axes of the stock
- ✓ Continuous inquire
- ✓ Copy or export the inquire results

**【Example】**

When designing the electrode, it is necessary to measure the stock information of the electrode, and the size and coordinates can be quickly obtained through the stock size.






**【Notes】**

- The size of the stock is queried according to the cuboid, excluding the cylinder.

**【Where is it】**

Part/Assembly Environment >> Inquire >> Inspect Entities >> **Stock Size**

## 1.5.4 Curvature Plot Supporting Projection

The optimized Curvature plot function supports selecting the curvature distribution of the curvature comb in the specified plane. The above can be used to guide the modification of the curve and finally get a smoother surface.

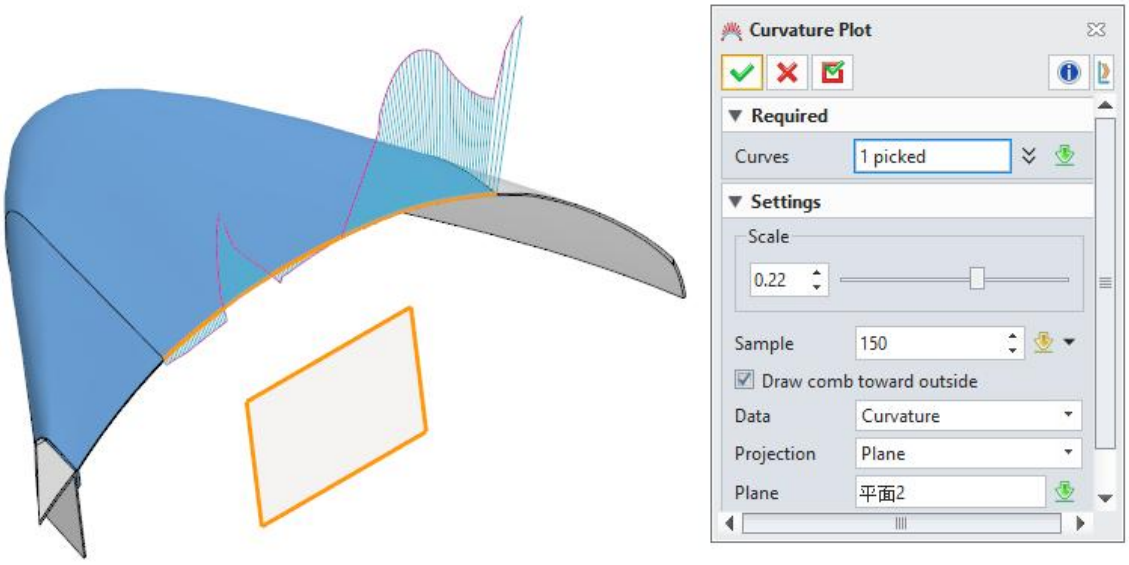
**【What users can do】**

✓ Get a curvature map of curves in a specified plane, view, or direction

**【Example】**

When designing the surface, you need to check the curvature information of the edges of the

structural surface in the specified plane.




**【Where is it】**

Part/Assembly Environment >> Inquire >> Inspect Entities >> **Curvature Plot**

## 1.5.5 Compare Parts Optimized

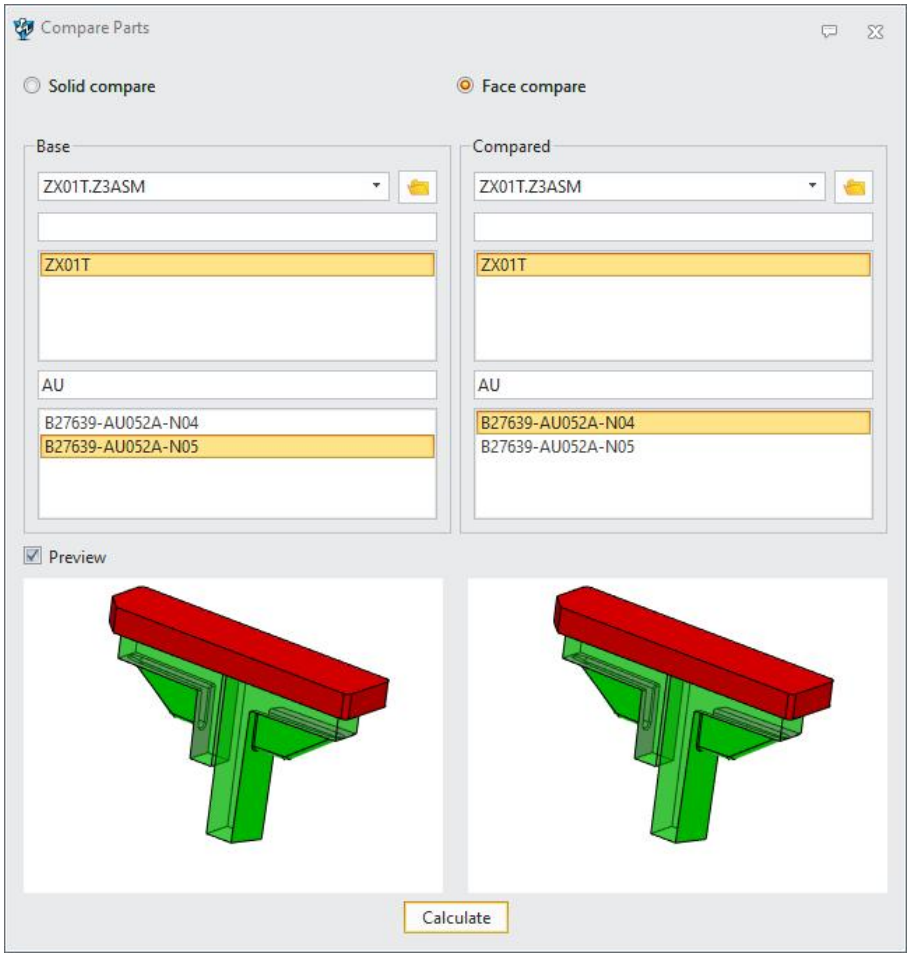
The Compare Parts function supports modeling comparison within the part during the multi-body design process. For example, compare whether there is a difference in the modeling entity within the part, or whether there is a difference in the modeling surface within the part.

**【What users can do】**

- ✓ The comparison mode can be set to Solid compare or Face compare
- ✓ When there are multiple entity models, you can find the target model by searching with keywords.

**【Example】**

During electrode design, users can compare electrode similarity by utilizing keyword searches to locate target geometries for comparison.




**【Where is it】**

Part/Assembly Environment >> Inquire >> Inspect Model >> **Compare Part**

## 1.5.6 ★New Part Clearance Check Function

In the part environment, the new part clearance check function can analyze the position relationship between multiple bodies, and can accurately identify the position relationship between two shapes as interference, contact or gap. For modeling entities with gaps, the corresponding gap values can also be accurately calculated.

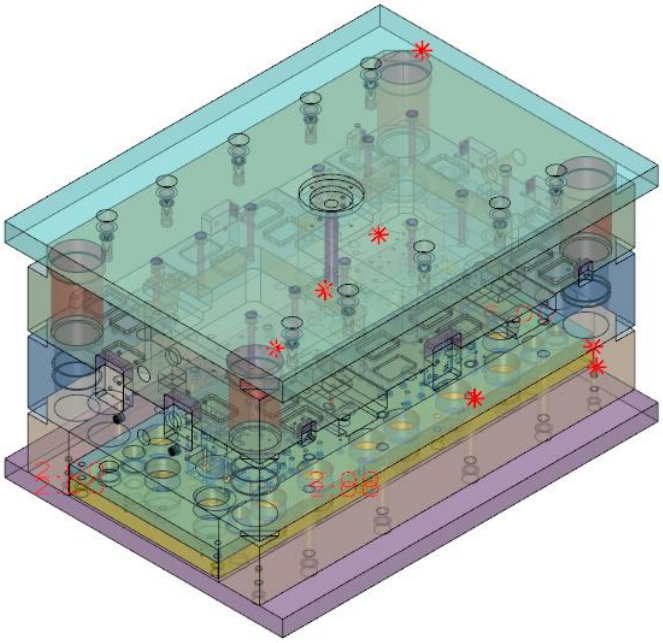
**【What users can do】**

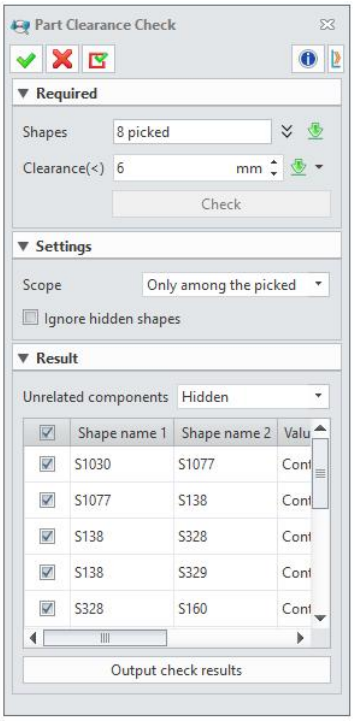
- ✓ Support to select multiple shapes and check the position relationship between pairs
- ✓ Set a gap value to perform checks only on modeling objects below the specified gap value

✓ Support ignoring hidden shapes, marking analysis results in the plot area and quick positioning

**【Example】**

In the mold design process, check whether the gap between the mold molding parts meets the need.






**【Where is it】**

Part Environment >> Inquire >> Inspect Model >> **Part Clearance Check**

## 1.5.7 ★Section View Optimized

In scenarios where there is interference in the section view, section view provides a list-based search method that supports quickly finding the components associated with the interference area and calculating the area of the interference. It allows users to select the corresponding interference from the list, highlighting it in the drawing area.

In addition, section view adjusts the default start position of the section plane offset accordingly,

you can adjust the section position more easily with the specified alignment plane as the starting position. And the handles are displayed in the drawing area as much as possible.

**【What users can do】**

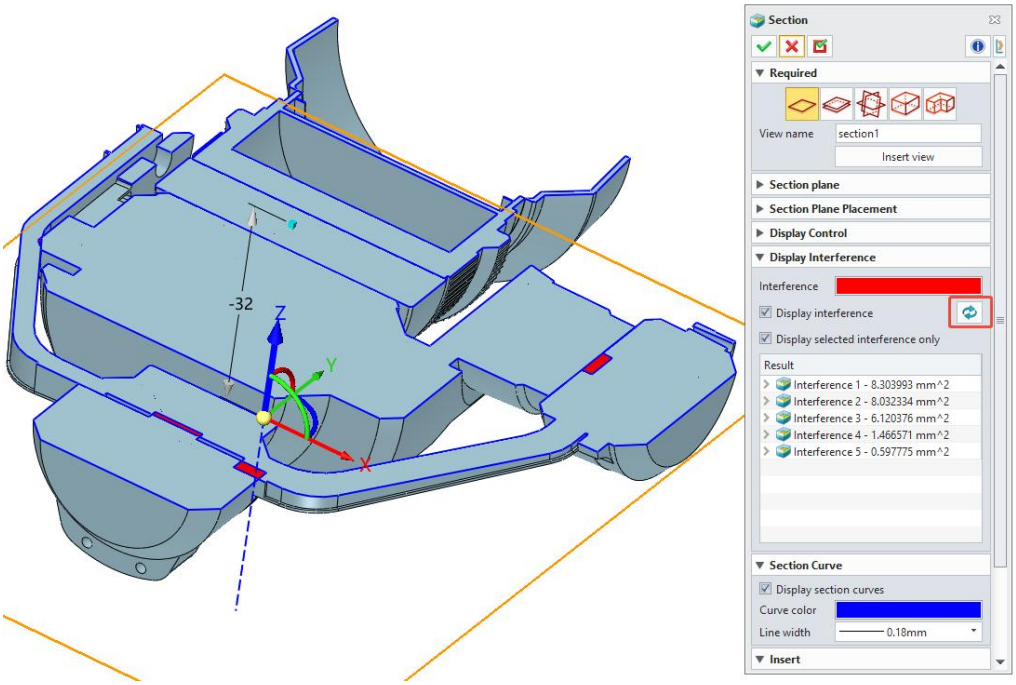
- ✓ View interference-related objects in the cut area
- ✓ View the interference area of the cut area.
- ✓ When scaling the model, the handle display in the drawing region as much as possible
- ✓ Section with profile supports the exclusion of a component
- ✓ When specifying an aligned face, the offset value is 0 mm

**【Notes】**

- Only single plane section supports the viewing of section interference results.

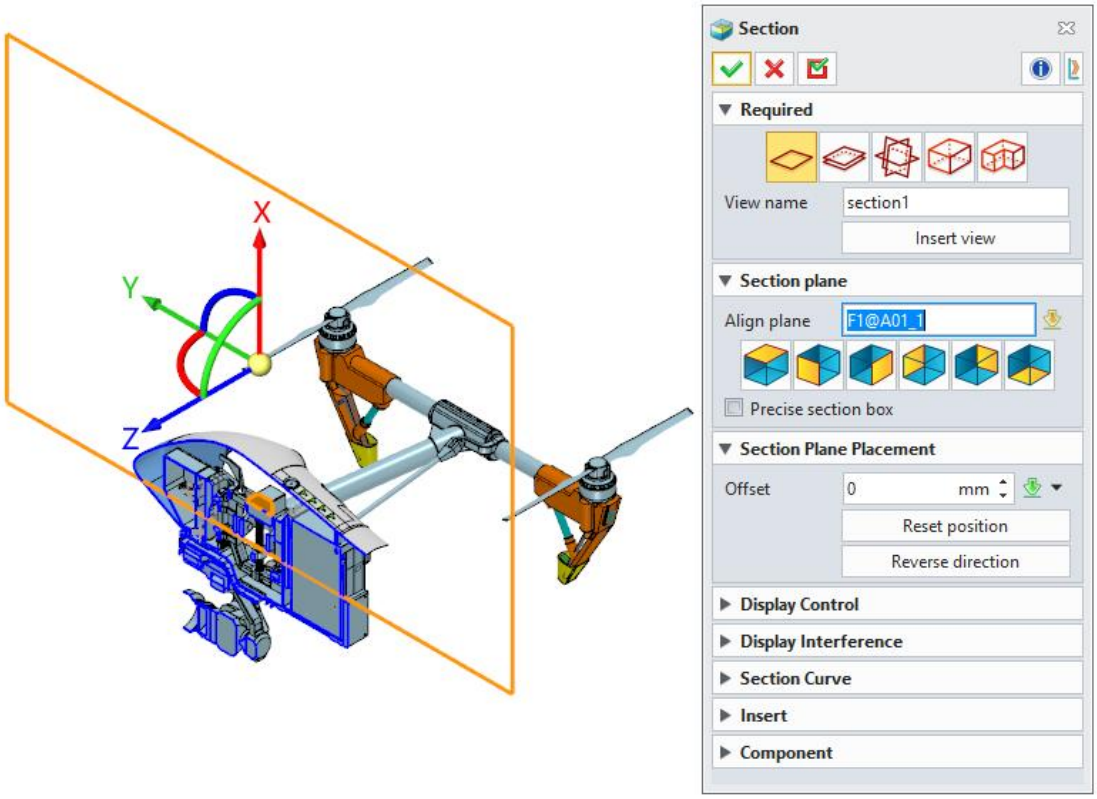
**【Example】**

1) When there is interference in the profile view, the current interference area can be calculated and the results displayed in the interference list..




2) When setting up a cross-sectional view through a plane, the alignment plane is the starting

position, with an offset value of 0.




**【Where is it】**

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

## 1.6 Attributes

### 1.6.1 ★Mass Properties Optimize

The mass properties function has been enhanced, with the addition of a centroid object and its management folder. The centroid is capable of updating with the quality attribute changes. Moreover, the centroid supports annotation and measurement.

When multiple entities have different materials, the material and density of the file will be annotated as "various".

**【What users can do】**

✓ Mass properties can be calculated according to the current visible and hidden state of components/entities.

✓ If the material of the current file is empty or deleted, the total mass and volume of the current file will be calculated based on the mass of the entities and components inside this file.

✓ Centroid object is calculated based on the latest model state

✓ Centroid object supports PMI annotation and measurement.

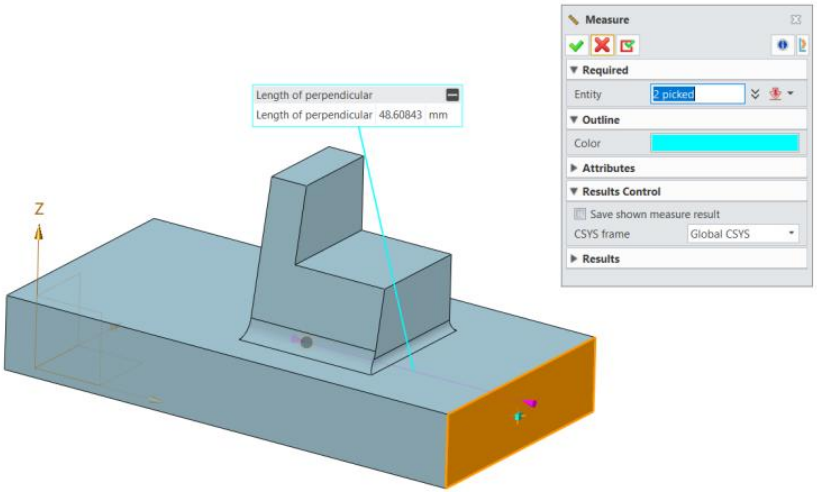
**【Notes】**

• In multi-shapes or in a file of plenty assembly environments, calculating mass properties in real time needs to refresh a large number of entities, which will cause significant efficiency problems. It is recommended to disable this function to avoid efficiency issues.

• When the material of the file is empty, mass properties will calculate the sum of the mass according to the shapes and components. Therefore, when all the material of shape/component is empty, the mass cannot be calculated.

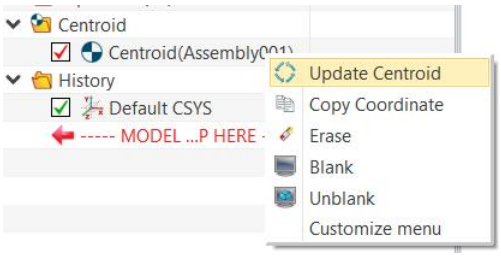
**【Example】**

1) Measure the distance from centroid to the specified plane or point or the coordinate of centroid.

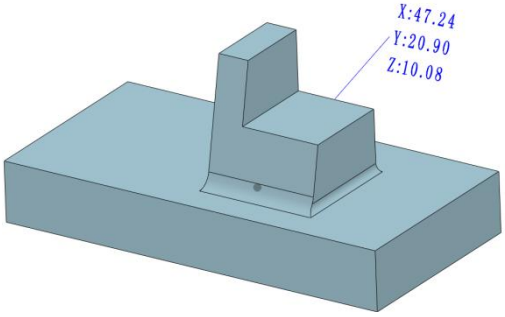



2) Use centroid object folder to manage centroid of current file such as updating centroid, copying

coordinate, blanking/unblanking centroid, erasing centroids.




3) Annotate centroid coordinates.

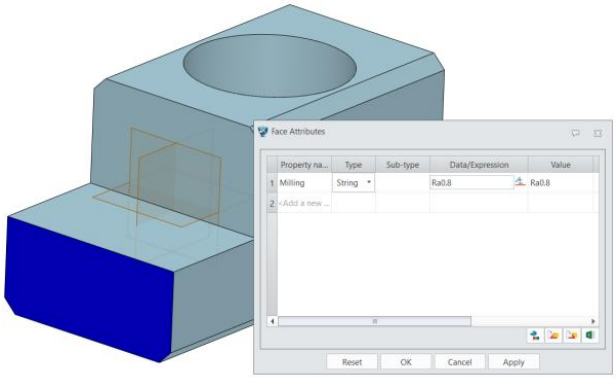



**【Where is it】**

Part/Assembly >> Attributes >> **Properties**

## 1.6.2 New Face Attributes Function

Adding attributes to faces is supported.




**【What users can do】**

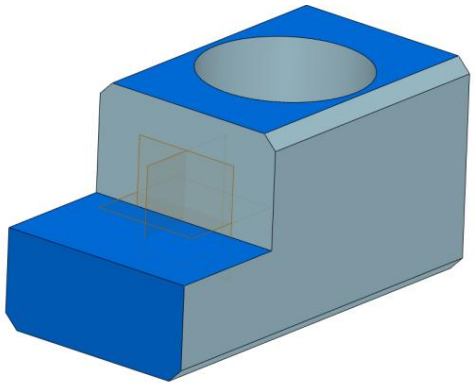
✓ Adding user attributes for selected face.

**【Notes】**

- Face attributes with the same name will share the same value.

**【Example】**

When developing customization function, developers can search for those faces with the same attributes, filter and batch assign colors to them.

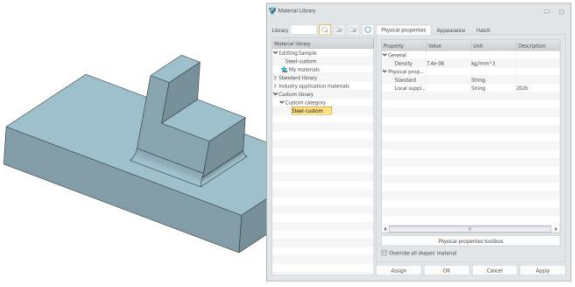
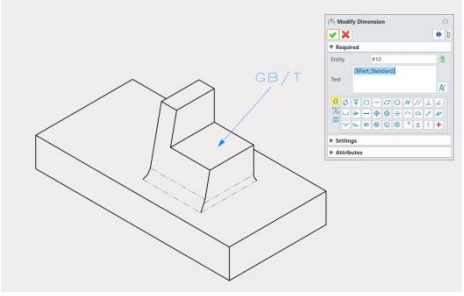


**【Where is it】**

Part/Assembly >> **Face Attributes**

## 1.6.3 String Physical Attributes for Material

The material supports the addition of string-type physical properties which can be invoked in environments such as the drawing sheet. The invocation method is [$Part_attribute_name], such as [$Part_Local_Supplier]=ZW3D.

| Modeling environment | 2D drawing |
| --- | --- |
|  |  |

**【What users can do】**

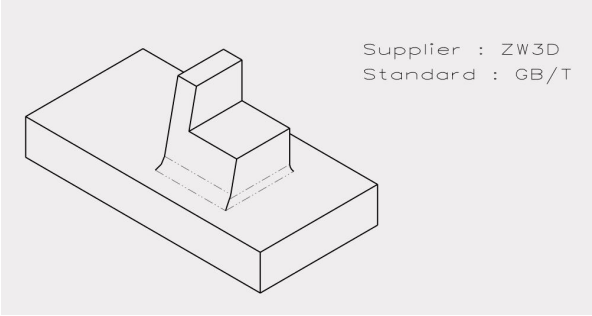
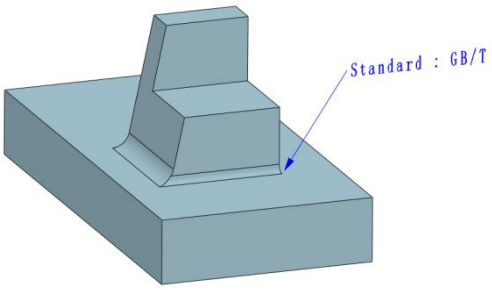
- ✓ Adding physical properties of any string type to the material and providing a description for it.
- ✓ Support physical attributes named in non-English characters.
- ✓ Support calling physical property value in user properties and variable managers.

**【Notes】**

- The name of the custom attribute shouldn't be the same as standard attributes.
- The space in the custom attribute name needs to be replaced with the underscore symbol "_". For example, if the attribute name is "Local supplier", the user needs to input [$Part_Local_Supplier] to refer to them.
- Custom physical attribute names need to follow the naming rules of the equation manager. Therefore, it doesn't support special characters such as "/" or " ".

**【Example】**

Attach custom physical attributes to the material of the current file, such as standards, material suppliers, etc. And refer to them in the notes of 2D drawing and PMI.

| Drawing Sheet | PMI |
| --- | --- |
|   |   |

**【Where is it】**

Part/Assembly >> Attributes >> **Material Library**

## 1.6.4 New Tree Properties

In assembly design, the number of components is an important reference indicator for the assembly.

Engineers use this quantity to make a rough assessment of the design results. The multi-column tree allows users to directly view the total number of components under the current assembly, helping engineers to evaluate the assembly design status in real time.

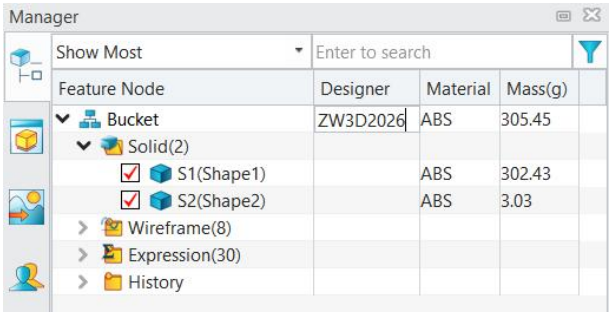
The assembly manager supports displaying more types of attributes in the part environment, including standard attributes, physical attributes, and user attributes. It also supports edited attributes by double-clicking them.

**【What users can do】**

✓ The assembly multi-column tree has added a component quantity property that can tally the total number of components included in the assembly.

✓ Quickly set materials for multiple shapes.

✓ In place editing of standard attributes or user attributes.




**【Notes】**

- Physical properties are calculated values, so they don’t support quick editing.
- Different types of attributes have different forms when edited in place, for example, when materials are edited, a material library window will pop up.

**【Where is it】**

Part/Assembly environment >> **Manager**

Assembly environment >> **Assembly Manager**

## 1.7 Customize Configuration Optimization

### 1.7.1 Quick Adjustment of Display Quality

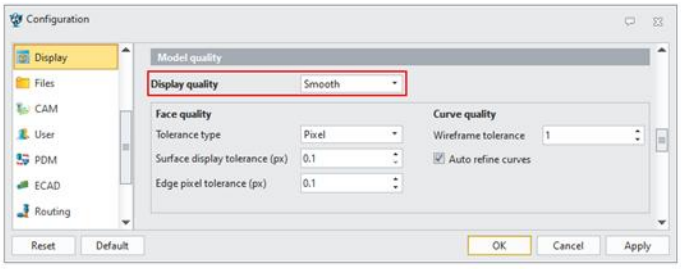
In the display configuration, the user can modify the mode or precision of the display quality to obtain different display effects.

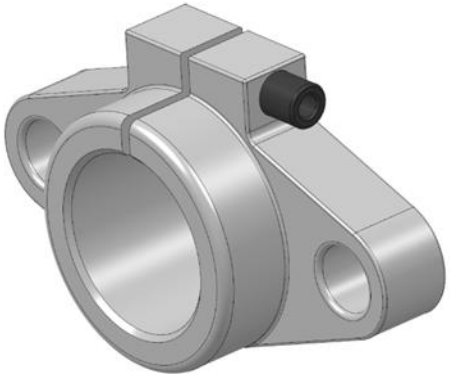
**【What users can do】**

- ✓ Quickly switch between different levels of display quality.
- ✓ User-defined display quality.

**【Example】**

After changing the display quality mode to smooth, click Apply or ok to get better display results.



**【Where is it】**

All Environment >> Configuration >> **Display**

### 1.7.2 Sketch Constraint Colors Supporting Global Configuration

In the color configuration, the user can modify the default constraint color of the sketch to keep the sketch constraint color display consistent for different files.

**【What users can do】**

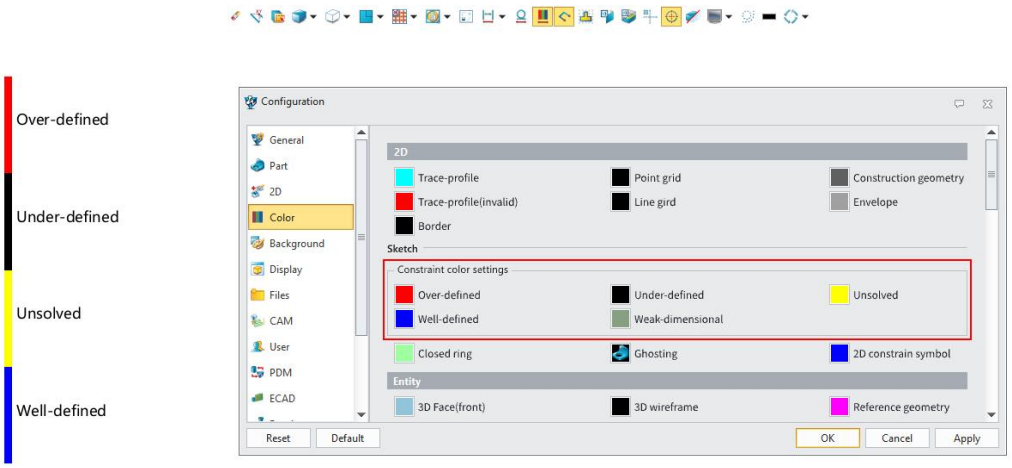
✓ Globally modify the constrained color of the sketch.

**【Notes】**

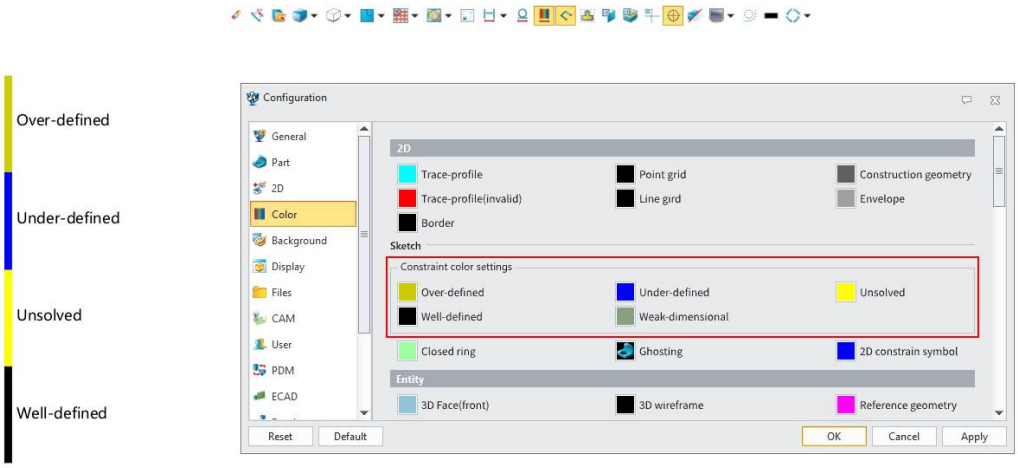
• The sketch constraint color of the old version file will be displayed according to the globally configured sketch constraint color when the file is opened in new version.

**【Example】**

1) In the software color configuration, modify the sketch constraint color and click Apply.




2) Constraint colors of the sketch environment are updated synchronously.




**【Where is it】**

All Environment >> Configuration >> **Color**

## 1.8 User Interface Optimization

### 1.8.1 ★New Micro-panel Interactions in Graphics Area

Users can quickly edit parameters through the micro-panel, such as modifying parameter objects, parameter types and parameter values.

**【What users can do】**

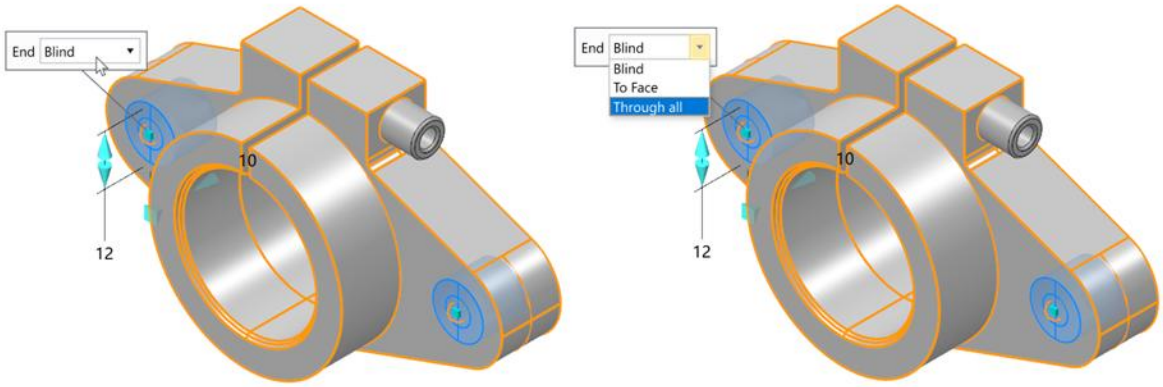
✓ In the graphics area, the micro-panel can directly activate parameter editing.

**【Notes】**

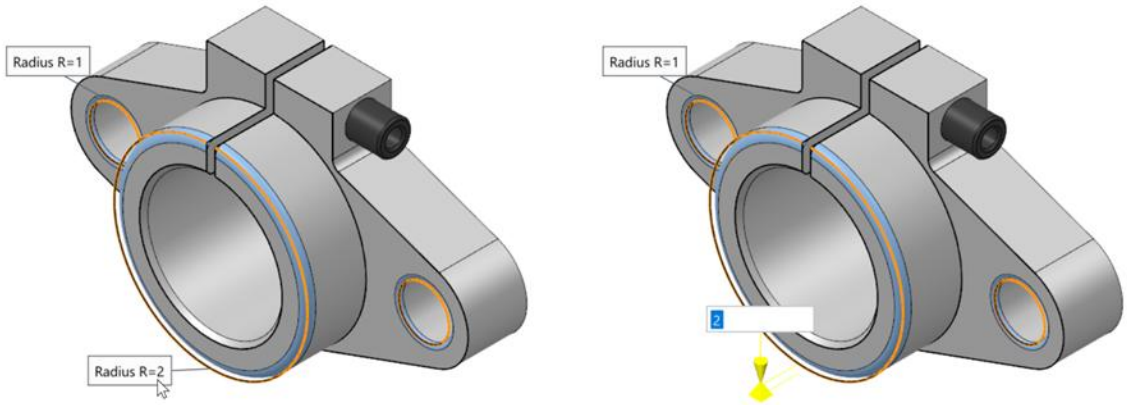
- The application range of the micro-panel is Fillet, Chamfer, Draft, Face Offset, Vol Offset, Shell, Thicken, Sweep, Swept Rod, Hole and Convert to Sheet metal.

**【Example】**

1) After activating the hole command and specifying the hole location, you can directly switch the end type.with the micro-panel.




2) After activating the Fillet command and specifying the radius parameter set,you can directly activate editing radius with the micro-panel.




**【Where is it】**

All Environment >> **Graphic Area**

## 1.8.2 Appearance Mapping Optimization

When users map the appearance of the model, they can quickly cut out the image by removing colors and apply multiple maps to one surface. In the view manager, you can quickly manage the layers and visibility of Maps.

**【What users can do】**

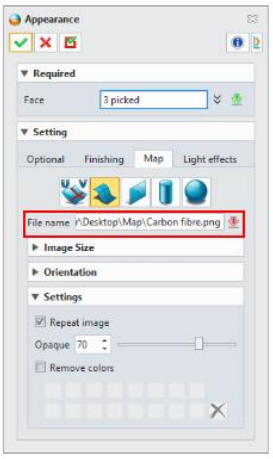
- ✓ Quickly remove unwanted color areas on the map.
- ✓ Multiple maps can be added to the same surface.
- ✓ Quickly manage the top-down and implicit relationships of maps.

**【Notes】**

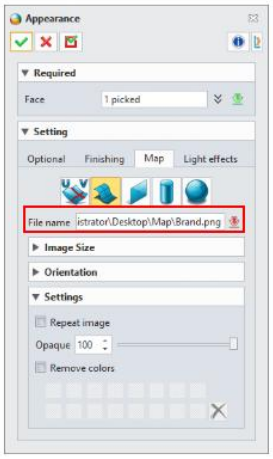
- After adding a texture map to a surface of the shape, a new face Texture node will be added.If a texture map is subsequently applied to the shape, the original face Texture Node will be overwritten.
- After adding a texture map to the shape, a new shape Texture node will be added.If a texture map is subsequently applied to a face of the shape, the Texture Node of the original shape will separate a face Texture node.

**【Example】**

1) First, add a texture map to the model.



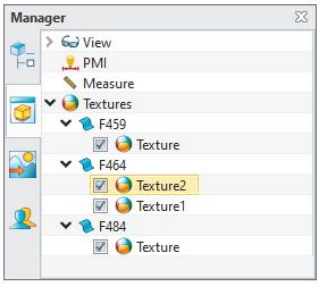
2) Then, add a brand map to the textured surface.

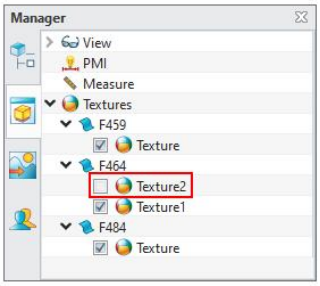


3) Start removing colors and double-click the swatch and click Custom.

4) Pick up the brand map to remove the area color and click Confirm to complete the cutout.

5) Manage all textures in Visual Manager.





**【Where is it】**

All Environment >> View Manager >> **Maps**

## 1.8.3 Feature Size Supporting Simple Edition

When users only need to simply edit the feature size, they can set the Dimension editing panel type to Simple, which can hide unnecessary advanced functions and make editing easier.

**【What users can do】**

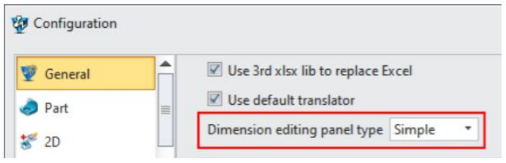
- ✓ The dimension editing panel provides two types: Simple and Advanced.
- ✓ Display of feature size can be cancelled by pressing Esc.

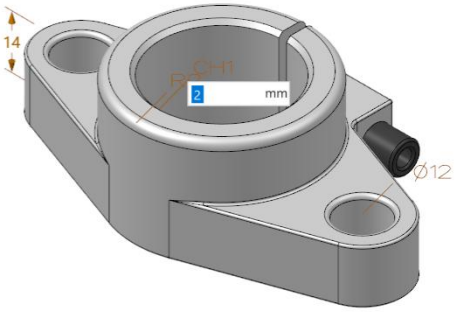
**【Notes】**

- When editing feature sizes in simple mode, clicking on the blank space in the graphics area will exit the editing state.

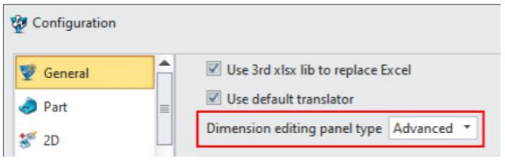
**【Example】**

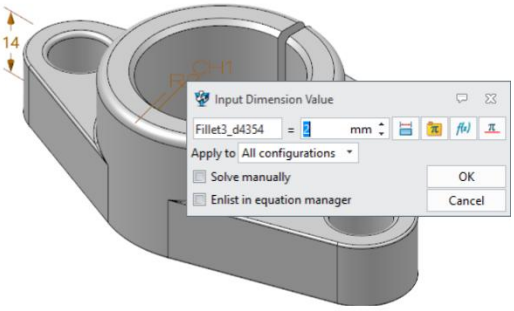
1) The Simple dimension editing panel supports quick dimension editing.




2) The Advanced dimension editing panel provides advanced editing function.




**【Where is it】**

All Environment >> Configuration >> **General**

## 1.8.4 Drawing Sheet Projection Supporting Quick Dragging

After selecting the drawing sheet projection, you can drag and drop at any position in the projection. After the drag is completed, the projection remains selected for continuous dragging.

**【What users can do】**

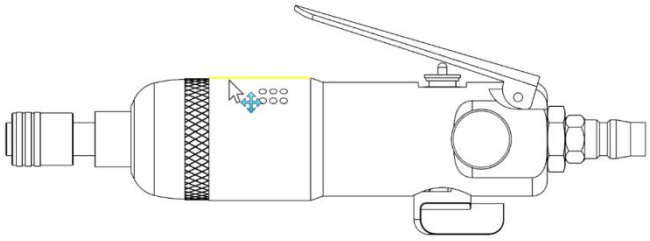
- ✓ It can be dragged anywhere in the highlighted drawing sheet projection.
- ✓ After dragging, the projection remains selected.

**【Notes】**

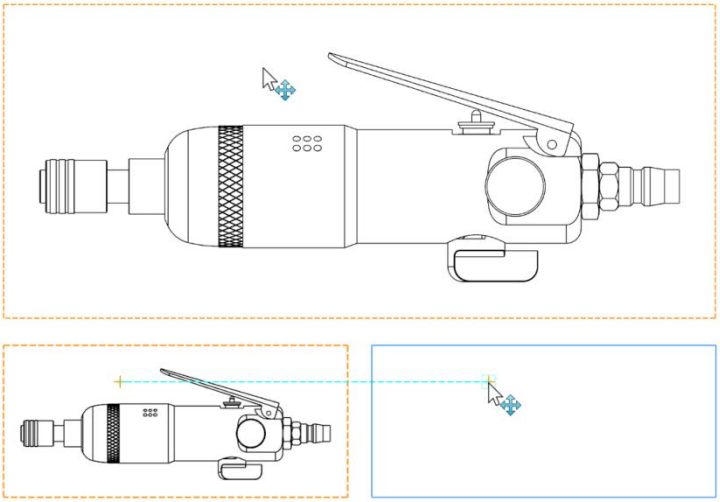
- The projection of the locked position cannot be dragged.

**【Example】**

1) Move the mouse to the object in the drawing sheet projection, and the cursor becomes a drag icon.



2) After selecting the projection, the cursor can be dragged anywhere in the projection.



**【Where is it】**

Drawing Sheet Environment >> **Graphic Area**

## 1.8.5 Quickly Locating Entities and Surface Objects

Clicking on a surface in the graphics area will highlight objects in the entity or surface folder, allowing users to quickly locate and perform other operations.

**【What users can do】**

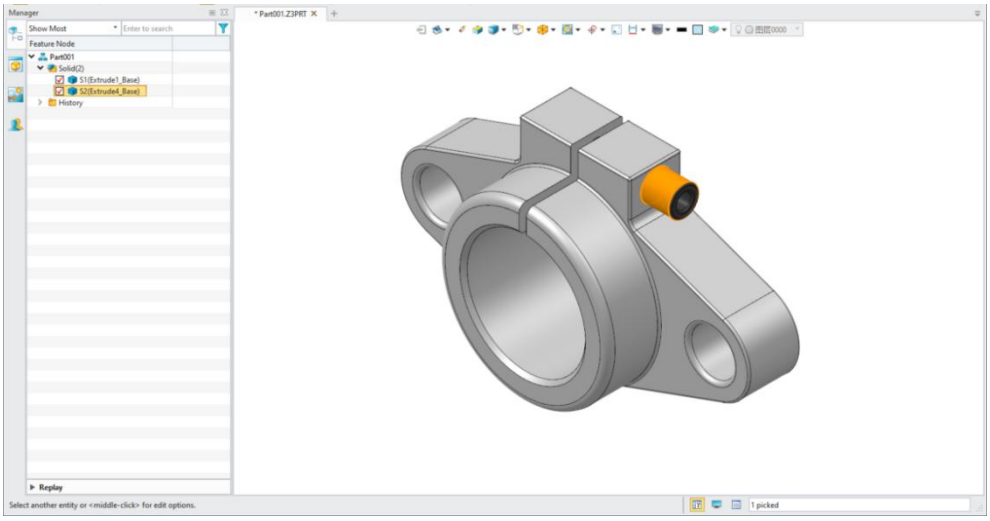
✓ In the default selection mode, clicking surfaces in the graphics area will highlight objects in the entity or surface folder.

**【Notes】**

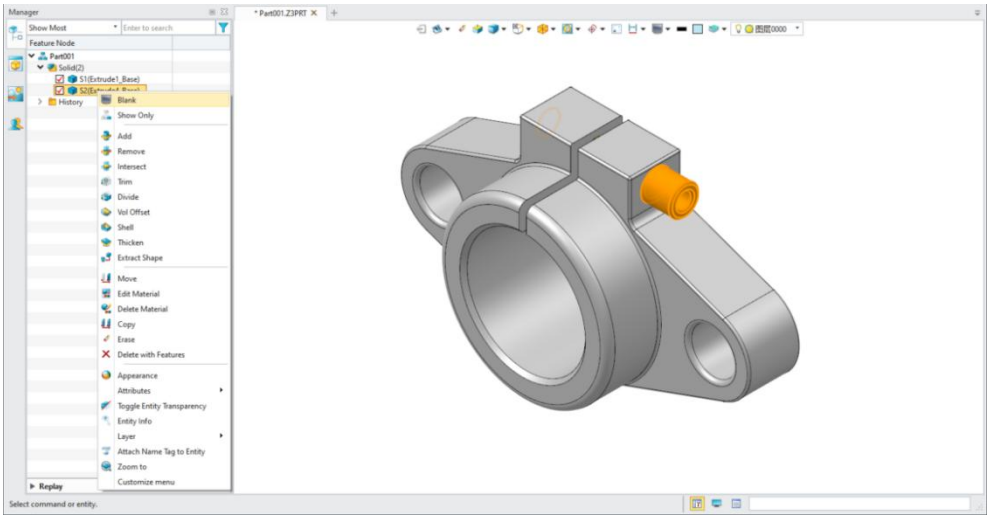
- In the default selection mode, boxing surfaces in the graphics area will not highlight objects in the entity or surface folder.
- When any command is started, clicking surfaces in the graphics area will not highlight objects in the entity or surface folder

**【Example】**

1) In the default selection mode, click the face of the model to highlight the corresponding entity object.



2) You can quickly right-click on highlighted entity objects.



【Where is it】

Part Environment >> **Graphic Area**

## 1.8.6 Quickly Hide Datum in 2D Sketch

In the 2D sketch environment, users can quickly control the visibility of the datum, datum axis and reference CSYS through the DA toolbar.

**【What users can do】**

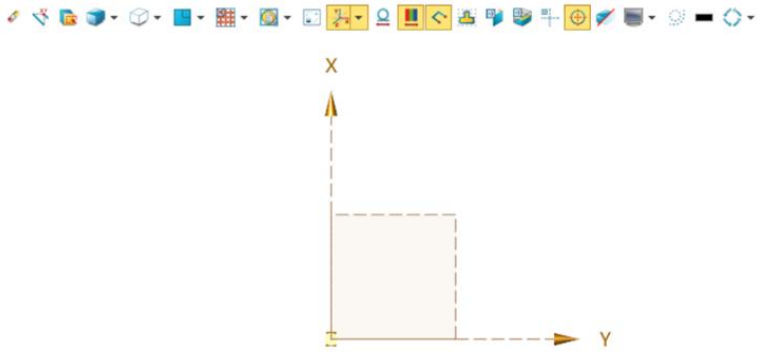
✓ DA toolbar can control the visibility of the datum in 2D sketch.

**【Notes】**

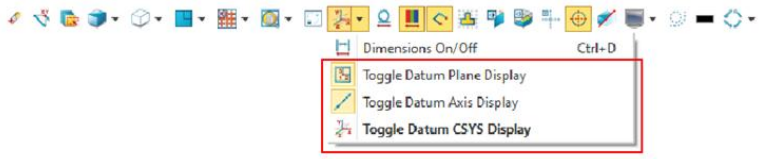
- Turn off the visibility of the datum in the DA toolbar, and the display function of the reference object will not take effect on the feature tree.

**【Example】**

1) Create a new 2D sketch without hiding the datum CSYS in the modeling environment.




2) Hide the datum CSYS in the DA toolbar.




【Where is it】

2D Sketch Environment **>> Document Aware Toolbars**

# 2 Translator

## 2.1.1 Input Format Upgrade（ZW3D2026 SP1）

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

| No. | Product | File Extension Name | Default Translator | Alternative Translator |
|---|---|---|---|---|
| 1 | Catia V4 | .model, .exp, .session | 4.1.9 - 4.2.4 | 4.1.9 - 4.2.4 |
| 2 | Catia V5 | .CATPart, .CATProduct, .CGR | V5R10 --- V5-6R2025 | V5R8---V5-6R2025 |
| 3 | Catia V5_2D | .CATDrawing | V5R10 --- V5-6R2025 | V5R8---V5-6R2025 |
| 4 | 3DExperience (CATIA V6) | .CATPart, .CATProduct | R2010x - R2025x | Exported from V6 R2025x and previous version |
| 5 | CGR | .cgr | Exported from V6 R2025x and previous version | Exported from V6 R2025x and previous version |
| 6 | NX(UG) | .prt | 11 - NX 2412 | 11 - NX 2412 |
| 7 | Creo(Pro/E) | .prt, .prt.*, .asm, .asm.* | 16 - Creo 12.0 | 16 - Creo 12.0 |
| 8 | Creo(Pro/E)_2D | .drw .drw.* | 2000i - Creo 11.0 | / |
| 9 | SolidWorks | .sldprt, .sldasm | 98 - 2025(only support 64-bit) | 98 - 2025(only support 64-bit) |
| 10 | SolidWorks_2D | .slddrw | 2004 - 2025 | 2013-2025(only support 64-bit) |
| 11 | SolidEdge | .par, .asm, .psm | 10 - 2025 | 18 - 2025 |
| 12 | Inventor | .ipt, .iam | 9 to 2026 | .ipt (6 - 2025)<br>.iam (11 - 2025) |
| 13 | ACIS | .sat, .sab, .asat, .asab | R1 - 2025 1.0 | R1 - 2025 1.0 |
| 14 | Rhino | .3dm | Version 2 - 8 | Version 2 - 8 |
| 15 | CADbro | .Z3D | | |

**【Where is it】**

Data Exchange >> Import

## 2.2 STEP Data Exchange Enhancement（ZW3D2026 SP1）

### 2.2.1 Assembly support exports only parts

The assemblies support exports only parts to STEP format. You can export all part files contained in the assembly with one click, quickly distributing the STEP files of different parts to the corresponding upstream departments.

**【What users can do】**

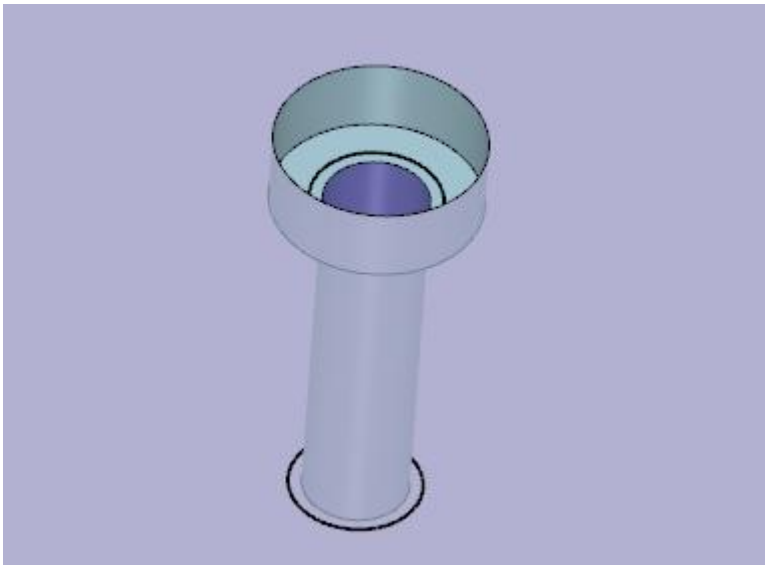
- ✓ Create a separate STEP file for each part in the assembly.

**【Where is it】**

Data Exchange >> Export >> STEP Format >> **STEP Settings**

### 2.2.2 Support for Exporting Threaded Holes Information

STEP export adds geometric data to express threaded holes. You can quickly identify threaded holes in STEP files. As shown in the figure below, the bottom hole at the start and end of the thread will be represented by wires.



**【What users can do】**

- ✓ Can measure the diameter of the wire to get the diameter of the threaded bottom hole.

✓ Can measure the distance between wires to get the thread depth.

**【Where is it】**

Data Exchange >> Export >> **STEP Format**

### 2.2.3 Support Reading More Product Information

STEP import supports reading product information such as part numbers, versions, definitions, terminology, sources, and descriptions. You can read the above product information in the user attributes.

**【What users can do】**

✓ After importing STEP, information such as Part Number, Revision, Definition, Nomenclature, Source, and Description can be read in the user properties.

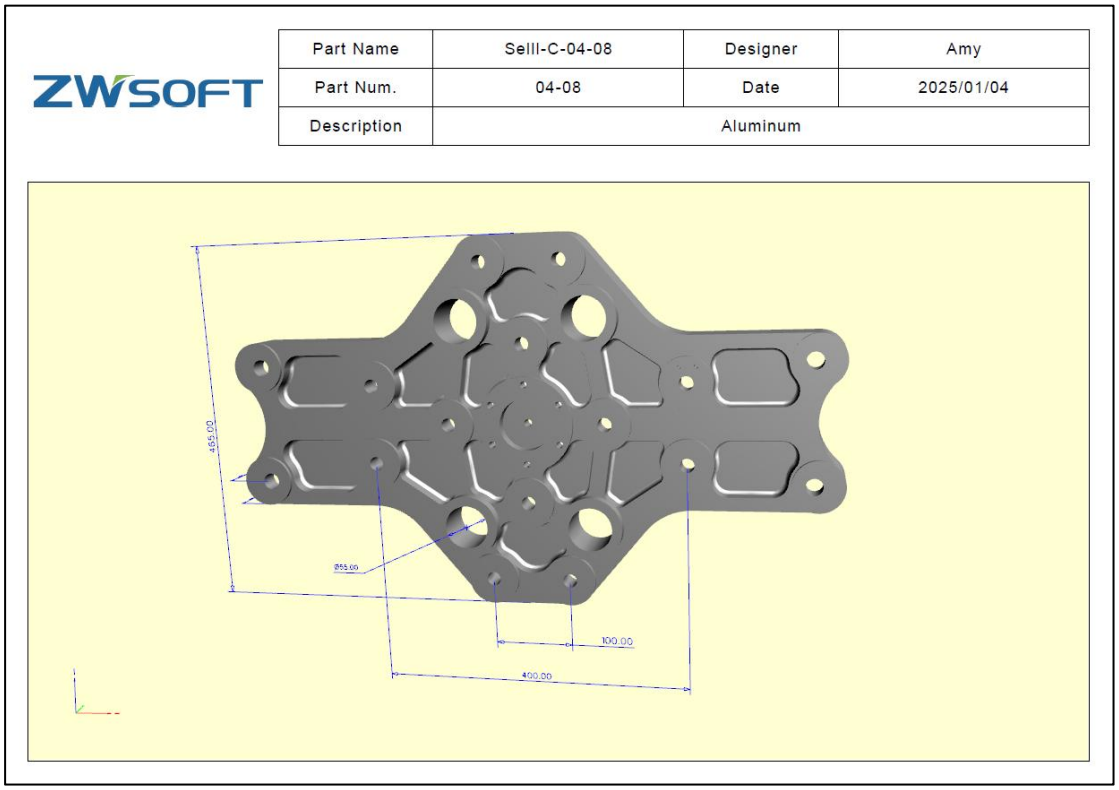
**【Where is it】**

Data Exchange >> Import >> **STEP Format**

## 2.3 PDF Export Enhancement

### 2.3.1 Exporting 3D PDF Supporting the Use of Template

The 3D PDF export supports template referencing and allows insertion of variables, annotations, tables, images, and other 2D elements. Users can leverage this feature to create interactive product manuals with embedded 3D models.




**【What users can do】**

- ✓ Use the drawing sheet function to create template files.
- ✓ Use template files when exporting 3D PDF.

**【Notes】**

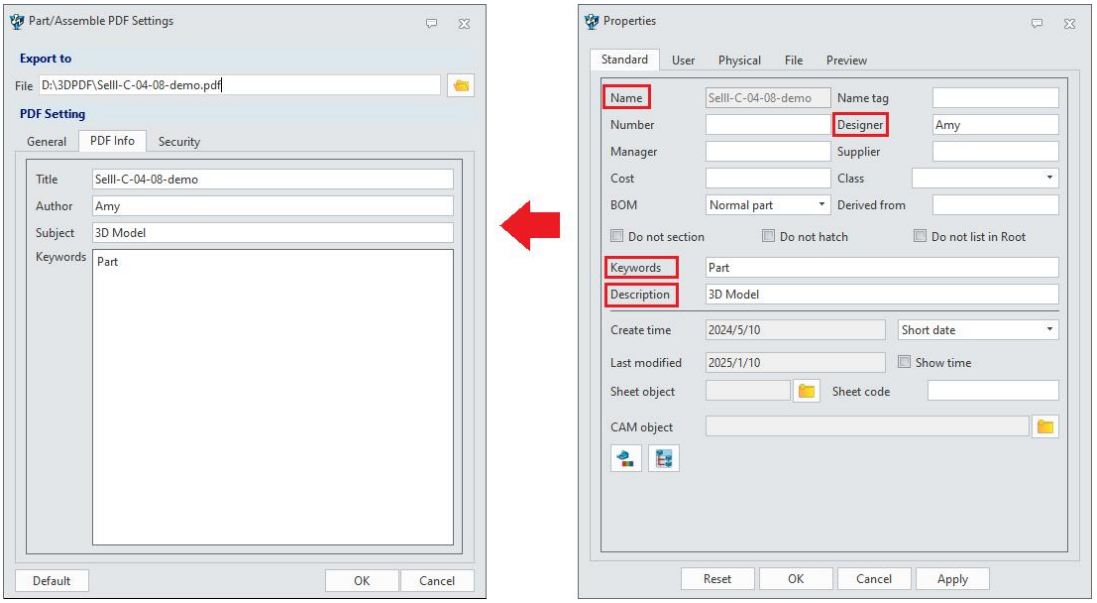
- Need to add View Layout Region in the template files to define the display range of the 3D model in 3DPDF.

**【Where is it】**

File >> Export >> PDF Format >> **PDF Settings**

## 2.3.2 Default Record PDF Info

PDF information can be exported based on the properties of original files.




**【What users can do】**

✓ Record PDF information like Name, Designer and so on.

【Where is it】

File >> Export >> PDF Format >> **PDF Settings**

### 2.3.3 PDF Exporting and Printing Supporting Bold Text

PDF export and print supports bold text to avoid unclear text when users scan paper drawings.

| Bold | Normal |
| --- | --- |
| NOTES:<br>1, FOR INSPECTION PURPOSES ONLY. | NOTES:<br>1, FOR INSPECTION PURPOSES ONLY. |

【What users can do】

- ✓ Users can choose between regular font weight or bold for exporting PDF and printing text.

【Where is it】

File >> Export >> PDF Format >> **PDF Settings**

File >> Print/Plot >> **Print Setup**

### 2.3.4 Hyperlink Text Supports Export PDF

When exporting drawings sheet to PDF, the software supports active hyperlinks that users can directly open within the PDF document.

【Where is it】

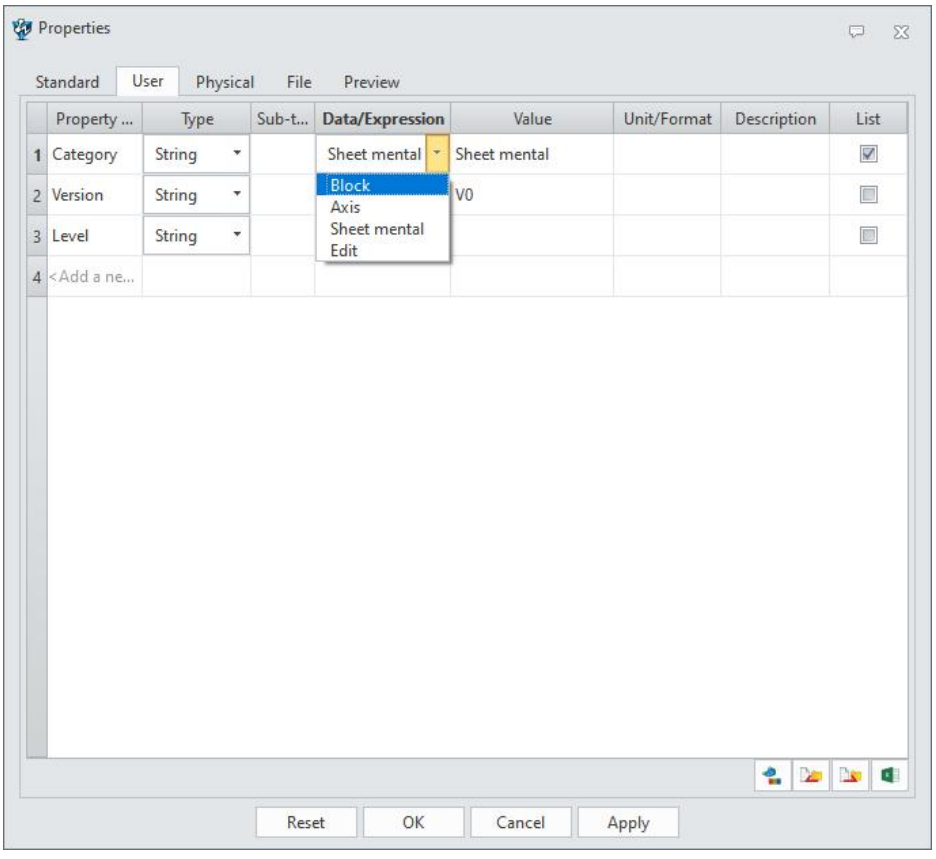
File >> Export >> **PDF Format**

## 2.4 3D Model Export Function Optimization

### 2.4.1 Inheriting User Properties by Template

After setting the template and importing the 3D model, the created file will inherit the user attributes of the original file based on the template's user attributes. The user attributes list will record the values of the template and the source file as options, and user attributes of the original file as the current value.

The following figure shows the input user attributes. The "Category" user attribute is in the list, where "Block" and "Axis" are template options, and "Sheet Metal" is the value of the original file. Users can flexibly choose the values they want.




**【What users can do】**

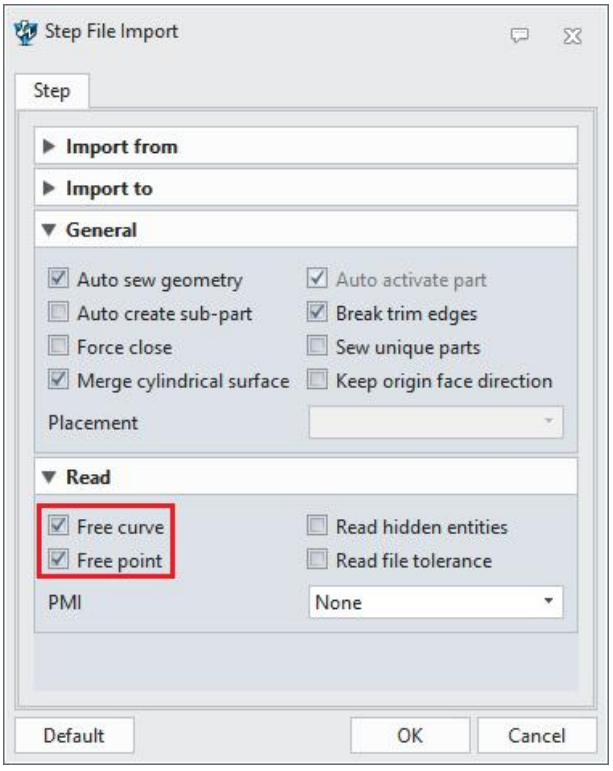
- ✓ Simultaneously retain the user attributes of both the template and the source file.

**【Where is it】**

File >> Import >> **3D Files**

## 2.4.2 Supporting Filtering STEP Free Curves and Points

Importing STEP files supports selectively importing free curves and free points.




**【What users can do】**

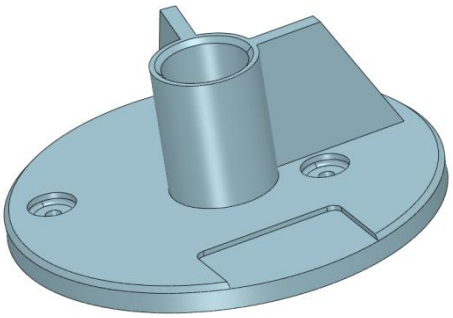
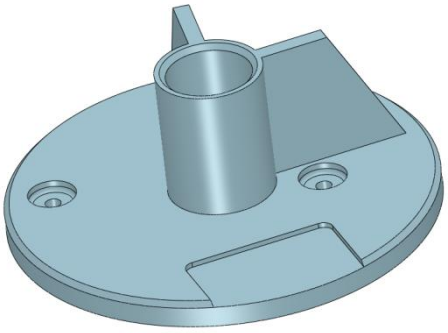
✓ Users can import free curves and points as needed.

**【Where is it】**

File >> Import >> **STEP Files**

## 2.4.3 Importing STEP Supporting Merging Torus and Conical Surfaces

Importing STEP files supports merging torus and conical surfaces, making it more convenient for users to perform modeling operations on the above periodic surfaces and reducing the selection of edges and surfaces.

| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|  |  |

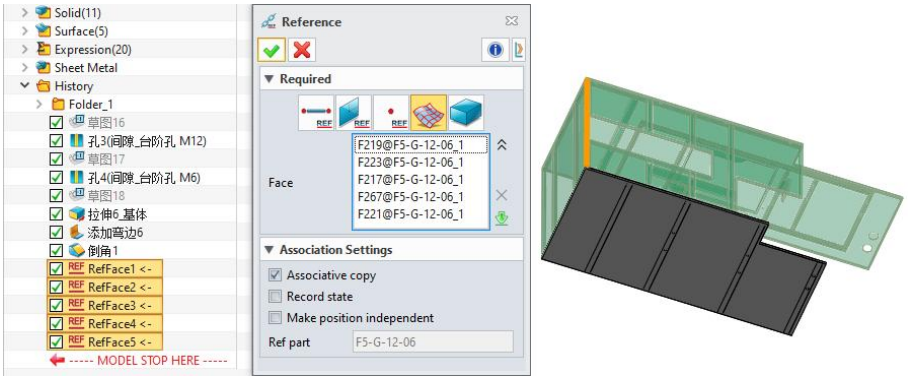
**【Where is it】**

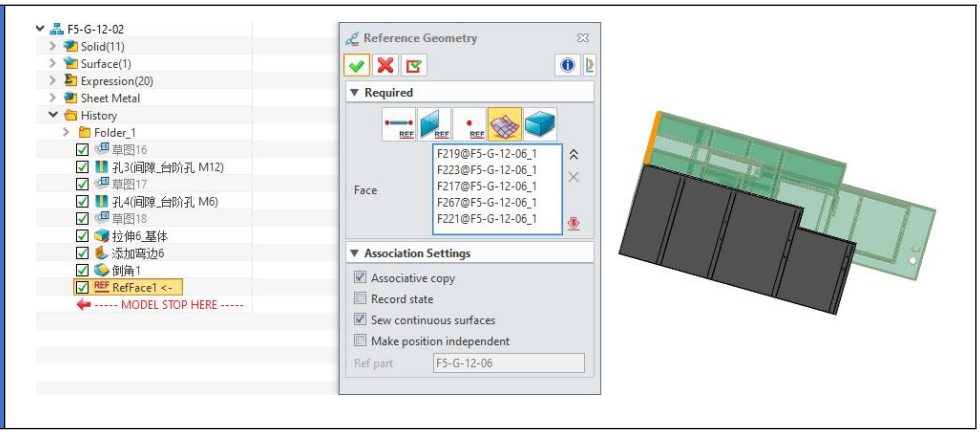
File >> Import >> **STEP Files**

## 2.5 Multi-Surface Reference

The "Reference" command has been optimized. When selecting multiple surface references, only one reference feature will be generated if the surfaces are continuous. This not only improves the design efficiency for users but also, to the greatest extent, reduces the number of model history features and the complexity of the model design.

**ZW3D 2025**



| ZW3D 2026 |  |
|---|---|

**【What users can do】**

✓ Record only one reference feature with multiple continuous surface objects as reference at once.

**【Notes】**

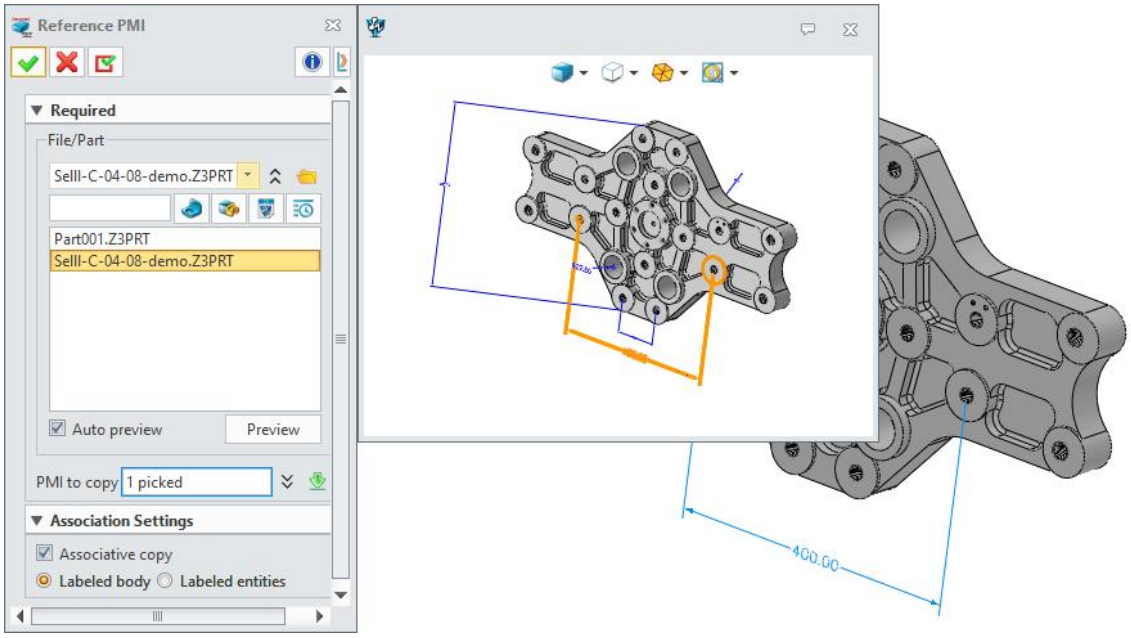
- If the selected surface is discontinuous, the software will create multiple reference features to complete the operation.

**【Where is it】**

Part/Assembly >> Data Exchange >> Import >> **Reference**

## 2.6 PMI Reference

In product model design, we usually use the already designed model for reference design. With the new reference PMI function, users can directly refer to the PMI annotation in other files for design. Design efficiency can be further improved.




**【What users can do】**

✓ Copy the PMI of the external file and the associated annotation objects.

**【Example】**

In the process of product design, it is necessary to simultaneously create both the finished model and the stock model. After completing the creation of the finished model and annotating it with PMI, the stock model can be created directly by referencing the PMI.

**【Where is it】**

Part/Assembly >> Data Exchange >> Import >> **Reference PMI**

## 2.7 Supporting ProE Format Connectors Importing

The conversion of connectors in ProE format is supported, which means parts or assemblies can be directly converted into connectors ports and terminals in ZW3D format, eliminating the need for users to redefine them, thereby improving modeling efficiency.

**【What users can do】**

✓ It is up to the user to decide whether to convert the part/assembly file to ZW3D format "connector";

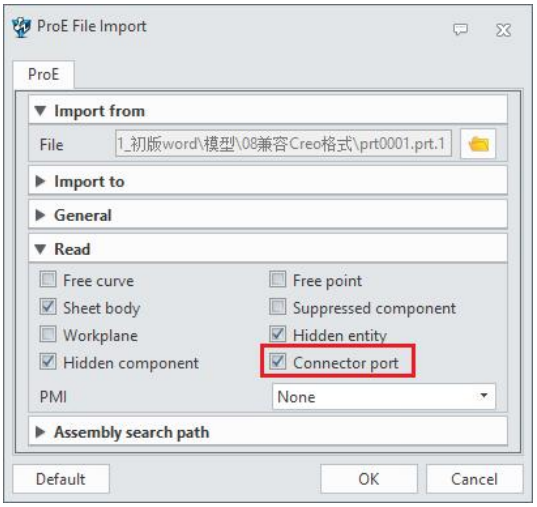
✓ You can identify the basic coordinate system of the Creo model as ports and other created coordinate systems as terminals by yourself.

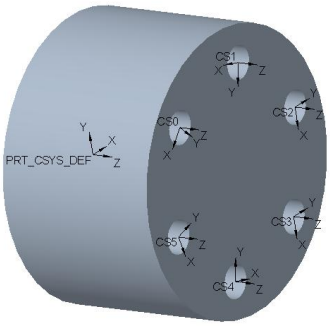
✓ The Z-axis direction is recognized as the terminal and port direction, and the XY plane is recognized as the end position of the port. The default extension value is 0.

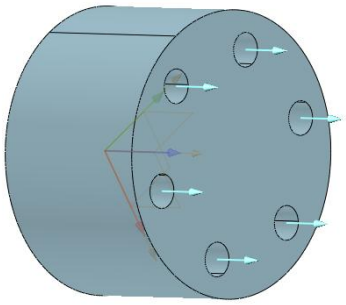
✓ The terminal name adopts the coordinate system name of the recognized terminal.

**【Example】**

Use the "Import" tool to select the ProE file that needs to be converted, check the "Connector Port" option, and the coordinate system of the ProE format file will be automatically converted to the ports and terminals of the ZW3D format file.



**【Where is it】**

ZW3D main page >> Data Exchange >> **Input**

# 3 CAD

## 3.1 Sketch & Wireframe Design

### 3.1.1 Constraint Optimizations

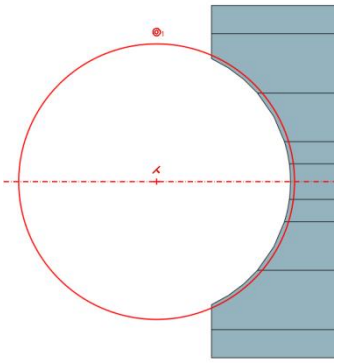
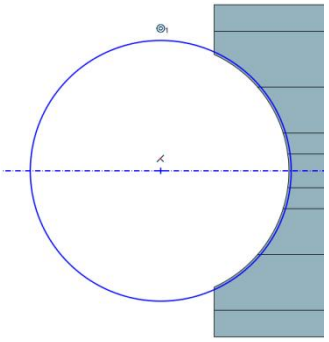
The sketch constraint solving ability has been enhanced. When users draw sketches by referring to the complex external surface contours, and constraint conflicts will no longer occur.

**【What users can do】**

✓ In the sketch environment, when drawing or offsetting with reference to external geometry, no constraint conflicts will occur.

**【Example】**

1) Adding a concentric constraint with external geometry will not result in a constraint conflict.

| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|  |  |

**【Where is it】**

Sketch>>**Constraints**

### 3.1.2 Dimension Optimizations

Optimize the selection logic for dimension objects and enhance annotation display quality to improve sketch dimensioning efficiency.

**【What users can do】**

✓ Dimensions will not directly select the midpoint now, users can activate the midpoint selection by hovering the mouse over the target object.

✓ The style of diameter and radius dimension is more concise.

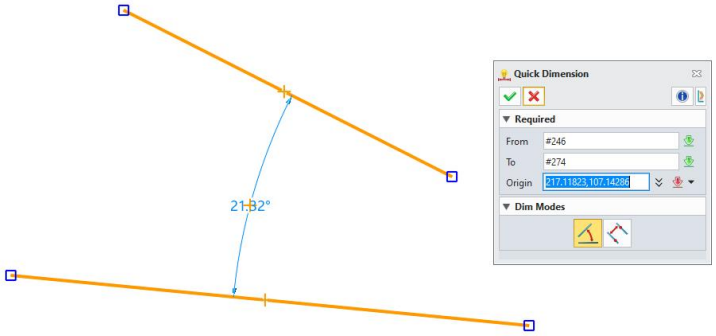
✓ The dimensioning command allows you to select completely external geometric objects to create dimensions.

**【Notes】**

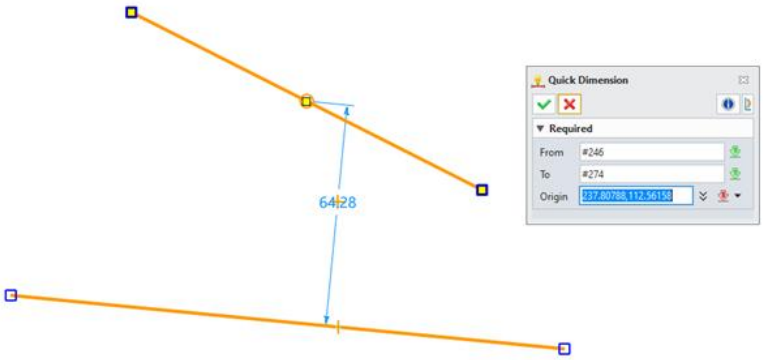
- Dimensions created from completely external geometric objects are non-editable.

**【Example】**

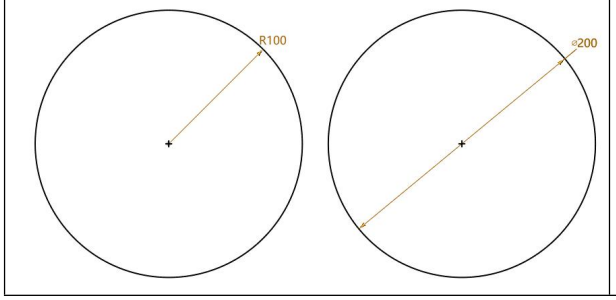
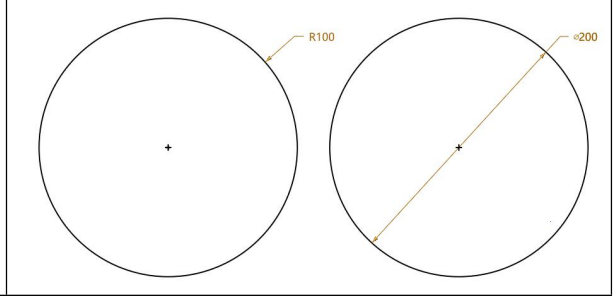
1) When clicking on the midpoint of a line in the dimension command, the line entity is selected.




2) In the dimension command, when the mouse hovers over the line for a period of time, the feature point will be highlighted, and the midpoint can be selected at this time.

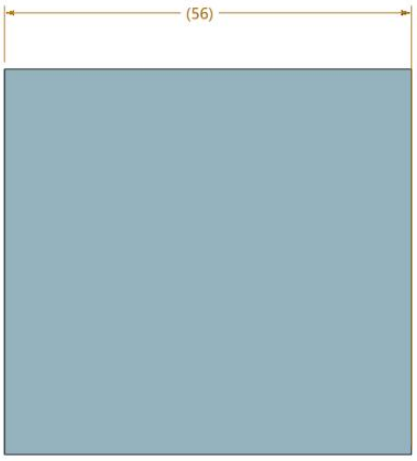


3) Create radius or diameter dimensions.

| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|  |  |



4) Use the dimension command to select fully external geometry to create dimensions.




**【Where is it】**

Sketch>>**Dimension**

Configuration>>Part>>General>>**Feature point highlight timer**

## 3.1.3 Reference Optimizations

When editing the sketch of parts during assembly, users can only refer to the position of external components without creating a reference relationship.

**【What users can do】**

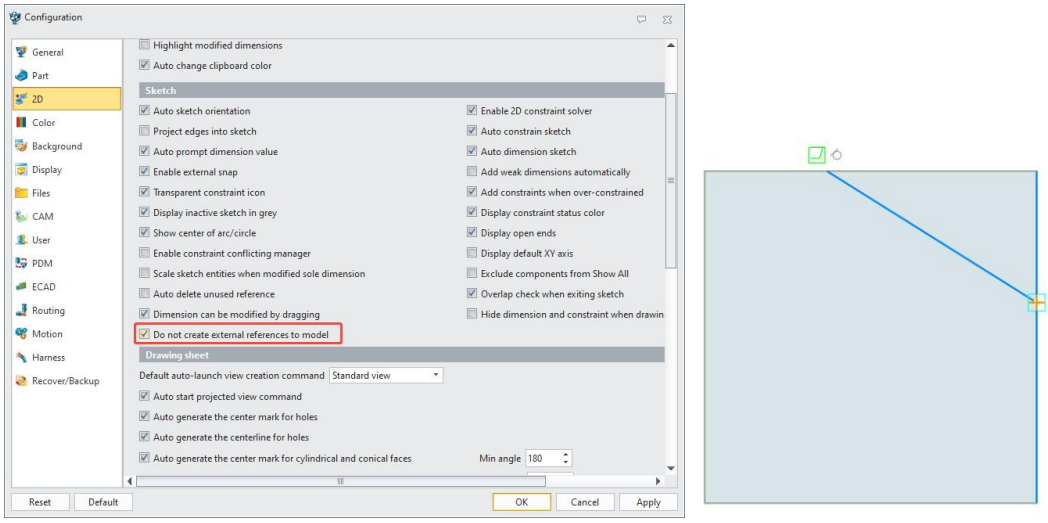
✓ Editing the sketch of parts during assembly does not create reference relationships by referencing the geometry of other components.

**【Notes】**

- Referring to the internal objects or sub level objects of the current editing model will still create reference relationships.

**【Example】**

Enable option: "Do not create external references to model." When the user edits a sketch of one of the parts during assembly and draws with reference to other parts in the assembly, the geometric position of other components can be captured, but reference relationships will not be created.




**【Where is it】**

Configuration>> 2D>>Sketch>>**Do not create external references to model**

## 3.1.4 Supporting Editing Parameters of Interpolation Points of Spline Curves

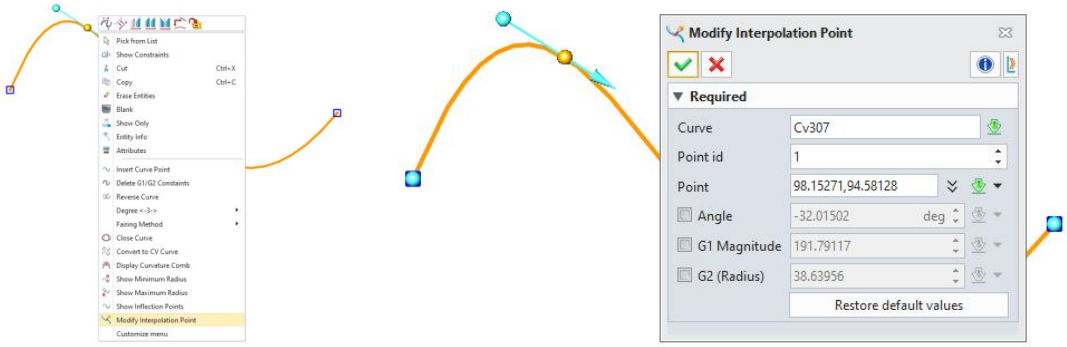
The sketch supports parametric editing of the interpolation points of spline curves.

**【What users can do】**

✓ The parameters of each interpolation point of the spline curve can be edited, enabling more precise definition of the spline curve.

**【Example】**

Activate the Modify Interpolation Point command from the right-click menu of the spline curve to edit the various parameters of the interpolation point in detail.




**【Where is it】**

Sketch>>Spline Curve>>Right Click Menu>>**Modify Interpolation Point**

## 3.1.5 Added with the Function of Deleting All Overlapping Objects

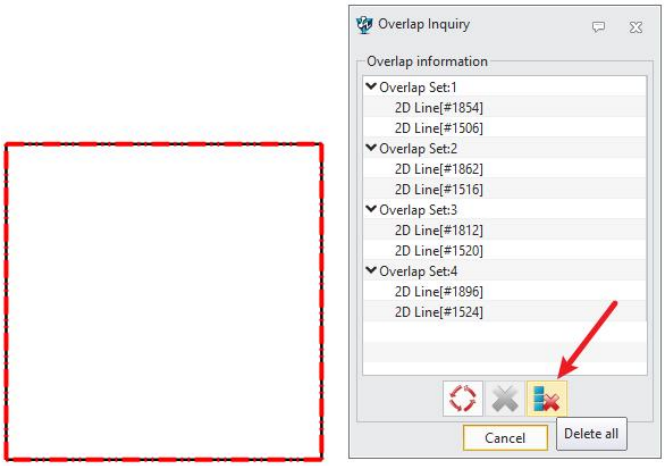
A "Delete All" function has been added to the sketch. This allows users to efficiently delete all overlapping objects in the sketch with just one click.

**【What users can do】**

✓ When there is an overlapping situation, all overlapping elements can be deleted quickly.

**【Example】**

When overlaps occur in the sketch, using the "Delete All" function can remove all overlaps.



【Notes】

- When using the "Delete All" function, objects with more associated constraints and annotations will be preferentially retained.

【Where is it】

Sketch >>Settings>>Overlap>>**Delete all**

### 3.1.6 Supporting Finding Intersection Points of Multiple Curves and Surfaces

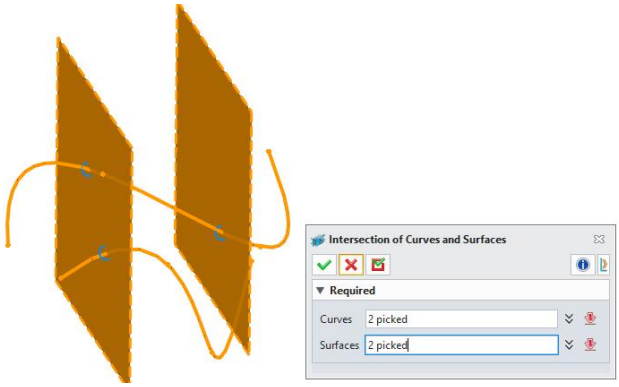
In the wireframe design scenario, finding intersection points between multiple curves and multiple surfaces is supported.

【What users can do】

✓ Supports calculating intersection points between multiple curves and multiple surfaces.

【Example】

Input two curved objects and one surface object to find the intersection points.




【Notes】

- When changes occur to the input curve and surface objects, the naming of the generated solid points will change.

【Where is it】

Part/Assembly >>Wireframe>>**Intersection of Curves and Surfaces**

### 3.1.7 Supports Adjusting the Placement Angle of Wireframe Text

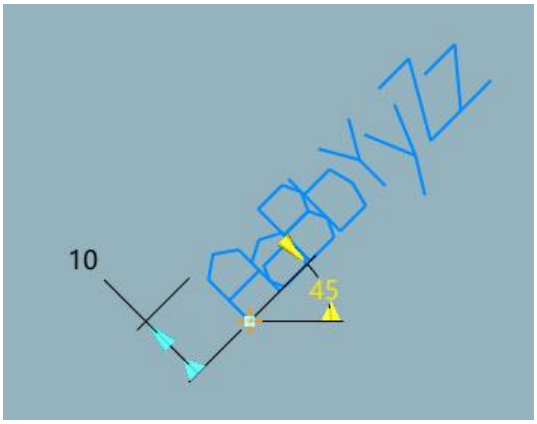
In wireframe design scenarios, it is possible to adjust the placement angle of wireframe text.

**【What users can do】**

✓ When placing wireframe text on a surface, you can adjust the rotation angle of the wireframe text.

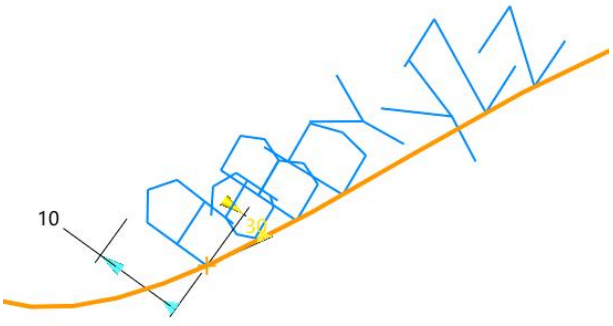
**【Example】**

When placing wireframe text on a curved surface, you can adjust the rotation angle of the wireframe text using the mouse.




**【Notes】**

• When the wireframe text is placed in an associated manner on a curve, each character rotates by the individually input angle.



【Where is it】

Part/Assembly >>Wireframe>>Curve>>**Wire Text**

## 3.1.8 Supporting Combined Projection of Multiple Curves

In the wireframe design scenario, it is supported to input multiple curves for combined projection

**【What users can do】**

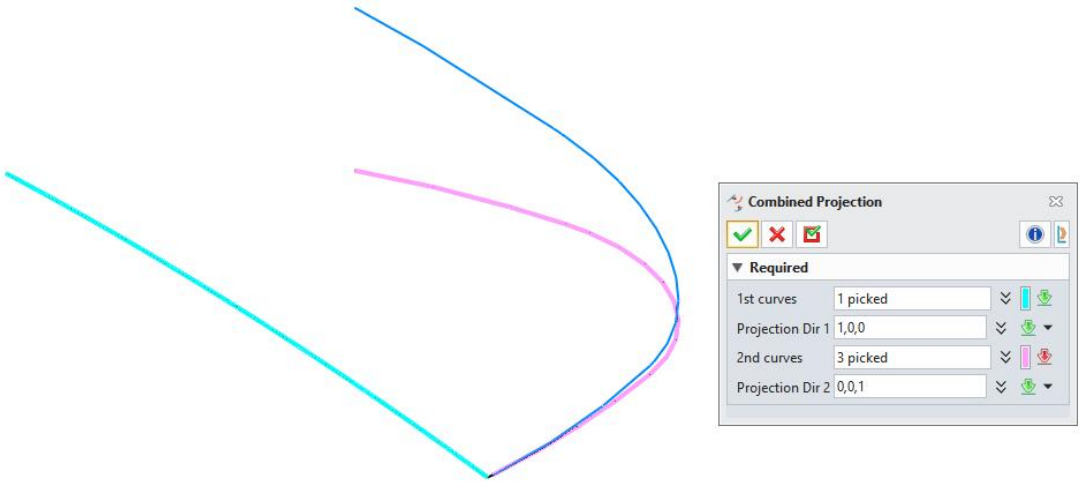
✓ Multiple continuous curves can be input for combined projection.

**【Notes】**

• When the input multiple curves are discontinuous, the curve of the combined projection result is also discontinuous.

**【Example】**

Input multiple continuous curves for combined projection.




【Where is it】

Part/Assembly >>Wireframe>>Curve>>**Combined Projection**

## 3.1.9 Optimization of Trimming Function（ZW3D 2026 SP1）

While trimming geometries generated by Reference function, the constraints related to other

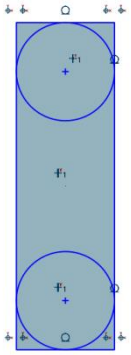
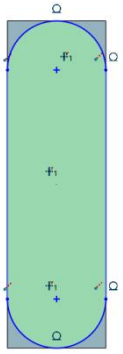
geometries will no longer be trimmed. That is, the original constraint status will be maintained before and after sketch trimming.

**【What users can do】**

✓ Trimming geometries generated by Reference function will not alter the constraint status of the sketch.

**【Example】**

Trimming geometry generated by Reference function will no longer cause the sketch to become Less-Constrained in the following figure.

| Sketch before trimming | Sketch after trimming<br>ZW3D 2026 | Sketch after trimming<br>ZW3D 2026 SP1 |
|---|---|---|
|  | |  |

**【Where is it】**

Sketch >> Edit Curve >> **Trim**

## 3.2 Shape Design

### 3.2.1 Extrude Optimization

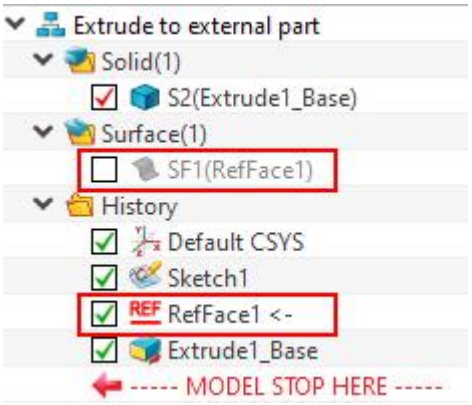
You can now select an external point or face from external part located in the same assembly for To Point/ To Faces/To Extended Face options.

**【What users can do】**

- ✓ Select external parts' feature as reference.
- ✓ Keep the association with the reference part.

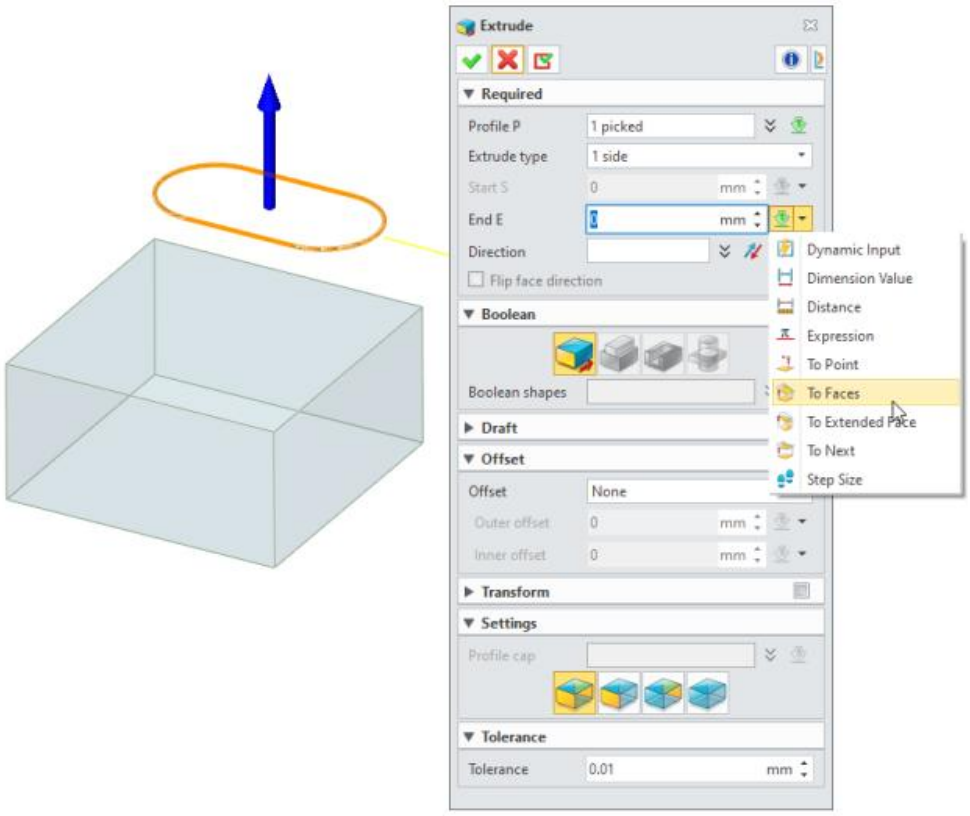
**【Notes】**

- A reference face will be auto created in the current part and kept hidden, do not delete it.

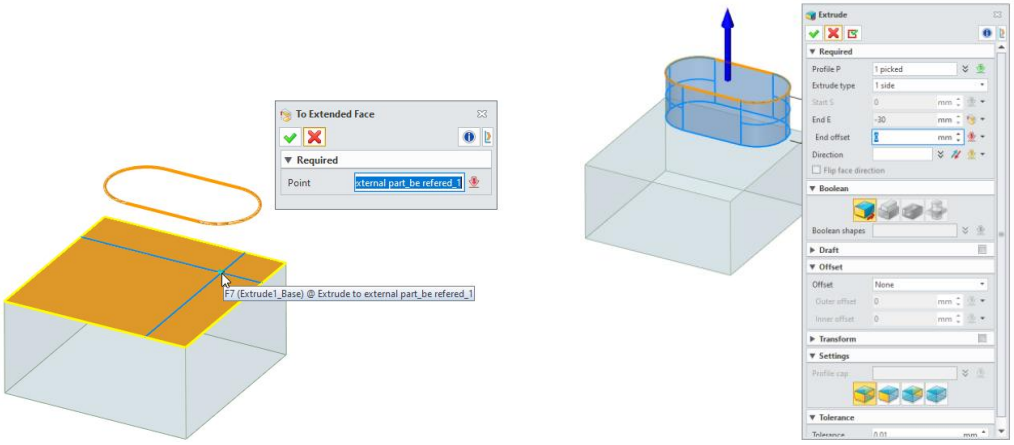



**【Example】**

1) Active one part of an assembly which contains 2 or more parts, click Extrude and select To Extended Face (or To Point/ To Faces) for End E.




2) Select a face on the external part, and the end of the extrude now coincides with the reference face.




## 【Where is it】

Part >> Shape >> Basic Shape >> **Extrude**

## 3.2.2 ★Fillet Optimized

### 3.2.2.1 New “Offset Surface” Method in Elliptical Fillet

The fillet command for ellipses has been enhanced with the addition of an "Offset Surface" type, which also supports maintaining the fillet to the edge.

Additionally, the "Offset Surface" type for asymmetric chamfers now also supports maintaining the chamfer to the edge.

**【What users can do】**

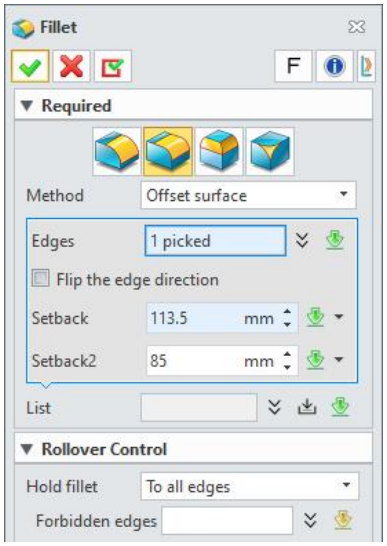
- ✓ In “Offset surface” method, holding to edges can be realized on one side.
- ✓ In “Offset surface” method, the support geometry can transfer from face to edge and then to face.

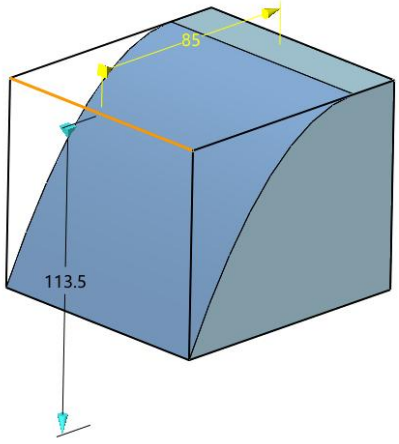
**【Notes】**

- For the side of setback 2, it cannot be held to edges.
- As the setback on one side increases, it will succeed in holding the fillet, reach a critical state, and then fail to hold the fillet.

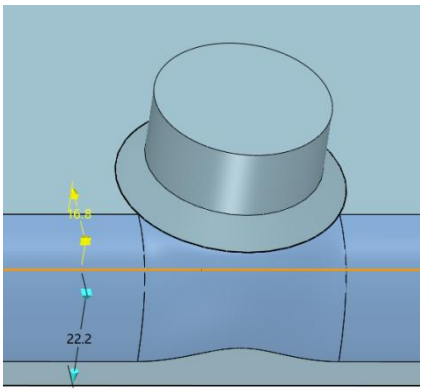
**【Example】**

1) Holding to edges can be realized on one side.



2) The support geometry can transfer from face to edge and then to face.




**【Where is it】**

Part/Assembly Environment >> Shape >> **Fillet/Chamfer**

### 3.2.2.2 Strengthening Hold Fillet to Edges

A new drop-down box replaces the original checkbox, which also allows users to specify edges as forced edges or forbidden edges in hold fillet control. When users check the box of "Show automatic hold fillet/chamfer" in the part configuration, there's a new "Automatic" option in hold fillet control, which can initially determine the prior result by different scenarios.

**【What users can do】**

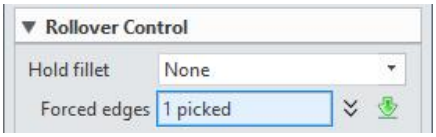
- ✓ It allows users to specify edges as forced edges or forbidden edges in hold fillet control.
- ✓ For "Automatic" mode, it can initially determine the prior result by different scenarios.

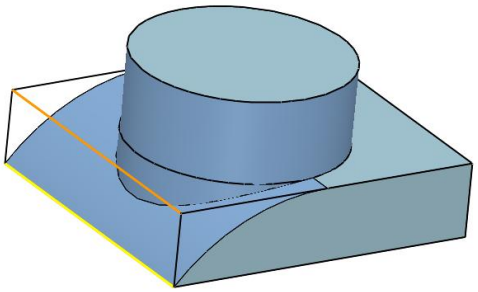
**【Notes】**

- "Automatic" sometimes may not provide the best result as users anticipate.

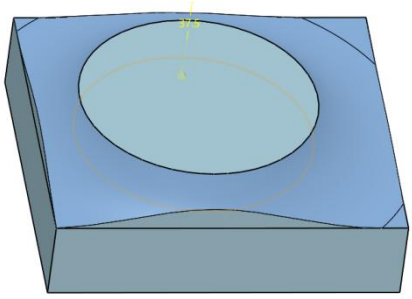
**【Example】**

1) Users specify edges as forced edges or forbidden edges in hold fillet control。



2) For “Automatic” mode, it can initially provide the prior result.



**【Where is it】**

Part/Assembly Environment >> Shape >> **Fillet/Chamfer**

### 3.2.2.3 New Overlap Method when Fillets of Different Convexities with Interference

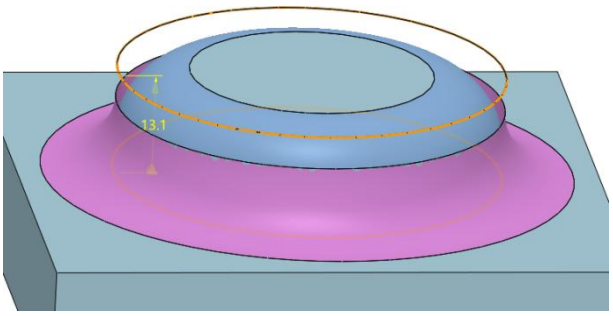
When the user inputs two fillet chains with different concavity-convexity and there is an overlapping area between them, the overlay order of the overlapping part is determined by recognizing the sequence of user input. The edge input by the user first is selected as the priority edge for fillets in case of interference, and the edge input later will default to overlay the previous fillet chain.

**【What users can do】**

✓ The overlap sequence can be switched by changing the input sequence of two fillets with different convexities

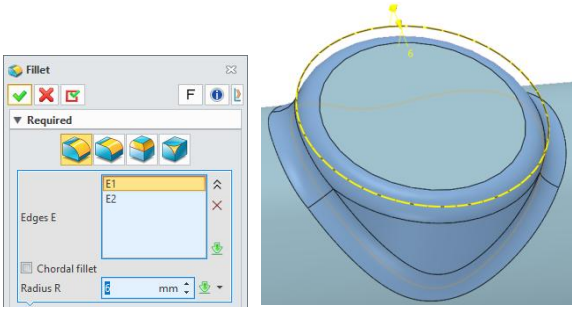
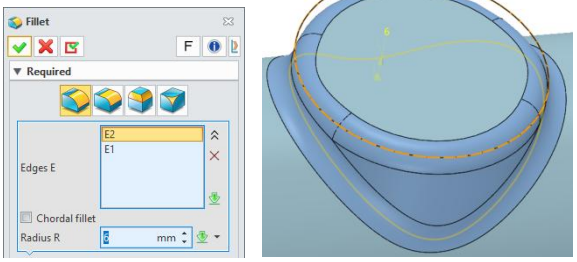
**【Notes】**

- Fillet chains with different concave and convex corners have to intersect, otherwise the fillet would be failed.




**【Example】**

1) The latter fillet will overlap the former one.

| Pick convex edge first | Pick concave edge first |
|---|---|
|  |  |



**【Where is it】**

Part/Assembly Environment >> Shape >> **Fillet/Chamfer**

## 3.2.2.4 New Conic Ratio in G2 Fillet Type

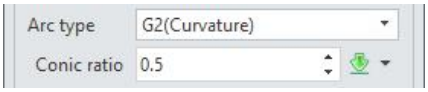
The new conic ratio replaces the original weight ratio, which allows users to input 0.05-0.95. The default ratio is 0.5.

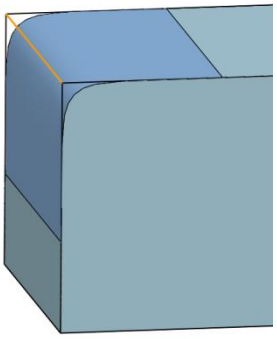
**【What users can do】**

✓ Currently, new G2 can avoid some extreme and irrational results in a more straight and convenient method.

**【Example】**

The fillet shape can be adjusted by conic ratio.



**【Where is it】**

Part/Assembly Environment >> Shape >> **Fillet/Chamfer**

### 3.2.2.5 New “Trace Tangent Fillet to End”

The fillet command has a new option called "Track Tangent Fillet to End". This option will determine whether the areas on both sides of the selected edge chain ends meet the tangent continuity requirements. If the continuity requirements are met, the fillet surface will track the tangent chain, and this process will overwrite the old fillet surface with a smaller radius.

**【What users can do】**

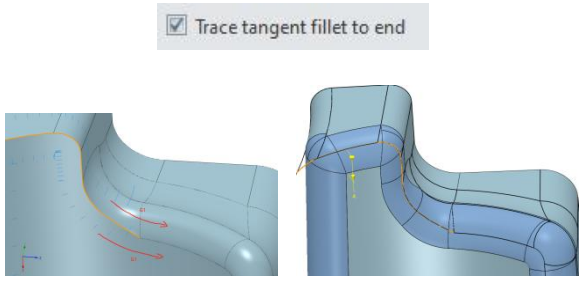
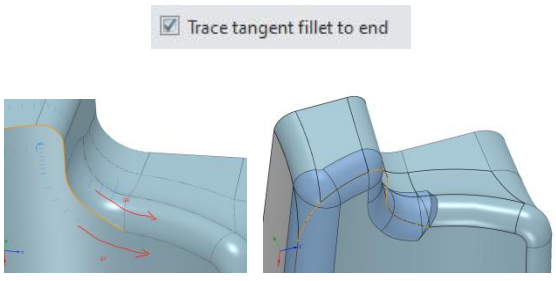
- ✓ Tracing to end can cover old fillets if it satisfies the requirement of continuity.

**【Notes】**

- Sometimes, the result of tracing tangent fillet may be different from that of picking tangent chain at the end of chain.
- It doesn’t exist in offset distance type.

**【Example】**

If the adjacent surfaces can reach G1 continuity at the end of edge chain, it will succeed in covering old fillets.

| Satisfy G1 continuity | Not satisfy G1 continuity |
| --- | --- |
|   |   |

**【Where is it】**

Part/Assembly Environment >> Shape >> **Fillet/Chamfer**

### 3.2.2.6 New Corner Type

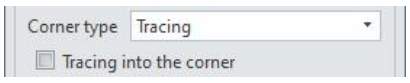
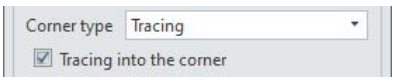
The new corner type combines original trace corner, blend corner and mitred corner. The automatic mode prioritizes the tracing result. If tracing cannot have a successful result, automatic mode will choose one among N-sided, FEM patch, and mitred.

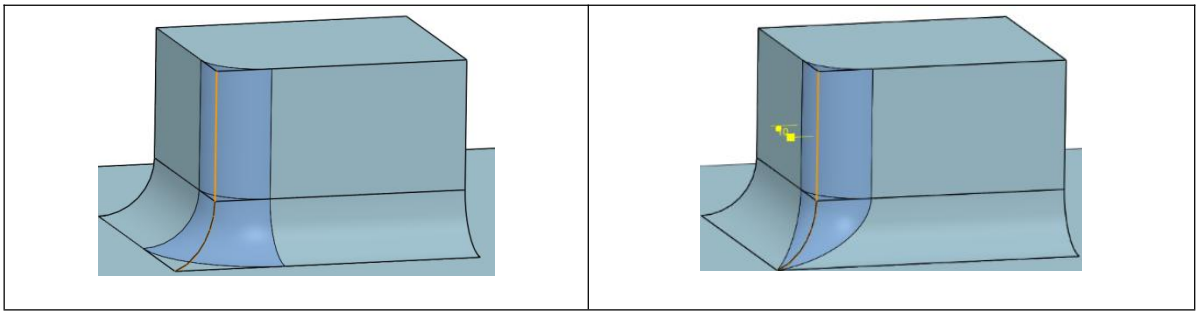
**【What users can do】**

✓ The automatic type theoretically will provide the best result. Tracing means a normal fillet face as a corner. N-sided provides a corner face created by N-sided patch. In FEM mode, the corner face is created by U/V surface. And mitred corners are made up of fillet faces that are merely intersecting.

✓ New tracing, FEM Patchm and mitred type corresponds to original trace corner, blend corner, and mitred corner separately.

✓ For the tracing type, surface can change from four-sided face to three-sided face by checking the “tracing into the corner” option.

| Corner is four-sided surface | Corner is three-sided surface |
| --- | --- |
|   |   |

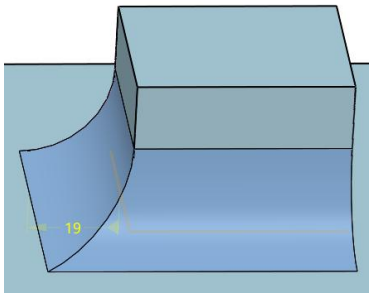
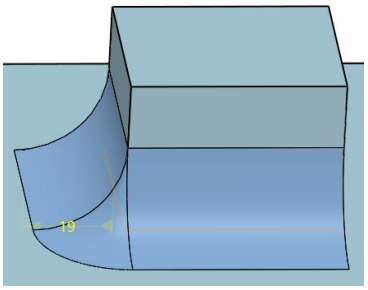


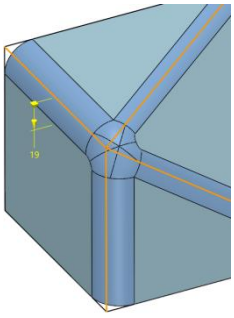
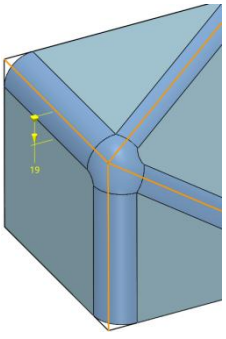
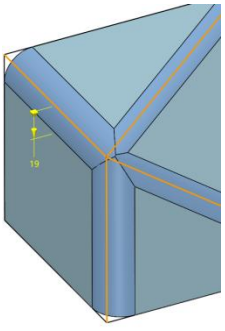
【Notes】

- "Automatic" theoretically provides the optimal solution among the last four types.

【Example】

Corner type can be altered as follows.

| Automatic | Tracing |
|---|---|
|  |  |

| N-sided | FEM Patch | Mitred |
|---|---|---|
|  |  |  |

【Where is it】

Part/Assembly Environment >> Shape >> **Fillet/Chamfer**

### 3.2.2.7 List in Face Fillet

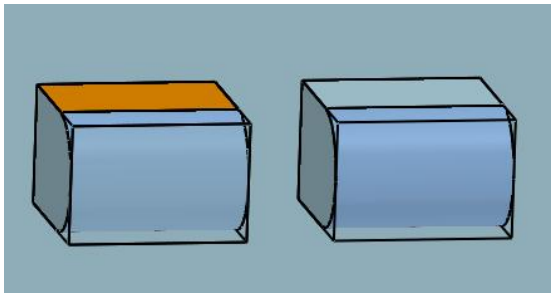
New list function in face fillet allows users to create multiple fillets at a time.

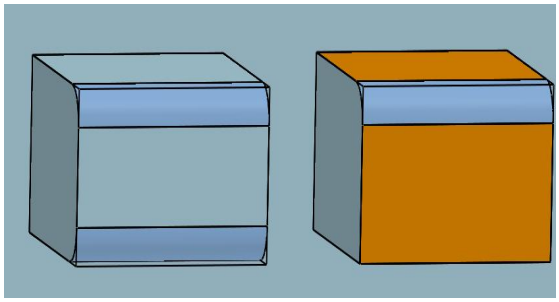
**【What users can do】**

✓ The list exists in both full-fillet and radius-fillet mode.

**【Example】**

Both full-fillet mode and radius-fillet mode support lists.





**【Notes】**

- In list, if multiple face fillets have interference, the function can not work for now.

**【Where is it】**

Part/Assembly Environment >> Shape >> **Face Fillet**

### 3.2.2.8 New Non-uniform Distance Mode for Vertex Fillet/Chamfer

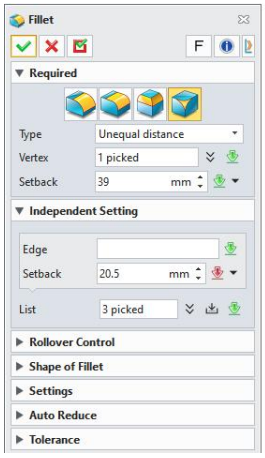
Vertex fillet/chamfer adds an unequal distance mode, supporting the setting of different distance values for each edge of the vertex

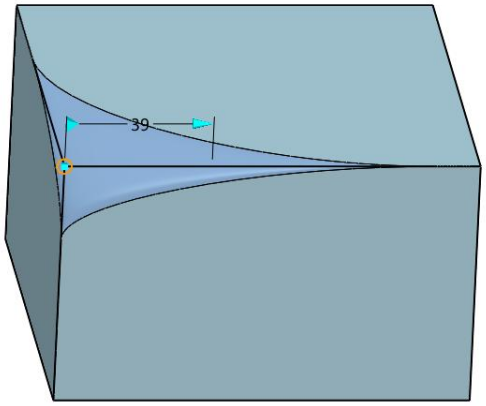
**【What users can do】**

The vertex fillet supports two modes: one is equal distance on all sides, and the other is unequal distance on all sides.

**【Example】**

An example of the unequal distance mode is shown below.




**【Notes】**

- The distance corresponds to the spatial distance from the vertex to the fillet boundary, rather than the actual length of the edge, especially for curved edges.
- When the vertex is associated with a smooth edge (the two faces associated with the edge achieve G1 continuity or above), the expected result may not be generated.

**【Where is it】**

Part/Assembly Environment >> Shape >> **Fillet/Chamfer**

## 3.2.3 “Sweep” Improvement

Structural design often requires maintaining the variation of the section of the sweep body along the path. The “parallel to plane” option is added to “section control” of sweep. This option allows the normal direction of the section of the swept body to change along the swept path while always being parallel to the selected plane.

**【What users can do】**

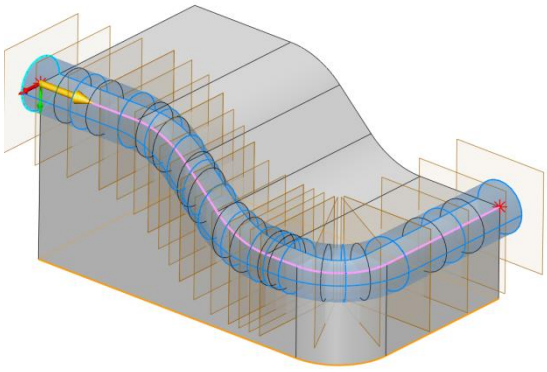
✓ Control the sweeping section to always be vertical to the selected plane, and the normal direction of the section changes with the sweeping path.

**【Notes】**

- The sweeping path needs to meet at least G1 continuity.
- The swept path cannot be vertical to the selected plane.

**【Example】**

Generate a swept body whose section is always vertical to the XOY plane and whose direction changes with the sweep path.



**【Where is it】**

Parts >> Shape >> Basic Shape >> **Sweep**

## 3.2.4 ★"Var Sweep" Improvement

There are often different shape constraints along the sweep path. Optimizing the "Var Sweep" function allows for the input of one main path plus any number of auxiliary paths to control the shape of the swept body. Both the main path and auxiliary paths support the input of chains composed of multiple segments as a single path. Both the main path and auxiliary paths support controlling the length of the sweep by dragging controls in the drawing area. Variable Sweep supports creating cross-sectional sketches in-place, and the cross-sectional sketches will be automatically positioned tangent to the starting point of the main path.

**【What users can do】**

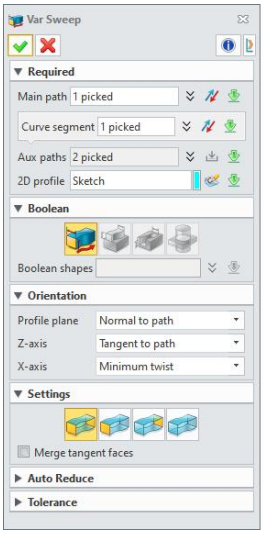
- ✓ Drag the controls in the drawing area to control the length of the sweep.
- ✓ Create a section sketch in place, and the intersection point between the path and the section will be automatically referenced to the inside of the sketch.
- ✓ The sectional sketch will be automatically positioned at the starting point of the main path tangentially.

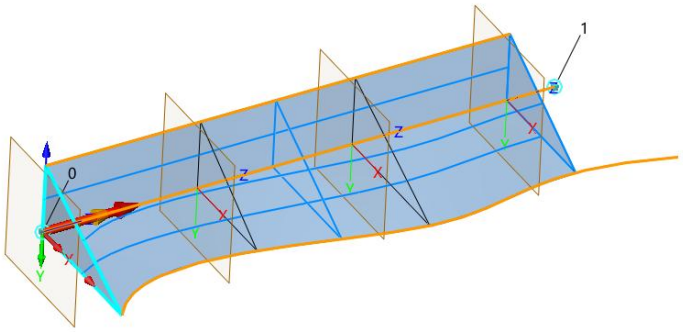
【Notes】

- The main path requires G1 continuity and auxiliary paths require G0 continuity.
- Only when the sectional sketch has a constraints relationship with the auxiliary path, the swept body changes along the path.
- Once the main path is deleted, the section sketch will be removed together.
- Create an internal sketch by the command panel to get the best variation effect.

【Example】

Define the main path using a straight line and define the auxiliary path using other curves. The auxiliary path control section automatically changes with the sweeping path.



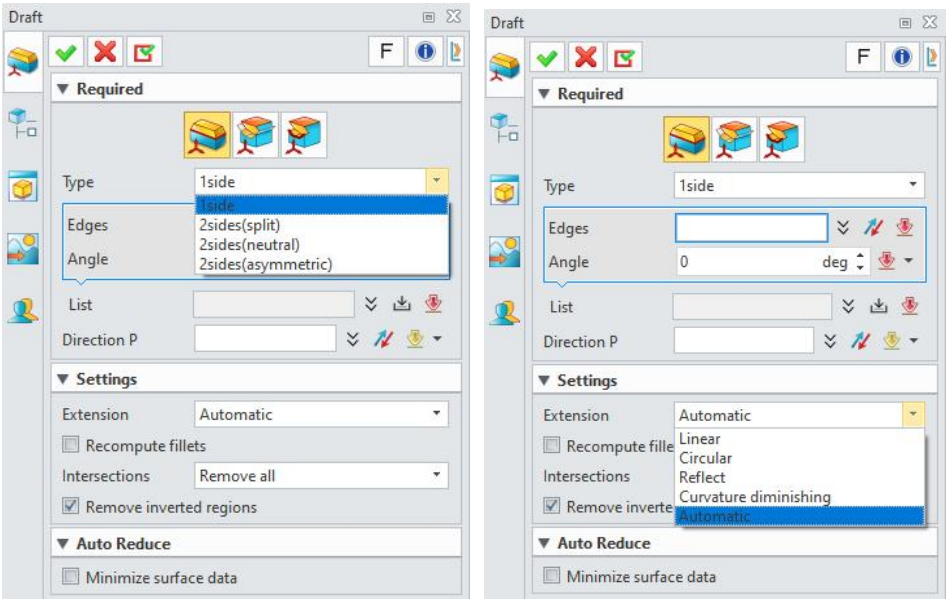

【Where is it】

Parts>> Shape >> Basic >> **Var Sweep**

## 3.2.5 ★ "Draft" Improvement

The draft feature simplifies the command interface to streamline the drafting process. A new "Automatic" extension method is provided. The "Automatic" extension can intelligently use different extension methods such as "Linear", "Circular", "Reflect", and "Curvature diminishing" for drafting based on the actual model situation, eliminating the need to frequently manually switch extension methods.

The default results will be more consistent with expectations.

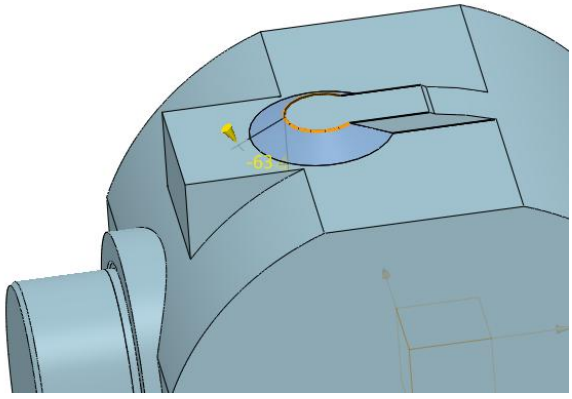



**【What users can do】**

✓ Added draft direction indication in the drawing area for viewing and adjusting draft direction by default settings.

✓ In the edge mode of the draft command, the command will automatically find the most suitable face among the two adjacent faces for draft. The directional button can switch the draft surface.

✓ The “Automatic” method can maintain the geometric properties of the draft surface. Such as the central axis, center of the circle, etc. And the surface changes more smoothly.

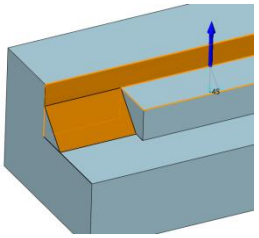
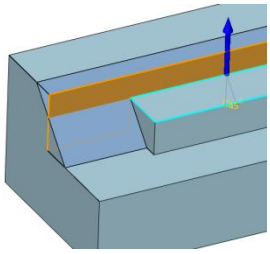
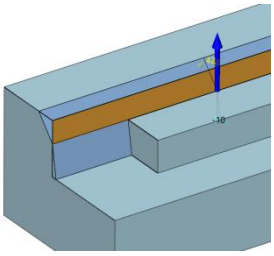
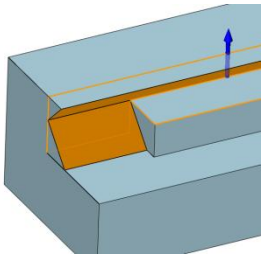


**【Notes】**

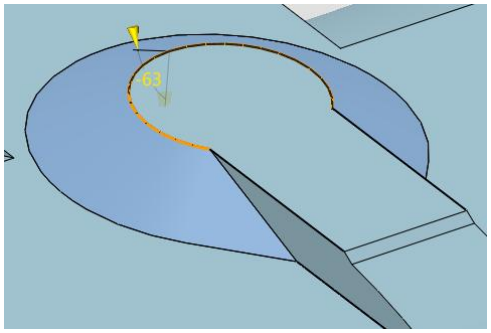
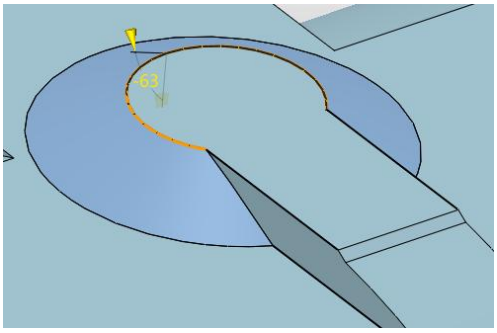
- The large angle drafting may produce unexpected results.

**【Example】**

1) Set the face as stationary faces and draft as below. When the stationary faces can divide the draft faces into two regions, the "type" with "1 side", "2 sides (split)", "2 sides (neutral)", and "2 sides (asymmetric)" would result in different draft shapes.

| 1 side | Neutral | 2 side (asymmetric) | Split |
|---|---|---|---|
|  |  |  |  |

2) By "automatic", extension method changing is no longer necessary. Without switch options such as "linear" and "circular", allowing cylindrical and conical surfaces to maintain their boundary arc properties even after draft.

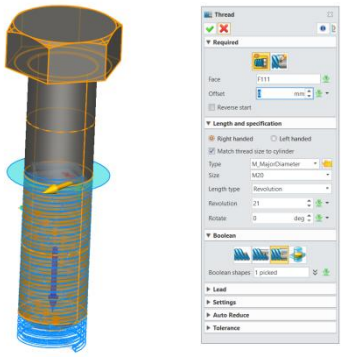
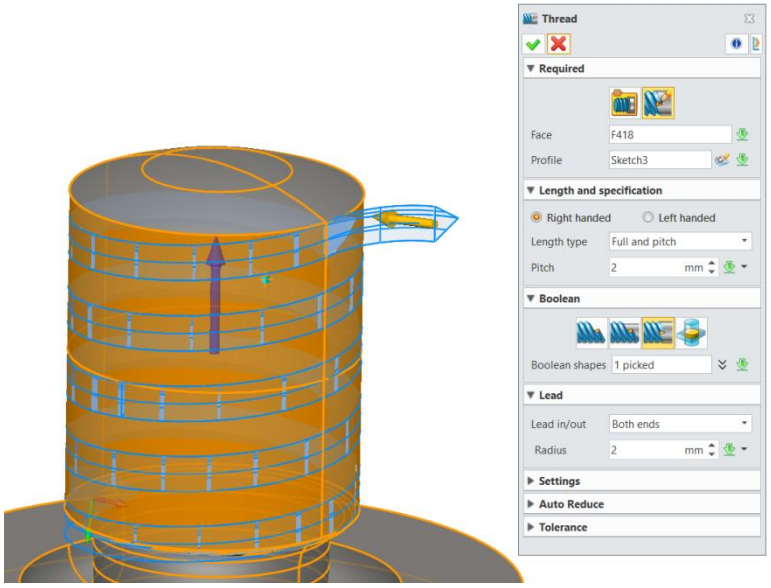
**【Where is it】**

Parts>> Shape >> Engineering Feature >> **Draft**

## 3.2.6 ★Thread Enhancement

The thread function supports the creation of standard cross-sectional threads. It enables intelligent matching of the dimensions and Boolean forms of standard-specification threads by selecting cylindrical

surfaces, hole surfaces, and specified specifications. Additionally, it has added the function of controlling thread shape through input of the total length. Currently, it supports the M series and Rp (55° cylindrical internal thread) series.

| Standard profile | Custom profile |
| --- | --- |
|  |  |

【What users can do】

✓ Standard profile mode supports identifying and matching specifications based on the diameter of the hole/shaft. Meanwhile, it supports two kinds of specifications: tap drill size and major diameter.

✓ The Boolean type of standard profile thread will automatically determine whether to perform cutting or adding operations based on whether the selected surface is on a hole or shaft.

✓ It can quickly and efficiently generate a large number of thread shapes with a large number of turns.

✓ Thread will be cut at the starting position of the selected surface.

✓ Support defining thread specifications through parameters such as length/number of turns/pitch (only in custom profile mode), and thread length can be set to "full".

✓ Support customizing profiles or adding more specifications to existing standard profiles.

✓ In custom profile mode, thread function supports creating sketches at the starting position based on the selected face.

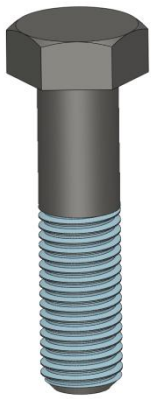
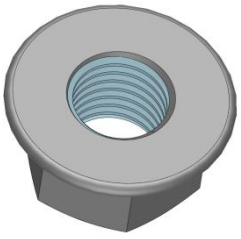
✓ The thread shape allows for further adjustments, including left/right handedness, offset (based on the starting position), and the rotation angle of the thread along the axis of rotation.

**【Notes】**

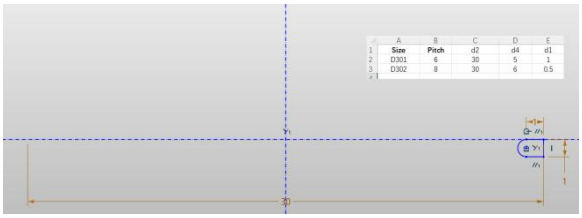
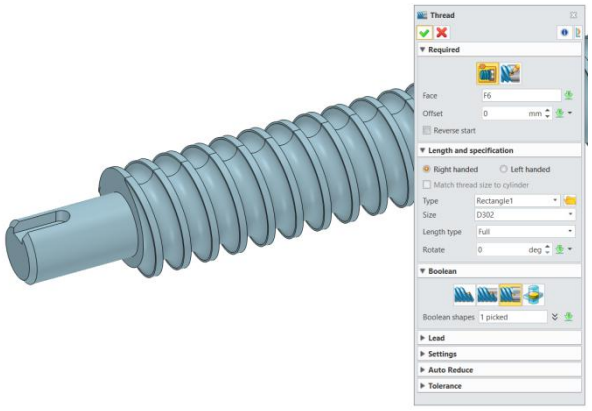
- Thread function only supports users to select and attach on a single cylindrical/hole surface.
- When inputting the size of the thread, it is necessary to ensure that the height of the thread profile is less than the pitch. Or in other words, the starting and ending section of each turn of the thread cannot overlap with each other, otherwise ZW3D may provide an unexpected result.
- The standard profiles are based on major diameter or tap drill size. When the hole/shaft diameter is too large or too small, it may result in ineffective cutting/adding.
- The thread profile dimensions vary under different standards (GB, ISO). Therefore, it is not recommended to use thread shapes directly for production and processing.

**【Example】**

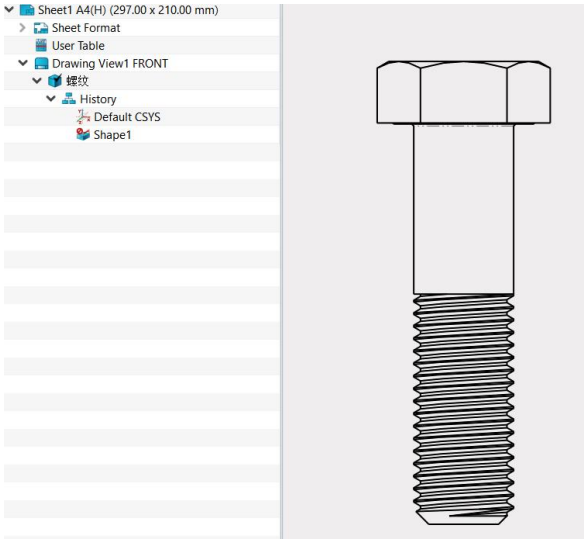
1) Attach actual thread to the cylindrical/hole surface to calculate the mass of large-sized threads precisely.

| Bolt | Nut |
| --- | --- |
|  |  |

2) Generate thread based on enterprise standard profiles, and quickly reuse them in the thread function, such as ball screw.

| Format design | Format reuse |
| --- | --- |
|  |  |

3) When exporting to neutral format, keep the thread shape for annotation or machining reference in drawing sheet and other environments.



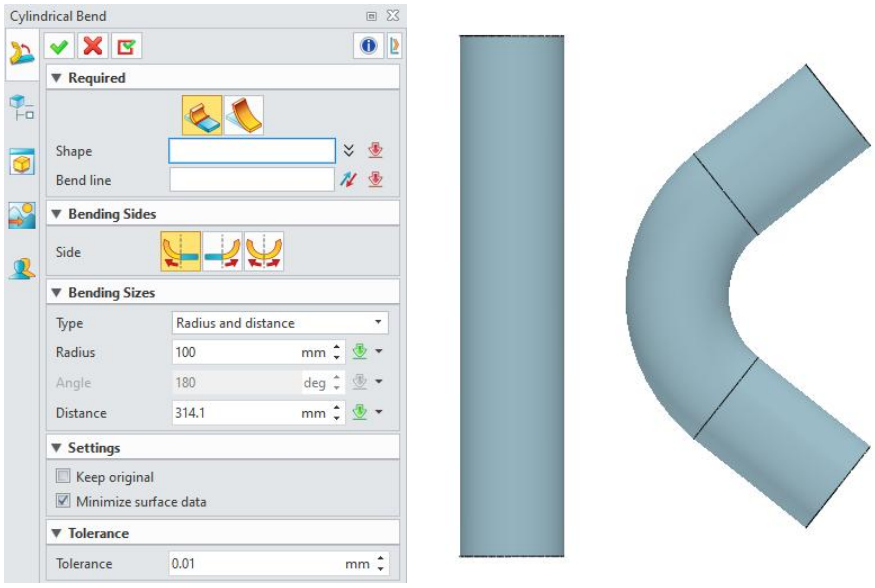

**【Where is it】**

Part/Assembly >> Shape >> Engineering Feature >> **Thread**

## 3.2.7 ★ “Cylindrical Bend” Improvement

In the design of flexible parts, it is often necessary to locally bend and deform them. Cylindrical

bend supports local bending now, allowing parts to achieve more flexible deformation designs.




Cylindrical Bending

**【What users can do】**

✓ Control the bending shape of parts through sketches and setting radius, angle, and distance parameters.

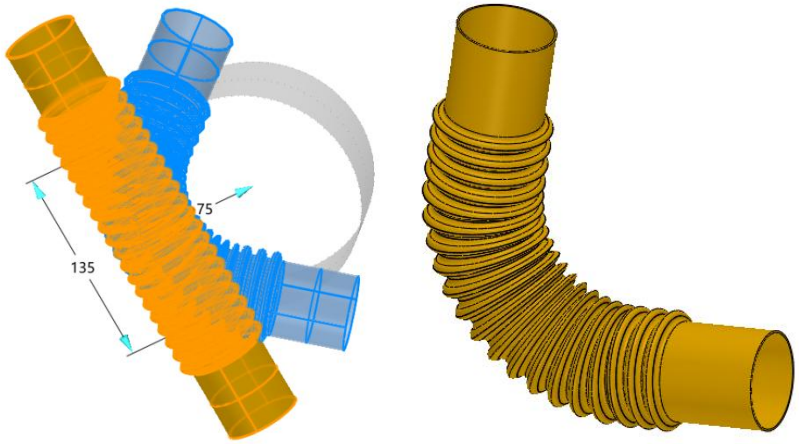
✓ Control the part to perform local bending on one side or in the reverse direction.

**【Notes】**

- The new mode of cylindrical bending only supports bending through sketches.

**【Example】**

Corrugated tubes are common bent parts. Pick up the corrugated tube and bending sketch in the new mode “by sketch” of cylindrical bending. Set bend distance 135mm and bend radius 75mm to bend the corrugated tube. The effect is as follows.




Bent corrugated tube

**【Where is it】**

Parts>> Shape >> Morph >> **Cylindrical Bend**

## 3.2.8 Pattern Supporting Skipping Overlapping Objects

The point pattern mode in the commands for pattern features/pattern geometries/pattern assemblies is optimized. When a pattern instance overlaps with the base one, a warning window will pop up, and support skipping the overlapping pattern instance.

**【What users can do】**

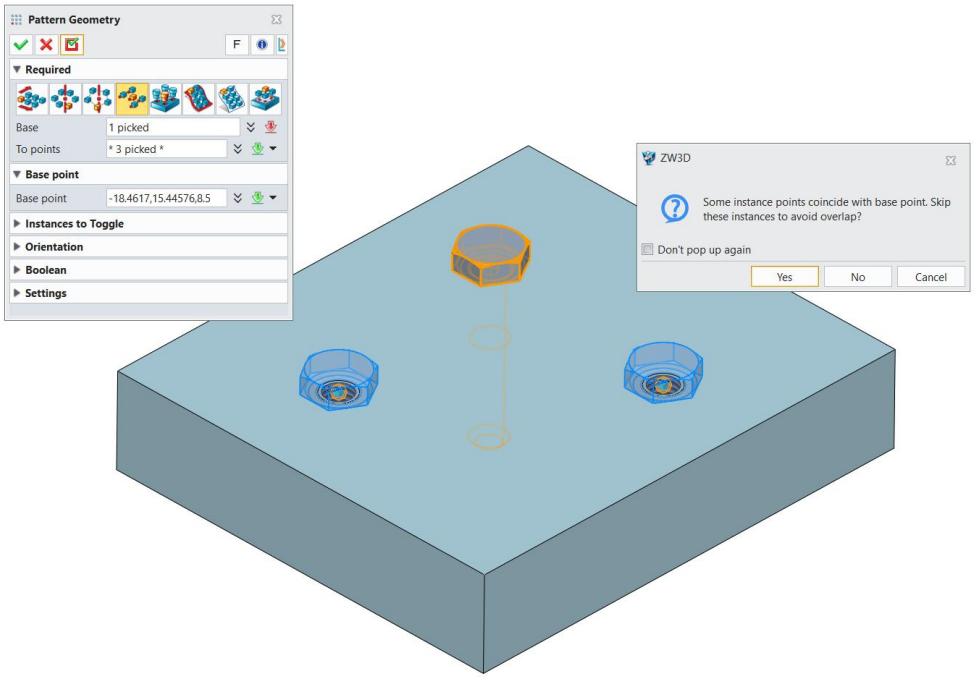
- ✓ Avoid pattern instances overlapping with the base.

**【Notes】**

- Once the "Don’t pop up again" is checked, the skipping overlap instance function will remain enabled until the next time ZW3D is opened.

**【Example】**

In pattern feature/ pattern geometry/ pattern components, exclude instances that overlap with the base instance to avoid redundant calculations of the pattern.




**【Where is it】**

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

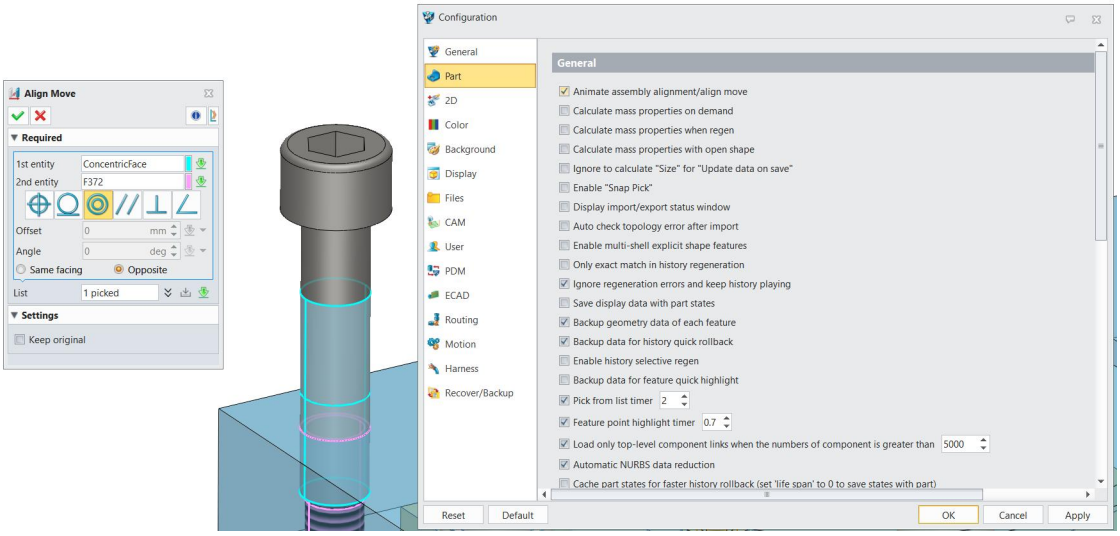
Assembly Environment >> Assembly >> Basic Editing >>**Pattern**

## 3.2.9 Align Move Supporting Disabling Animation

Align move function supports turning off animations. After turning off the "Animate assembly alignment/align move" in configuration, the final result will be displayed directly when moving two shapes.

**【What users can do】**

✓ In multi-shapes design, you can turn off "Animate assembly alignment/align move" in configuration to skip the alignment animation and quickly align two entities.

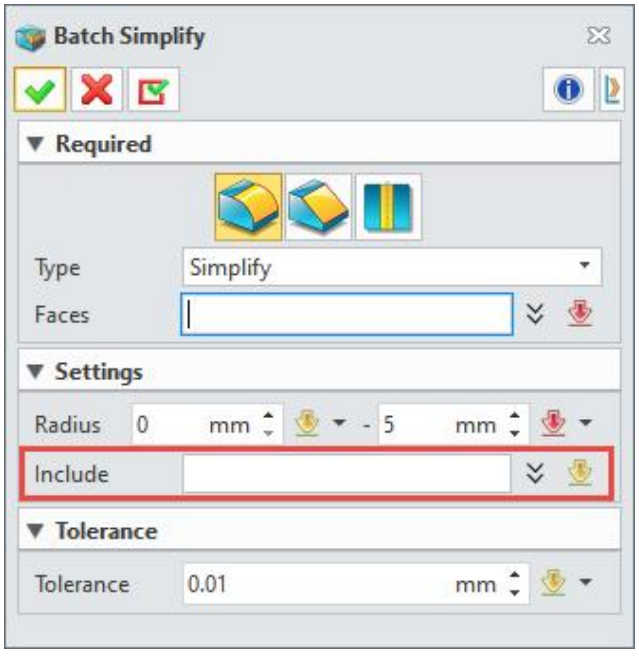



**【Where is it】**

Part/Assembly >> Shape >> Basic Editing >> **Align Move**

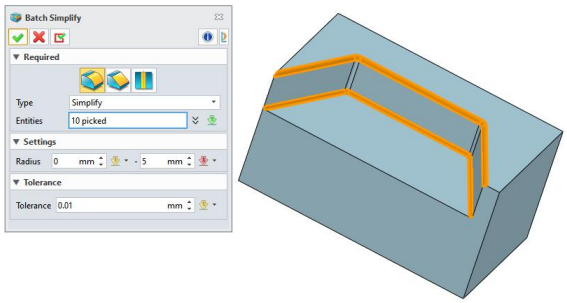
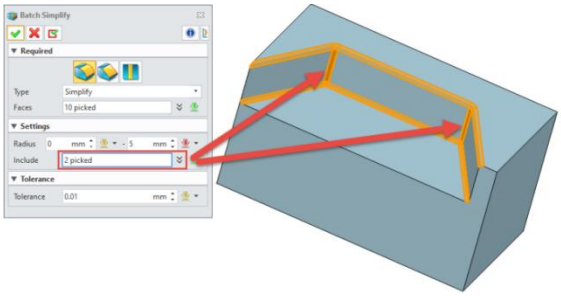
## 3.2.10 Batch Simplify Function Improvement

Batch Simplify function adds an "include" option box, which allows users to select some faces individually as simplified input.



**【What users can do】**

✓ Some faces objects that are not in the range of batch selection can be selected as input and simplified.

| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|  |  |

**【Where is it】**

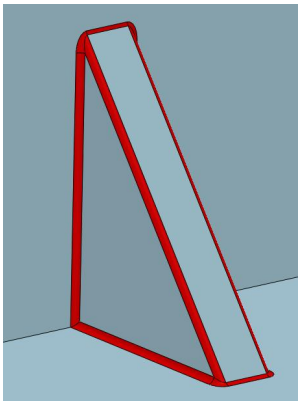
Part/Assembly Environment >> Mold >> Tools >> **Batch Simplify**

## 3.2.11 Rib/Rib Network Optimization

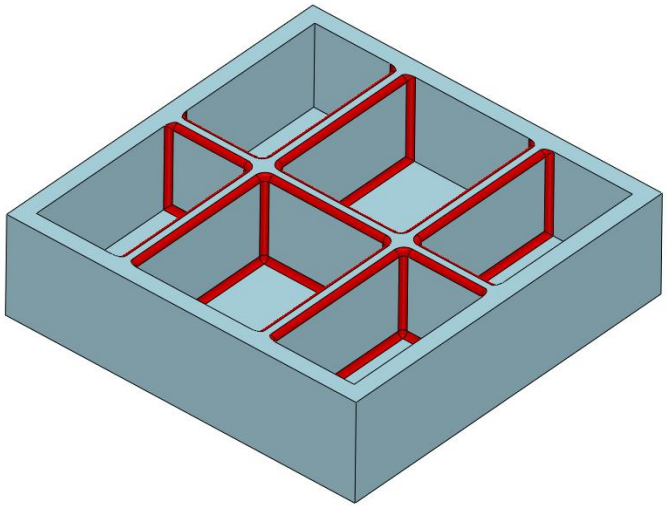
In the creation process of ribs or network ribs, it is necessary to add fillets to the ribs/network ribs. Rib/Rib Network supports the direct addition of fillets in the command, making it more convenient for users to use.

**【What users can do】**

✓ Directly add fillets in the Rib/Rib network command.



Rib



Rib Network

**【Notes】**

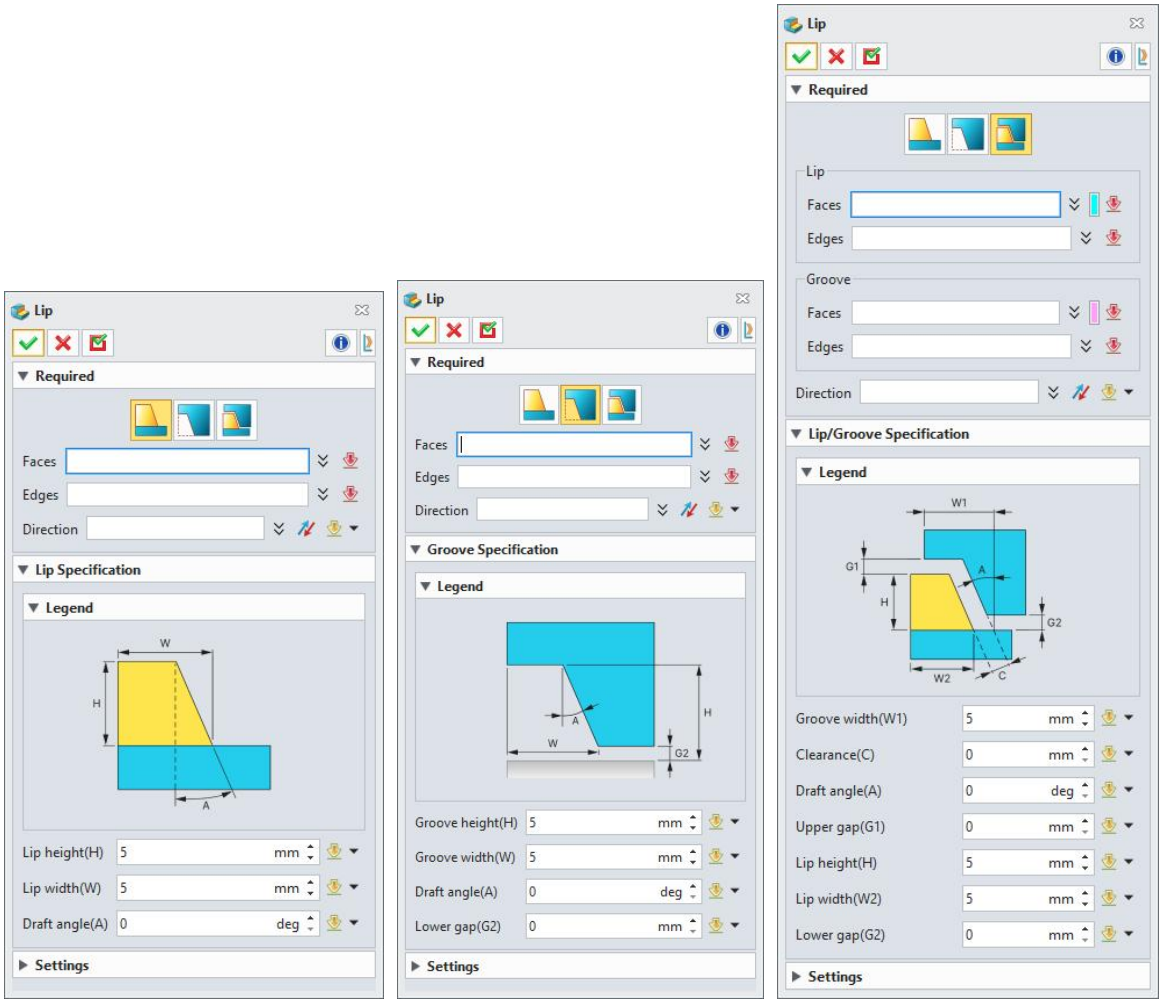
- The fillet in the command is divided into an internal edge fillet and exposed edge fillet, which can be used by users as needed.

## 【Where is it】

Part/Assembly Environment >> Shape >> Engineering Feature >> **Rib/Rib Network**

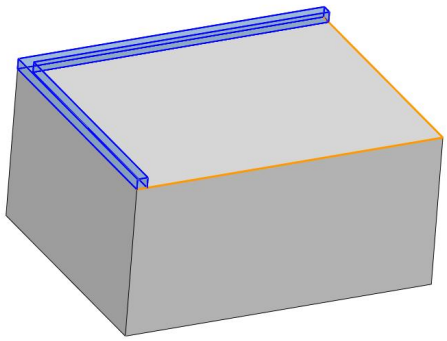
## 3.2.12 Lip Optimization

The lip command expands the scope of capabilities and simplifies the operation process. Users can choose to create male lips, female lips, or overlapping male and female lips based on actual situations. Creating male or female lipscan be done in individual parts, while overlapping male and female lips can be done both in parts and across components in assemblies.

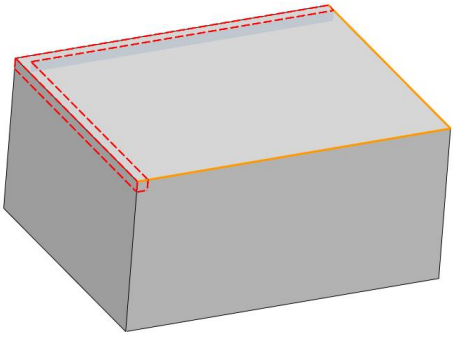



**【What users can do】**

✓ Create male or female lips separately in the part, and support adjusting the shape of the lips through parameters.

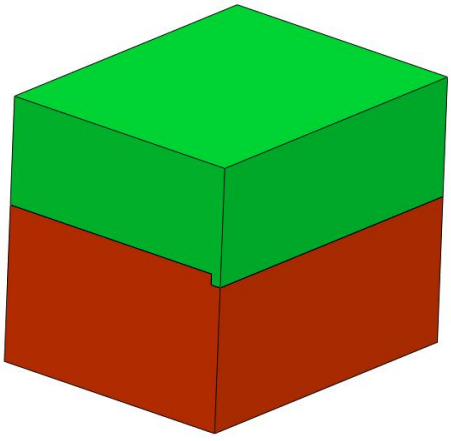


male lips



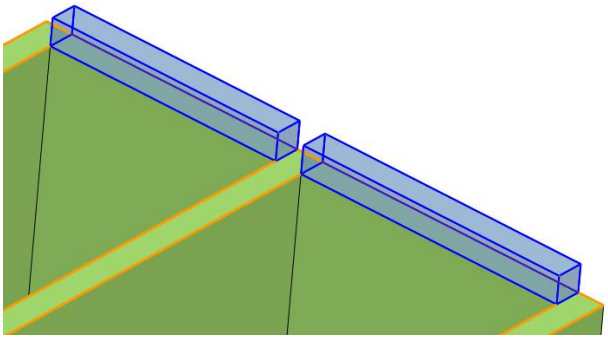
female lips

✓ Create pairs of male and female overlapping lips on different bodies within the same component, and support adjusting the shape of the lips through parameters.

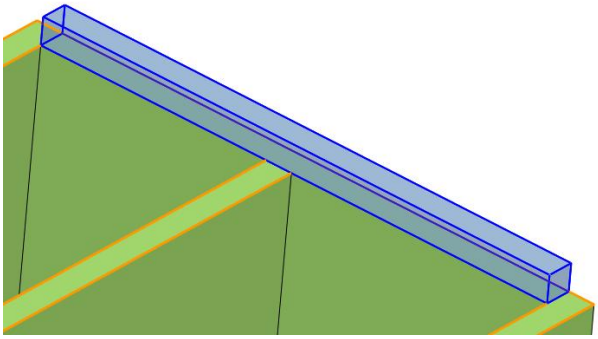


✓ Create pairs of male and female overlapping lips in different parts under the same assembly, support adjusting the shape of the lips through parameters, and keep the parameters of the male and female lips synchronized.

✓ Support skipping gaps, which can maintain the continuity of the lip edge in scenes with tendons and ribs.

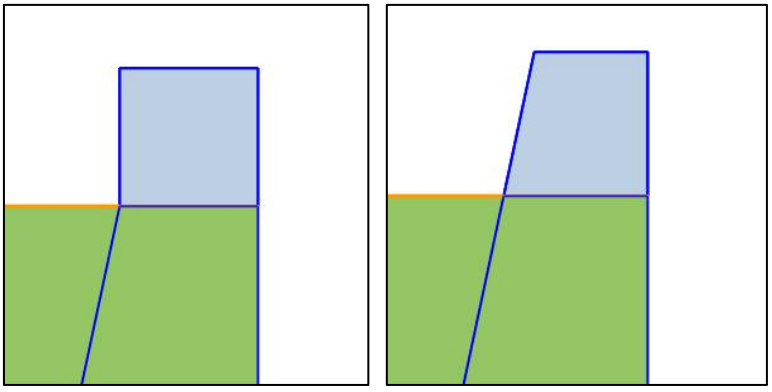


✓ Not skipping the gap



✓ Skip the gap

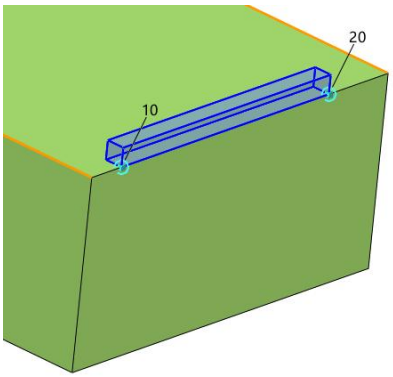
✓ Support two methods: maintaining the direction along the lip edge and the direction along the thin wall.



Along the lip direction

Along the thin wall

✓ Support creating lip edges locally on the edge line, leaving gaps at both ends.




**【Notes】**

- The angle between the lip direction and the normal direction of the surface should not be too large, otherwise it will cause the lip to be too pointed and not meet the requirements for lip opening and closing.
- The male and female lips created across components need to be redefined in place in the assembly and cannot be individually redefined in the part environment.

**【Where is it】**

Part/Assembly Environment >> Shape >> Engineering Feature >> **Lip**

## 3.2.13 ★Slot Feature

New slot function supports adding grooves on the plane or ring grooves on cylindrical/conical surfaces. Slot feature supports users to pick multiple points on a single surface. It is convenient to add engineering features such as keyways and O-ring grooves to shaft parts. Users can also add sliding fit grooves for connecting parts or standard parts.

**【What users can do】**

✓ Define slot features by inputting parameters and attach them to shapes quickly.

✓ Supports ring slot and plane slot, which can be added with fillets.

✓ Supports picking multiple points on a single face to add multiple slots and multiple solids.

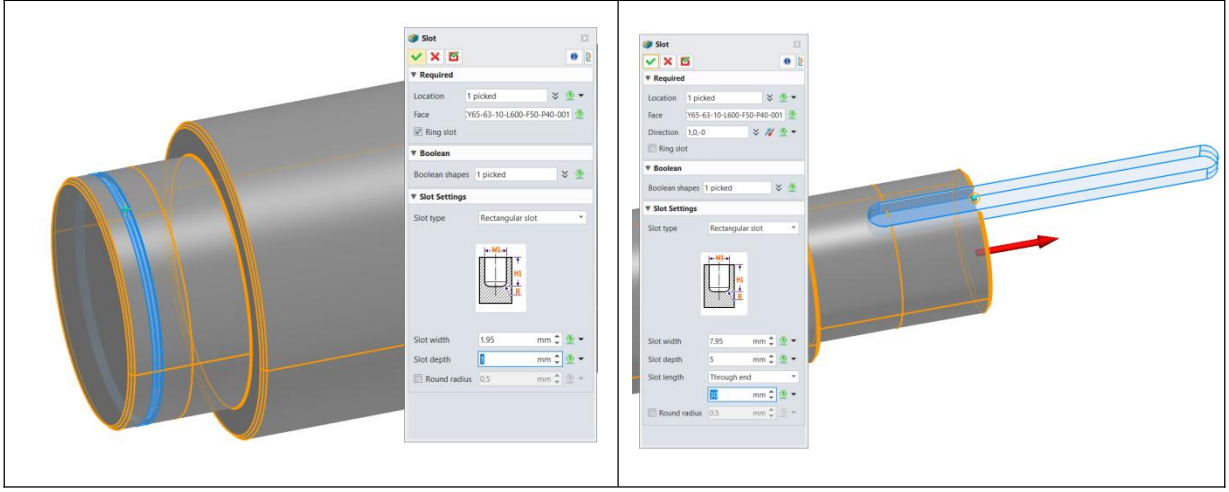
✓ Supports four commonly used slot shapes: rectangular, spherical, dovetail, and T-slot.

**【Notes】**

- Total depth of the slot should be less than the radius of the cylindrical/conical face where the slot is located.
- When multiple slot feature positions are too close, the slot area will be merged. It is recommended to check whether the results meet expectations after the slot feature is created.

**【Example】**

Add undercut, keyway, or wheel crossing grooves on a shaft.

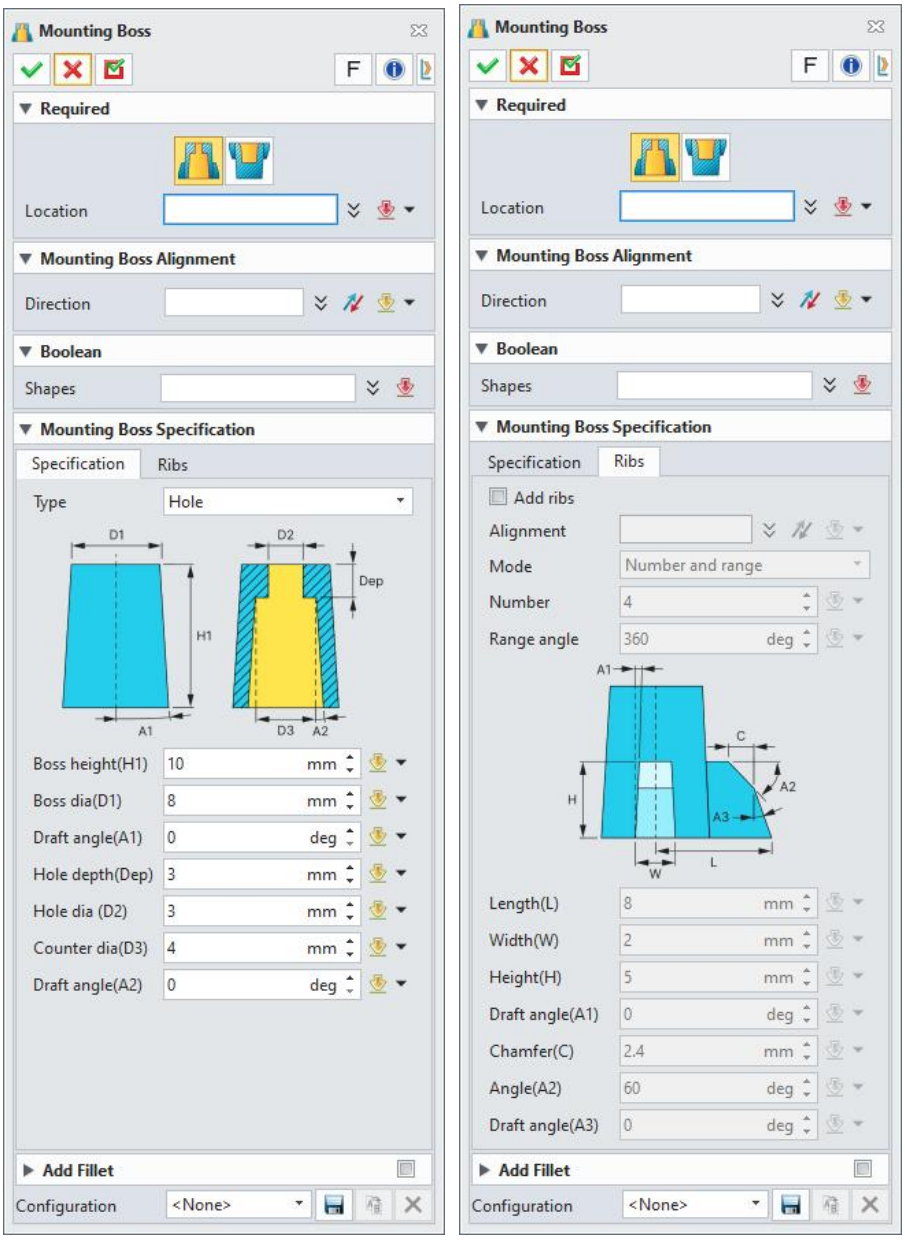
| Undercut | Keyway |
| --- | --- |
|  | |

**【Where is it】**

Part/Assembly Environment >> Shape >> Engineering Feature >> **Slot**

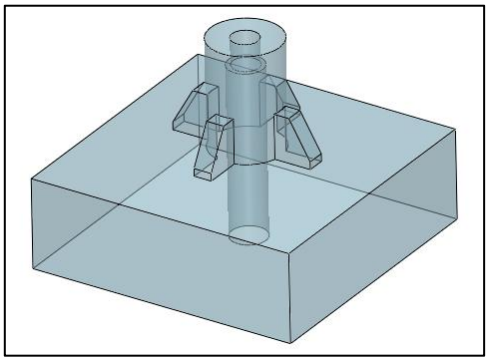
## 3.2.14 ★New Mounting Boss

Mounting Boss is a commonly used structure in plastic part design. A plastic part may have multiple Mounting Boss with different positions, sizes, and shapes. To meet this scenario, the new Mounting Boss commands can quickly create and modify Mounting Boss using Mounting Boss commands, greatly improving the design efficiency of plastic parts.

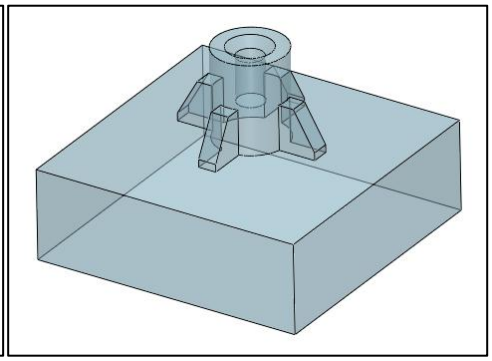



**【What users can do】**

✓ Support the creation of two forms of head bosses (through holes) and threaded bosses (threaded holes).

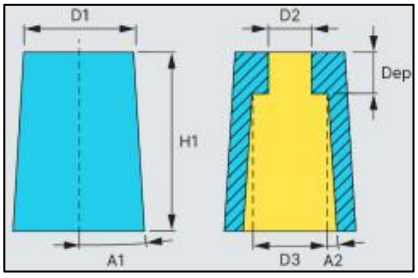


Head Boss

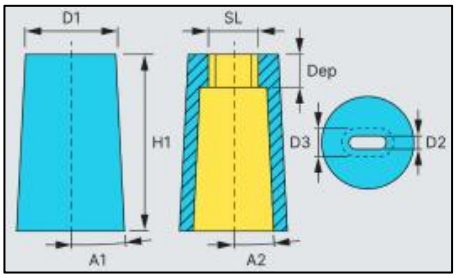


Thread Boss

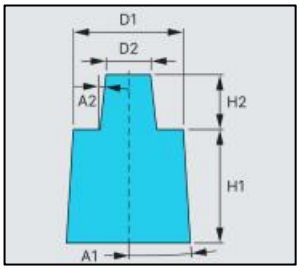
✓ The head boss supports three types: holes, slots, and pins, and the shape of the Mounting Boss can be adjusted through parameters.




Hole

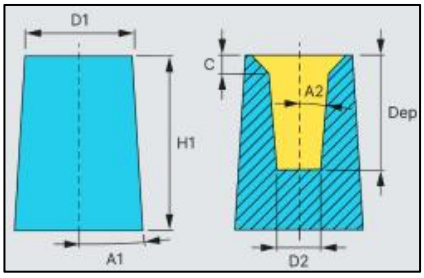



Slot

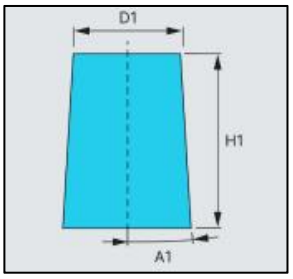



Pin

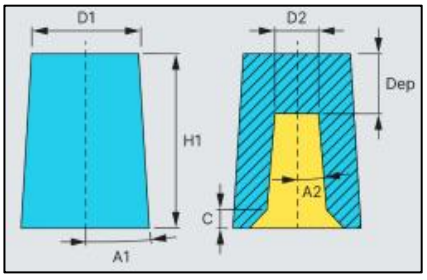
✓ The threaded boss supports three types: positive hole, without hole, and reverse hole, and the shape of the Mounting Boss can be adjusted by parameters




Positive hole

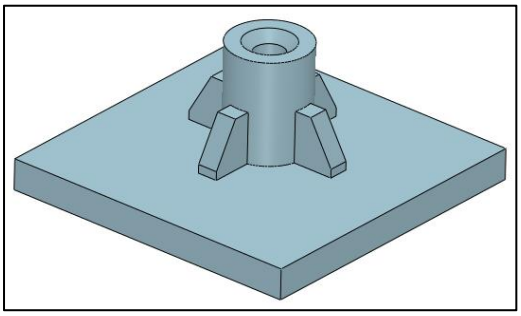



Without hole

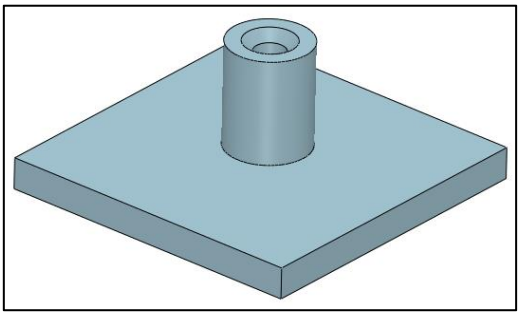



Reverse hole

✓ Support the control of whether to create ribs, as well as the quantity and arrangement angle of ribs.

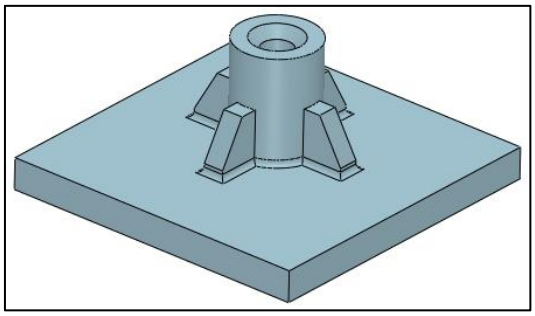


With ribs

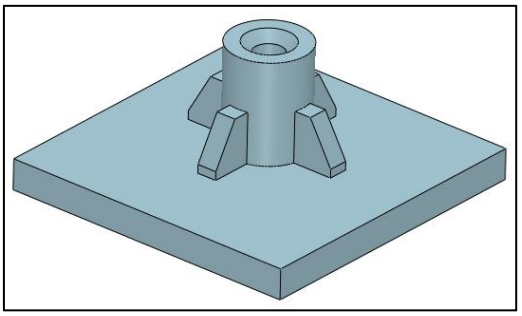


Without ribs

✓ Support controlling whether to create bottom rounded fillet.



Create bottom fillet



Do not create bottom fillet

✓ Support configuration file recording of Mounting Boss parameters, users can quickly create Mounting Boss that are the same as before by selecting the configuration.

**【Notes】**

- The Mounting Boss cannot be too close to the edge of the face, otherwise it will fail to create.
- Setting the radius value of the bottom fillet too large may cause the fillet to not be generated properly.

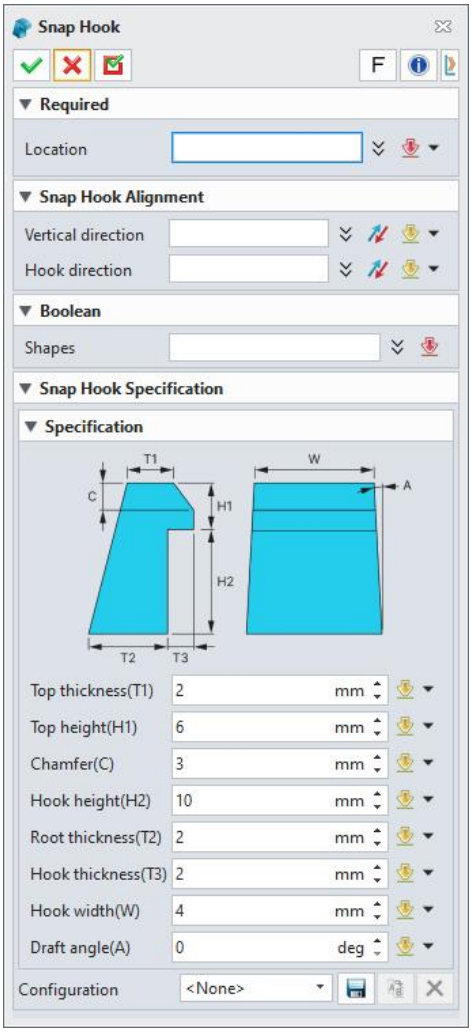
**【Where is it】**

Part/Assembly Environment >> Shape >> Engineering Feature >> **Mounting Boss**

## 3.2.15 ★New Snap Hook

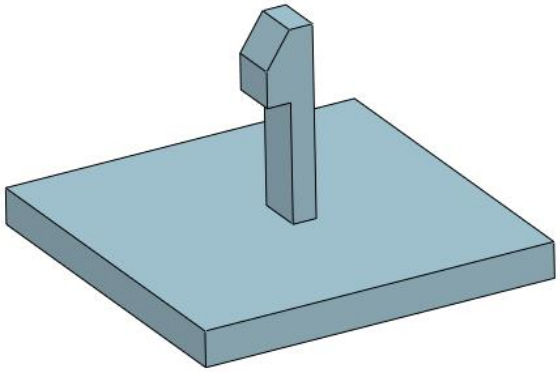
Snap Hook is a commonly used structure for connecting and fixing plastic parts. In design scenarios where strength requirements are not particularly high, snap hooks can replace screws as a fixed

component structure. The new Snap Hook command allows users to quickly create snap fasteners by entering their position and parameters. Users can also configure snap fastener parameters in advance to generate snap fasteners with just one click. In scenarios where snap fasteners are frequently created, snap fastener commands can greatly improve design efficiency.




**【What users can do】**

✓ Quickly create snap hooks in parts, supporting adjusting the shape of the snap hook through parameters.



✓ Support configuration file recording of snap hook parameters, users can quickly create snap hooks that are the same as before by selecting the configuration.

**【Notes】**

- The snap hook's position cannot be too close to the edge of the face, otherwise it will fail to create.
- The snap hook should intersect with the entity in the selected direction, otherwise it will fail to create.

**【Where is it】**

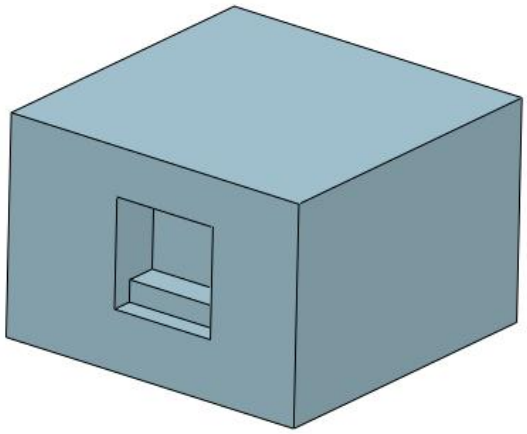
Part/Assembly Environment >> Shape >> Engineering Feature >> **Snap Hook**

## 3.2.16 ★New Snap Hook Groove

Snap Hook Groove is a commonly used structure for connecting and fixing plastic components, usually configured with a Snap Hook to secure plastic components. The new Snap Hook Groove command allows users to quickly create a Snap Hook Groove by entering its position and parameters. Users can also configure the Snap Hook Groove parameters in advance and generate a Snap Hook Groove with just one click. In scenarios where Sanp Hook Grooves are frequently created, Snap Hook Groove commands can greatly improve design efficiency.

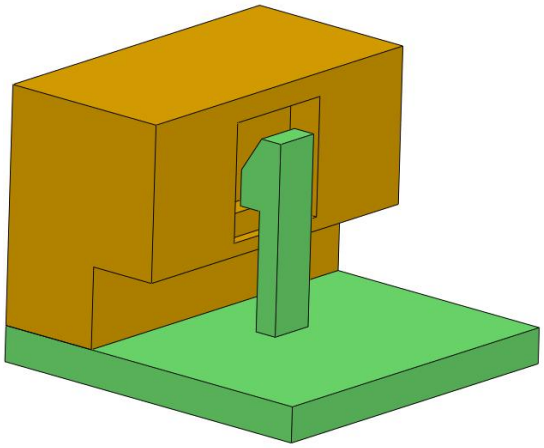
**【What users can do】**

✓ Quickly create snap hook grooves in the parts and support adjusting their shape through parameters.



✓ Support selecting the surface of the snap hook that needs to be matched and quickly creating a snap hook groove that matches it.

✓ When assembling in place editing, it supports picking snap features across components and creating corresponding grooves, which can maintain data association after creation.



✓ Support configuration file recording of snap hook groove parameters, users can quickly create the same one as before by selecting the configuration.

**【Notes】**

- The groove position must intersect with the Boolean body, otherwise the creation will fail.

## 【Where is it】

Part/Assembly Environment >> Shape >> Engineering Feature >> **Snap Hook Groove**

## 3.2.17 Extrusion Divide & Extrusion Trim Enhancement

In mold designing, the products require the use of lines for cutting and trimming the core, as well as cutting inserts and sliders. Similarly, air duct designing in the equipment industry requires the calculation of the sectional area. "Extrusion Divide" & "Extrusion Trim" can quickly segment entities and make design easier and more convenient.

**【What users can do】**

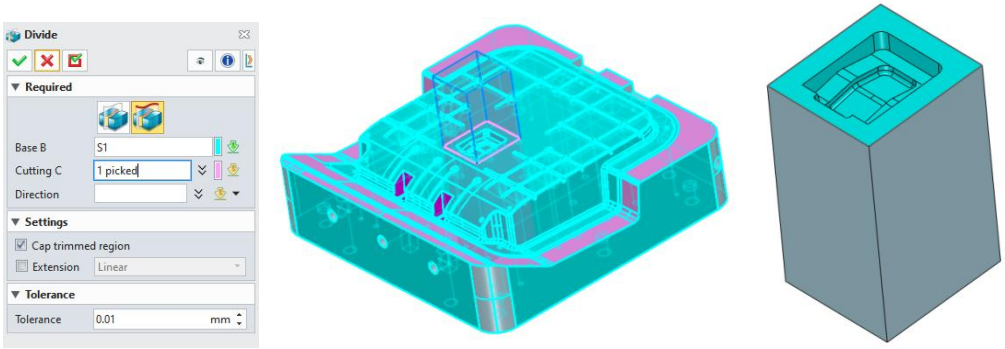
✓ Divide/Trim the shape in the specified direction by sketches, lines, or edges.

**【Notes】**

- The “Face Divide” and “Face Trim” no longer allow input of sketches.

**【Example】**

Using sketches to surround the features that need to be segmented can quickly segment/trim the shape.

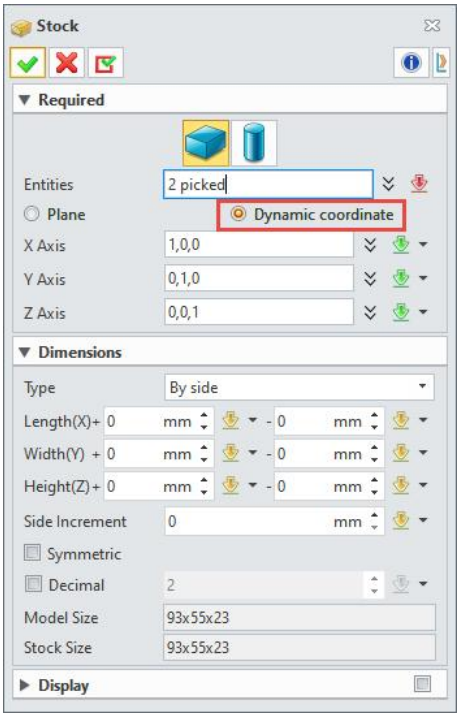



**【Where is it】**

Part >> Shape >> Basic Shape >> **Divide/Trim**

## 3.2.18 Stock Function Improvement

Stock function is a high-frequency function in mold design. Improving the usability of Stock function can greatly improve efficiency. Stock function adds the option of "Dynamic coordinates" and users can quickly adjust the shape and size generated by the stock through dynamic coordinates.

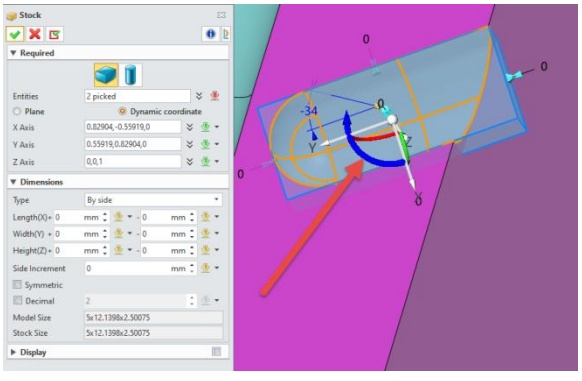
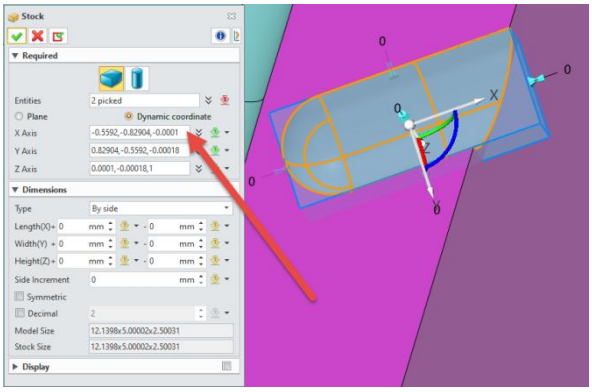



**【What users can do】**

✓ The orientation of stock is directly controlled by rotating the dynamic coordinates of the drawing area.

✓ By selecting the alignment direction of the X, Y and Z axes through the command panel, the orientation of the stock can be controlled.

**【Example】**

| Rotational dynamic coordinates | Select the alignment direction |
|---|---|
|  |  |

**【Where is it】**

Part/Assembly Environment >> Shape >> Engineering Feature >> **Stock**

Part/Assembly Environment >> Mold >> Tools >> **Stock**

Part/Assembly Environment >> Electrode >> Assist Tools >> **Stock**

## 3.3 Assembly Design

### 3.3.1 ★Assembly Constraints

In assembly design, users often spend a significant amount of time positioning components, with the most common method being to install components in specified positions using various constraints. The creation process for assembly constraints is optimized, focusing on inferring the intent of the constraints. When users select objects in the assembly environment, the software infers the intent for constraint creation, simplifying the steps for creating constraints and improving the efficiency of component installation and positioning.

**【What users can do】**

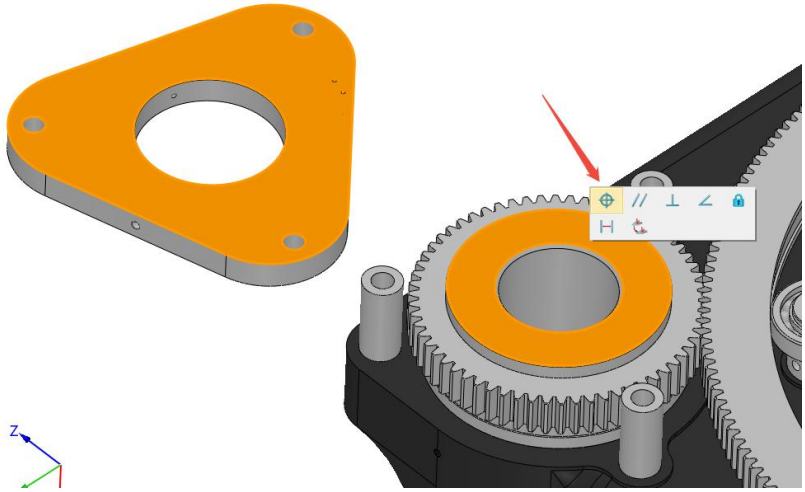
- ✓ Quick creation of constraints by selecting objects in the drawing area.
- ✓ Judging the creation type of constraints based on selected objects.
- ✓ Locking multiple components at once.
- ✓ Support for creating constraints on objects selected from any component in the coordinate system.
- ✓ Automatic conversion of constraint icons.
- ✓ Adaptive filling of angle values for angular constraints.
- ✓ Querying constraint status to delete selected constraints.

**【Notes】**

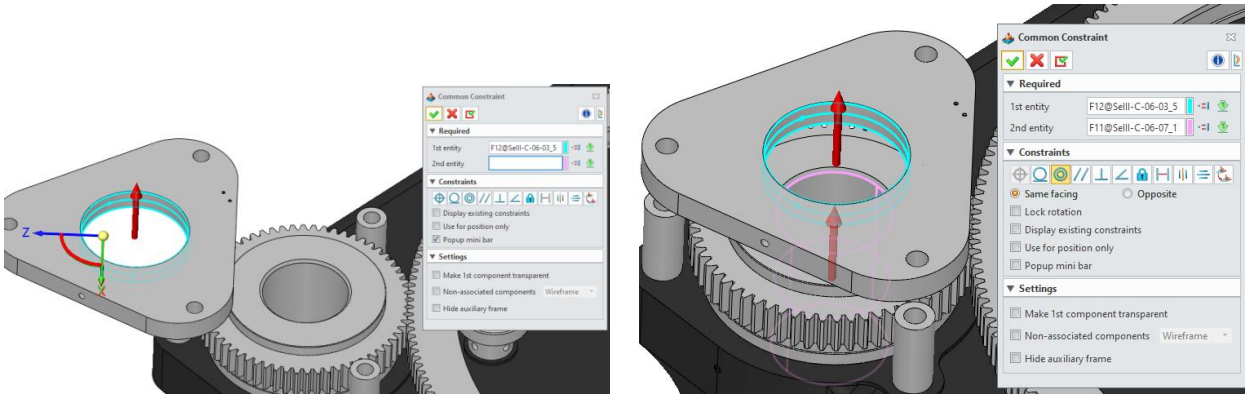
- ✓ Constraints created in the old version with the 'coincidence offset' option checked will automatically convert to distance constraints when opened in ZW3D 2026.

**【Example】**

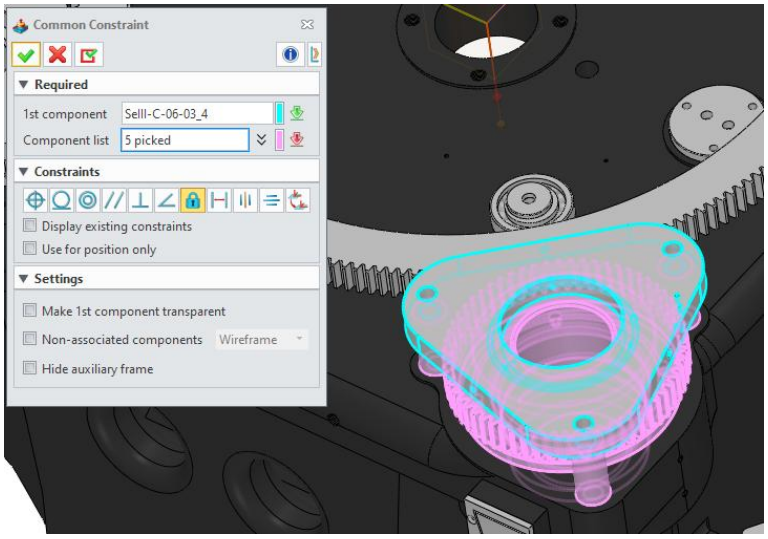
1) In the drawing area, select faces on two different components, choose the constraint type to create from the popup tooltip, and complete the constraint creation.



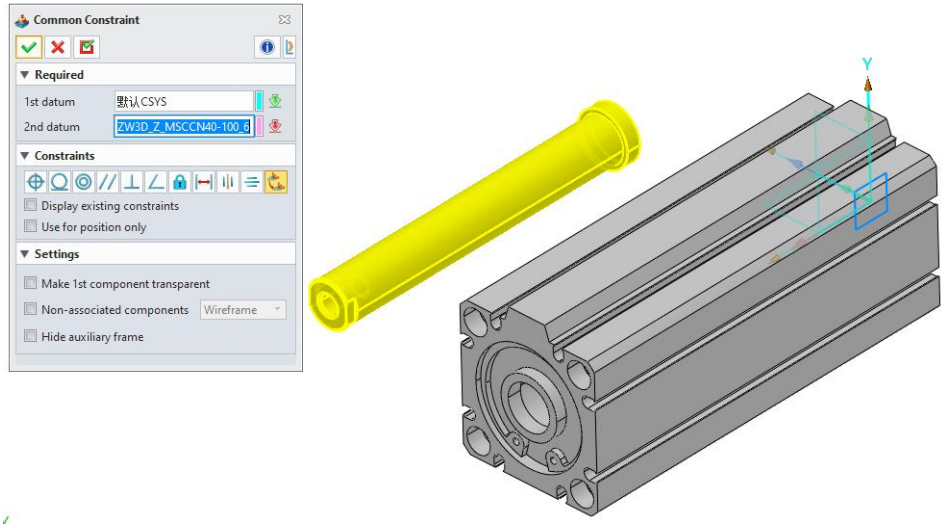
2) Start the constraint command, select the objects for which constraints need to be created, such as two cylinders, and automatically recognize and generate a concentric constraint.




3) Start the constraint command, select the lock constraint, choose multiple components that need to have lock constraints created, and complete the constraint creation in bulk.




4) Start the constraint command, select the coordinate system constraint, choose the components for which the default coordinate system constraint needs to be created, and automatically input the default coordinate system to complete the constraint.




**【Where is it】**

Assembly Environment >> **Constraint**

Assembly Environment >> **Plot area**

## 3.3.2 ★Copy with Constraint

In assembly design, there are a large number of commonly used and standard components. For

components that have already been precisely positioned, quickly reusing constraints to copy components to new locations can significantly improve assembly efficiency. The "Copy with Constraints" command has been enhanced to support copying multiple components at once and reusing the originally selected constraint objects. In addition, this command now supports complex constraint types such as centering, symmetry, slots, etc., and the interaction mode is also more convenient and efficient, thereby greatly promoting the efficient reuse of components.

**【What users can do】**

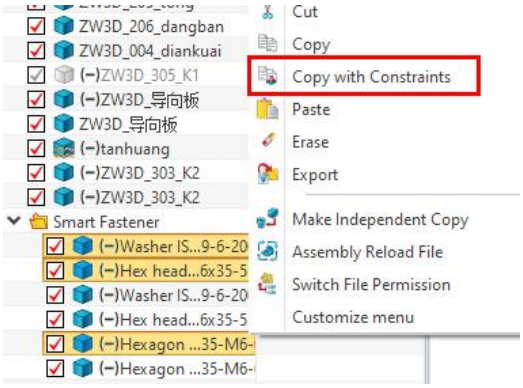
✓ With copy with constraints, multiple components can be copied at once and the original constraint objects can be reused.

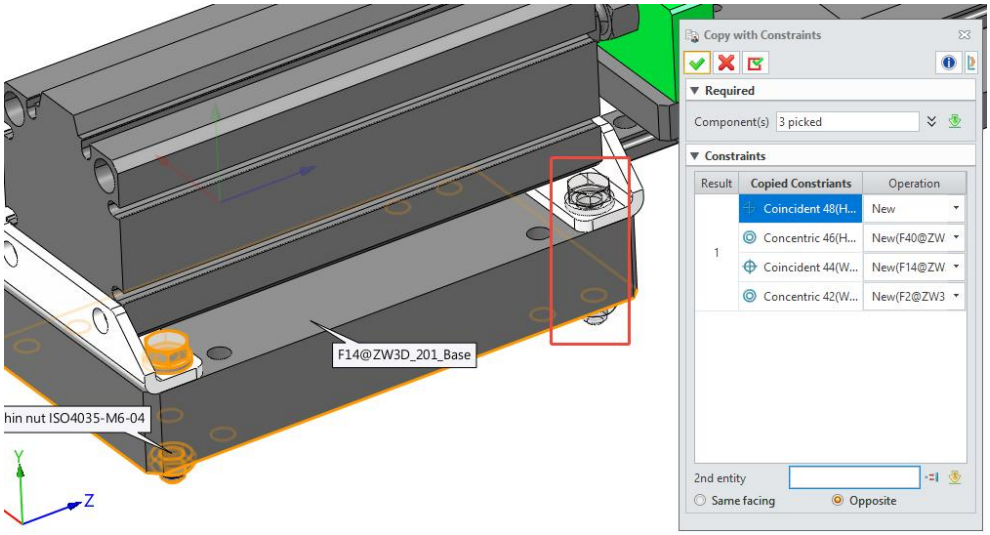
✓ The original selected constraint objects can be quickly reused through the middle mouse button.

✓ Copy with Constraints supports complex constraint types such as centering, symmetry, slots, paths, etc.

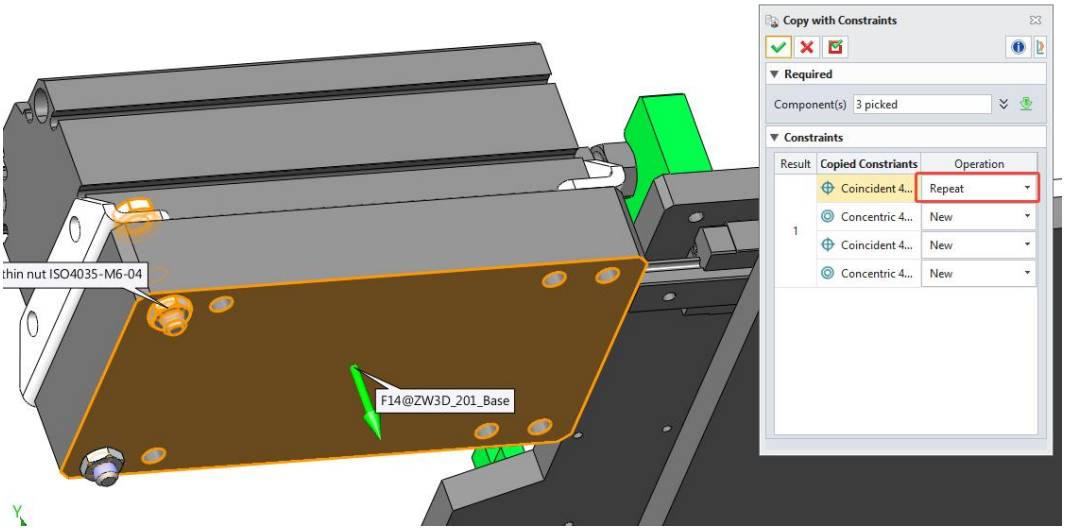
**【Example】**

1) Select bolts, washers, and nuts at one time, then execute the "Copy with Constraints" command via the right-click menu. Select the position of the constrained object where you want to copy them, and you can batch generate the fastener assembly.






2) When the target constraint object matches the original constraint, quickly reuse the original object by pressing the mouse middle button.




**【Where is it】**

Assembly Environment >> Assembly Manager >> **Right click menu**

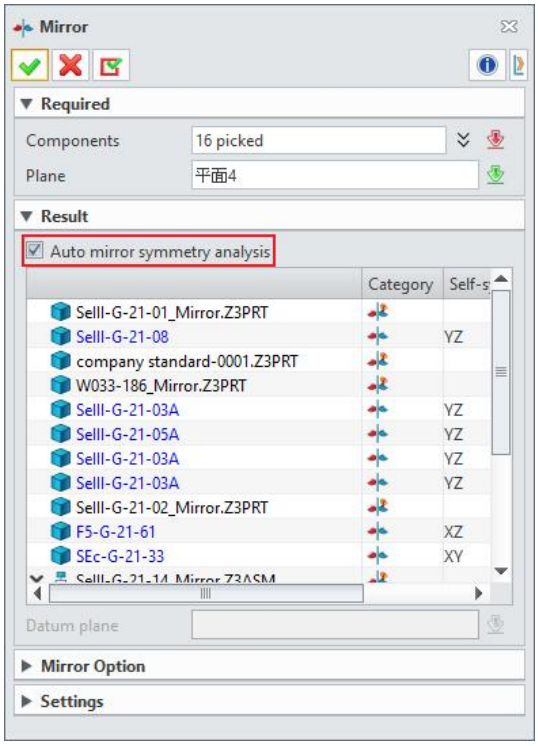
### 3.3.3 ★Assembly Mirror Optimization

Assembly Mirror is a commonly used design function, but due to the asymmetry of the mirror model, the direct result of mirroring often does not meet the requirements. Adjustments to the self-symmetry plane or mirroring method are usually necessary, which leads to lower efficiency.

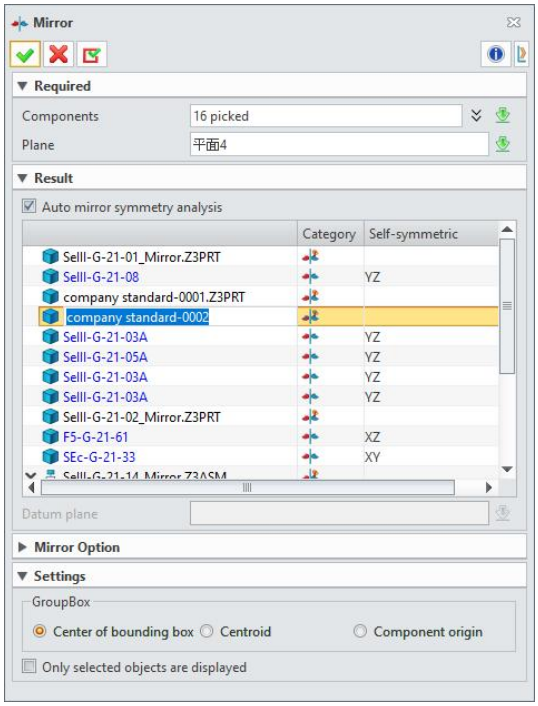
The new "Auto mirroring symmetry analysis" function allows the system to automatically perform symmetry analysis and select the appropriate mirroring method. In addition, other functions of assembly mirroring have also been enhanced in this version. The overall capability and ease of use of assembly mirroring have been significantly improved.

**【What users can do】**

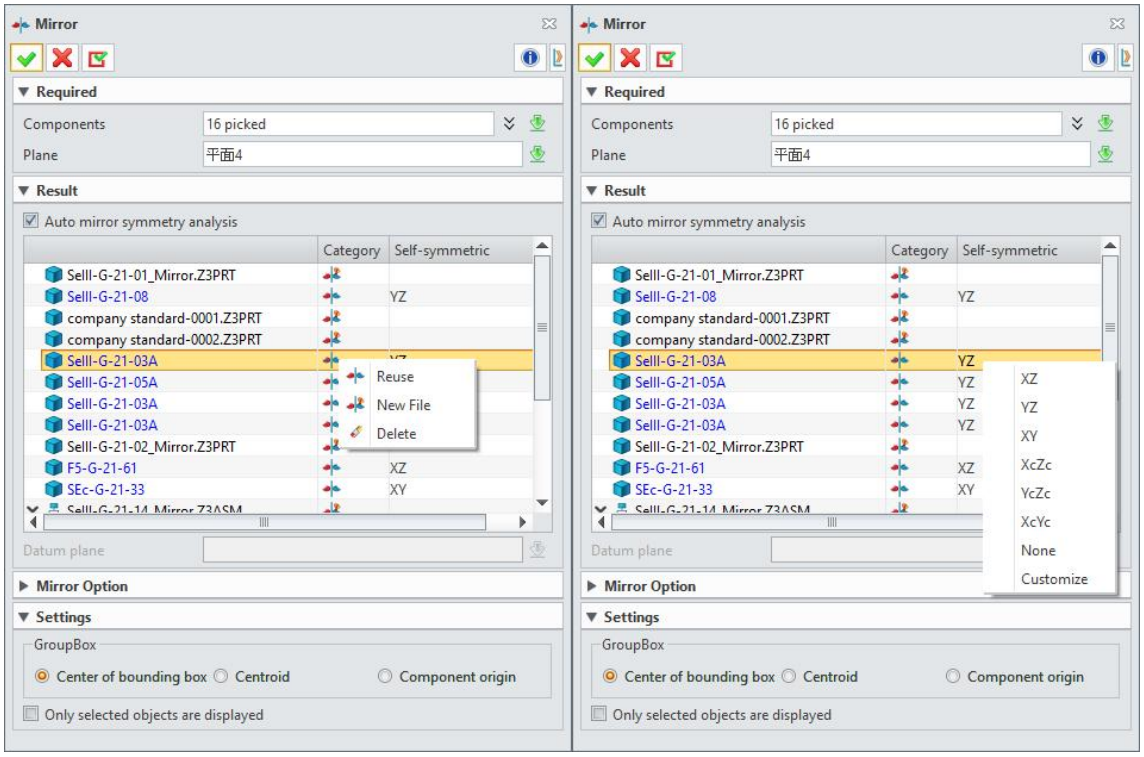
✓ It supports automatic symmetry analysis. Mirror can automatically provide the optimal mirror self - symmetry plane or mirroring method.




✓ You can change the name of a newly created mirroring file.

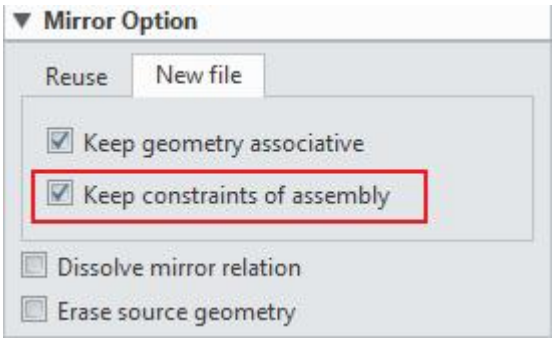


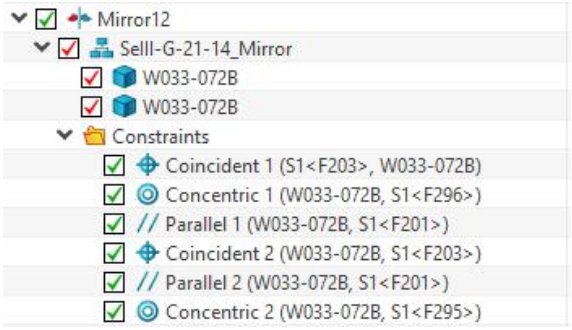

✓ You can adjust the mirroring mode of an mirroring file independently.




✓ After checking the "keep constraints of Assembly" option, it supports retaining the assembly

constraints of the source file for the newly generated mirrored file.



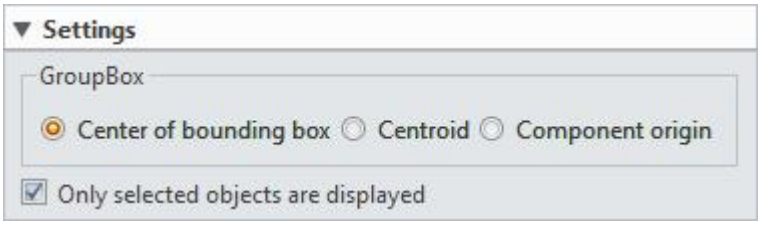

| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
| |  |

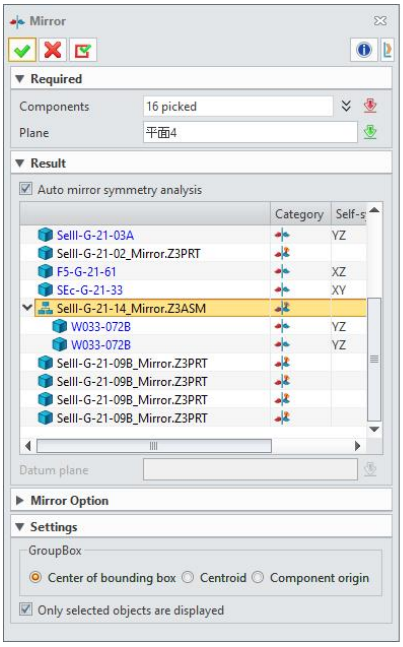




✓ User attributes and expressions are reserved for mirroring.

✓ After you select [Only selected objects are displayed] in Settings, you can view the mirroring preview result of an object separately.





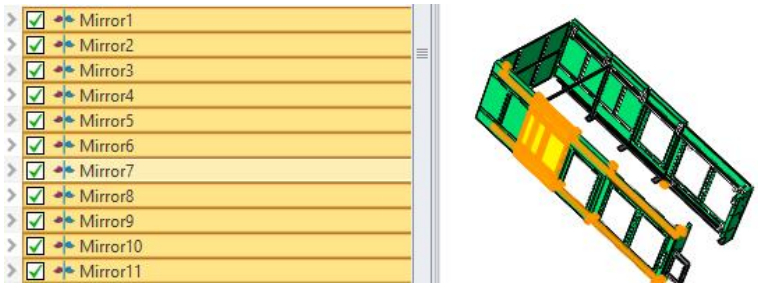
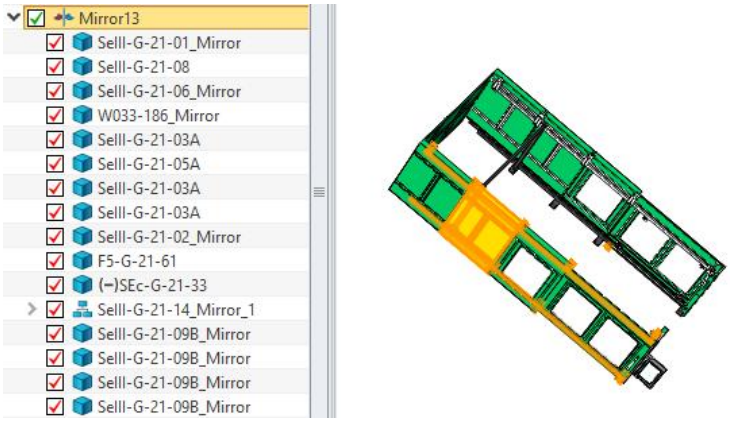
**【Notes】**

✓ If automatic mirror symmetry analysis is enabled, the total duration of the assembly mirror command increases.

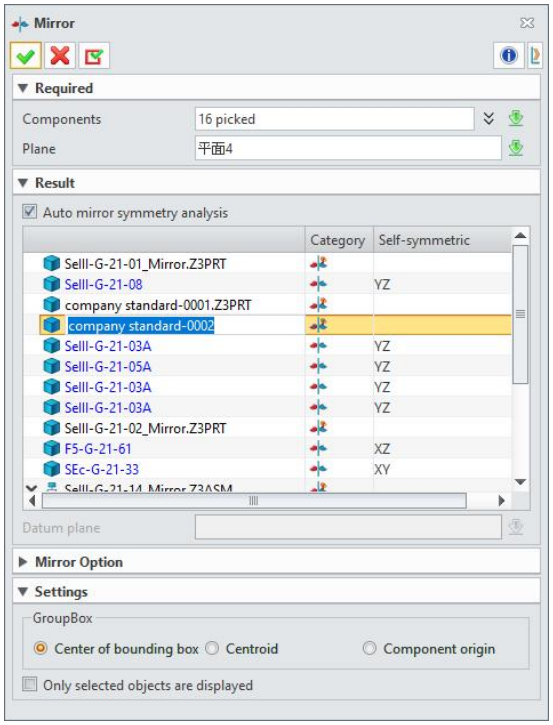
✓ When assembly constraints are inherited, if the assembly constraints in the file conflict with the symmetrical relationship, these conflicting assembly constraints will not be inherited.

**【Example】**

1) In the design of injection molding machine door components, the left and right baffles belong to the mirroring assembly relationship, but because of the difference in the symmetry of each part, the mirroring mode of different components needs to be set separately. In previous versions, you had to use multiple mirror operations, each set up separately. This can now be done directly with a single mirror operation.

| | |
|---|---|
| **ZW3D 2025** |  |
| **ZW3D 2026** |  |

2) In the assembly mirror command, you can double click on the file name of the newly generated mirrored file to rename it.



【Where is it】

Assembly >> Basic Editing >> **Mirror**

## 3.3.4 Assembly Pattern

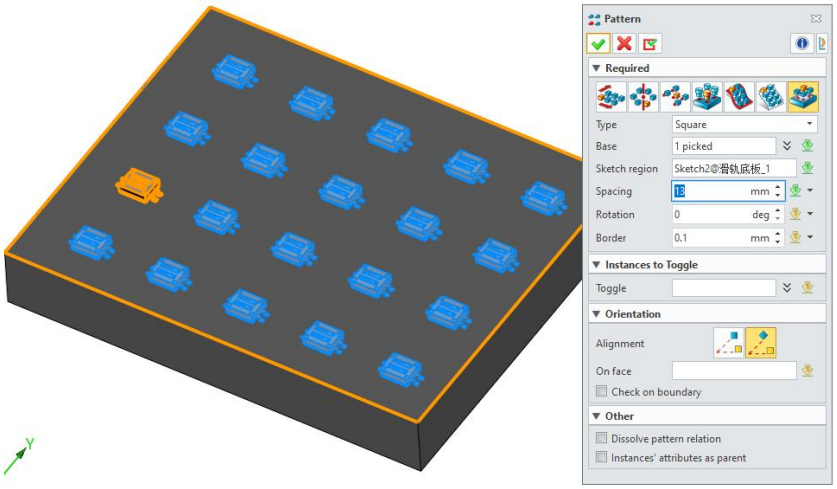
The assembly pattern has added a new fill pattern type. It supports generating point-to-point pattern based on reference components in the environment, thereby expanding the use cases for assembly pattern.

**【What users can do】**

- ✓ Create a fill pattern of components by selecting a sketch area.
- ✓ Implement assembly pattern using point-to-point pattern created within reference components.
- ✓ Use pattern/mirror features as input for pattern creation.
- ✓ Double-click on the assembly tree to redefine the pattern/mirror feature.

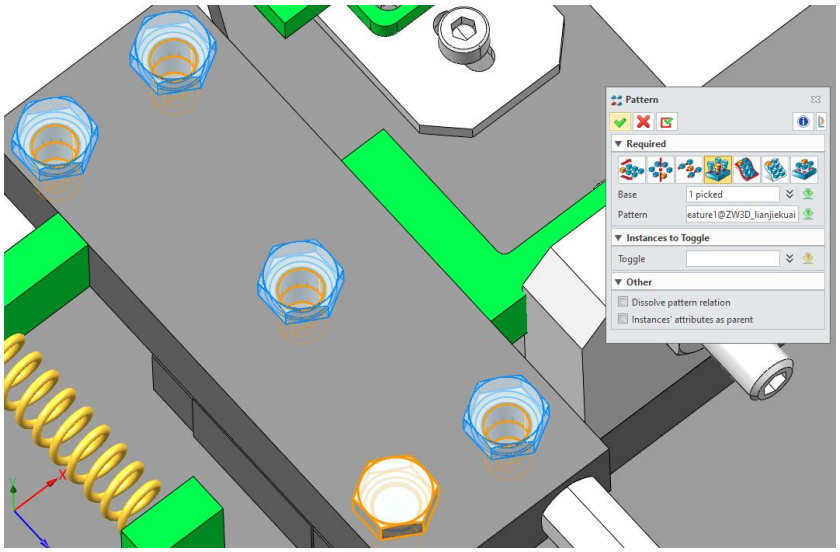
**【Example】**

1) Start the pattern command, select the fill array type, input the components to be arrayed and the target sketch area to fill, and automatically generate the result of the fill pattern.

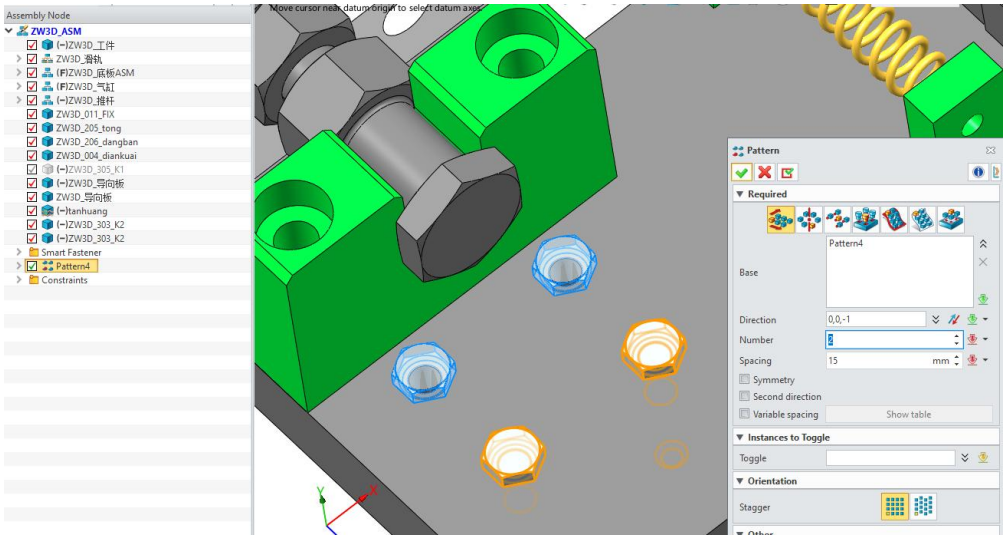



2) Create a point-to-point pattern for assembly holes in a part, and when installing bolts in the assembly, use the pattern created from the reference component to generate the installation

positions for the point-to-point bolts.




3) When generating a pattern from existing pattern/mirror features, the selected source components of the pattern will be automatically input to create a new array.




## 【Where is it】

Assembly Environment >> **Pattern**

Assembly Environment >> **Assembly Manager**

## 3.3.5 BOM Function Optimization

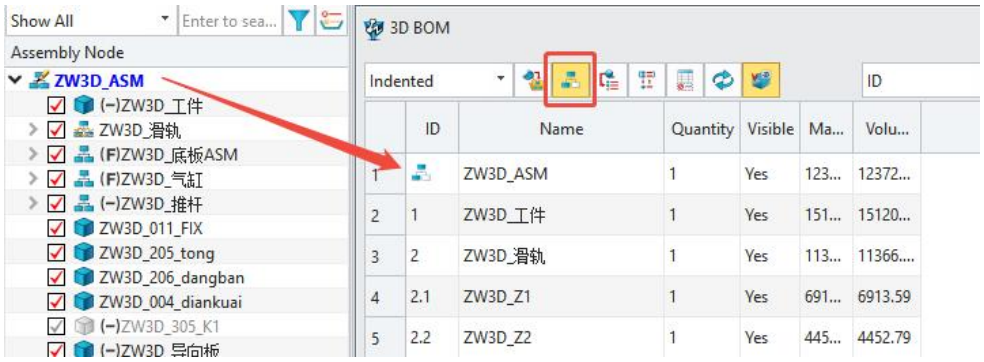
The 3D BOM for assembly expands the scenarios of use, supporting the modification of properties for both the main assembly and its subordinate components directly within the 3D BOM.

**【What users can do】**

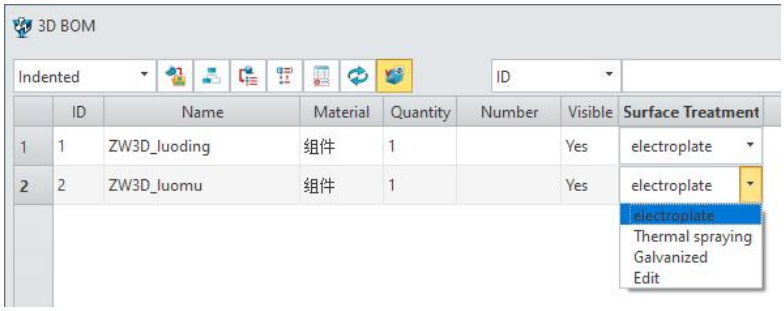
- ✓ Modify the top assembly properties within the 3D BOM.
- ✓ Update calculated properties to synchronize the total assembly mass.
- ✓ Modify components preset properties via dropdown options in the 3D BOM.
- ✓ Open files by selecting them in the 3D BOM.
- ✓ Set whether components are displayed in the BOM from the assembly tree.

**【Example】**

1) In the 3D BOM, by opening the "Include Top Assembly" control, you can simultaneously modify the properties of both the top assembly and its components.




2) Pre-set the content of dropdown properties and perform property switching using the dropdown options within the 3D BOM.



3) In a multi-column tree, select display the "Listed in Assembly BOM Table" property for a specific component. Setting "Do not List in BOM" for that component will ensure it does not list in the 3D BOM within the current assembly.

**【Where is it】**

Assembly Environment >> **3D BOM**

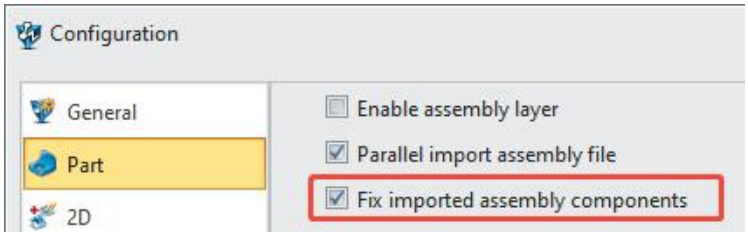
Assembly Environment >> **Assembly Manager**

## 3.3.6 Assembly Import Optimization

In an assembly design, it is usually necessary to refer to other assembly files to complete the design, and the referenced assembly does not need to have a change in position. It will import assembly files in other formats in a fixed way by default.

**【What users can do】**

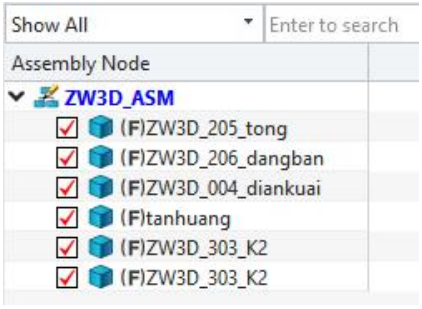
✓ Imported assembly files in other formats default to fixing all components.




**【Example】**

1) Drag and drop directly from the Windows folder to open the STEP assembly file that needs to be imported into ZW3D.

2) All components in the opened assembly file are fixed by default.




**【Where is it】**

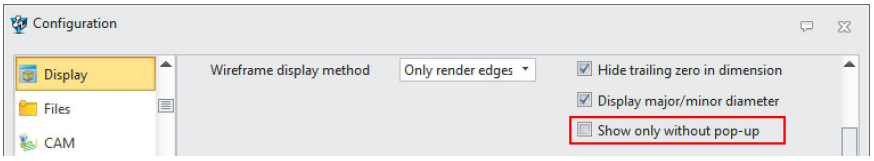
Configuration >> Part >> **General**

## 3.3.7 Show Only Optimization

The show only feature allows users to retain the show only settings when switching to other environments, thus maintaining the show only status for subsequent operations.

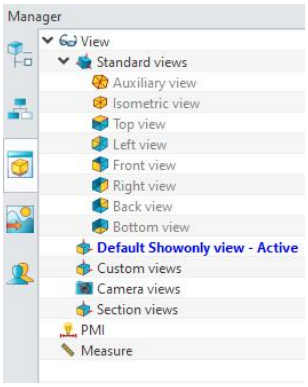
**【What users can do】**

- ✓ Switching editing objects retains the show only status.
- ✓ Set whether to show a popup for show only in the configuration settings.



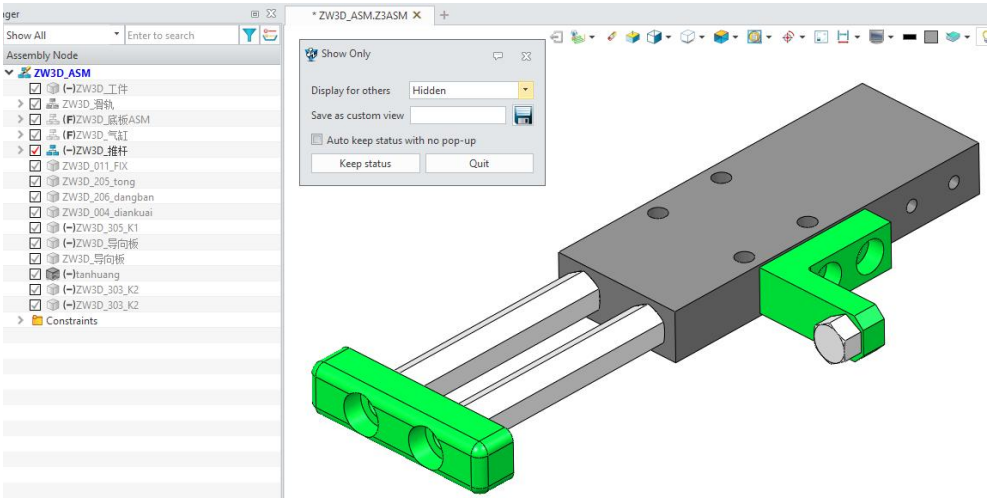

**【Notes】**

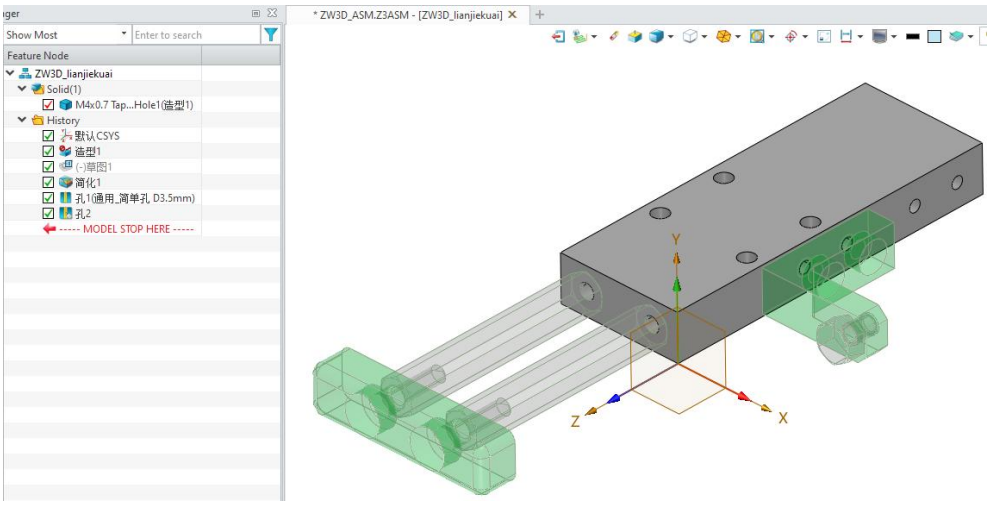
- ✓ You can switch back to the non-isolated display status in the view management.



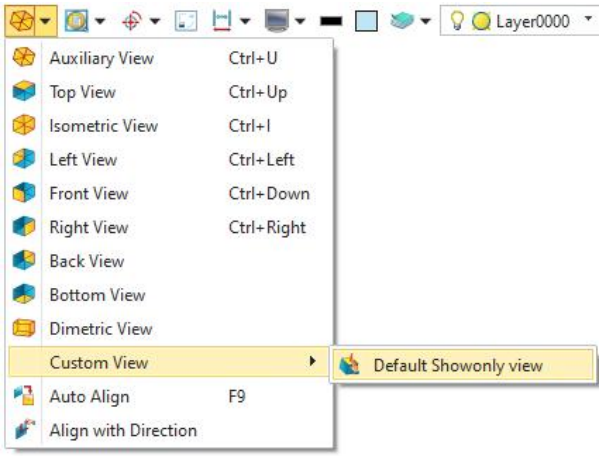

**【Example】**

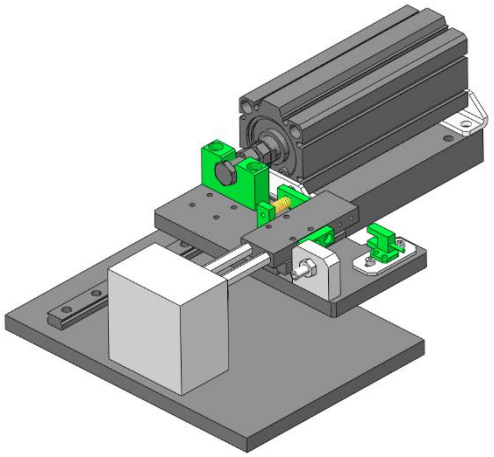
1) By selecting multiple components and right-clicking to choose to show only, you can edit the sub-components of the assembly while maintaining the original show only settings.



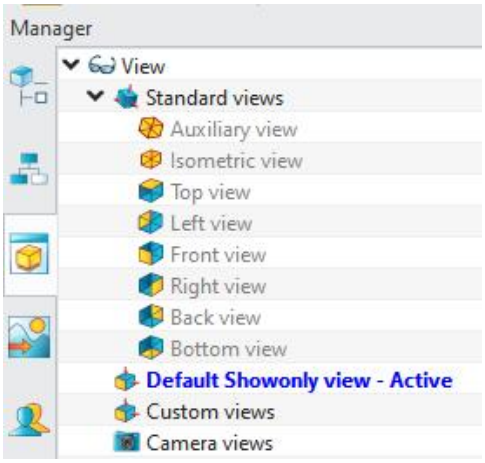



2) After returning to the originally show only of the assembly hierarchy, switch to the default view that only displays the view in the DA toolbar to restore the display state prior to the isolated view.



3) Double clicking in the View Manager **Default Showonly view**, which can also restore the original isolated display state.




## 【Where is it】

Assembly Environment >> Assembly Manager >> **Right click menu**

Configuration >> Display >> **Toggle settings**

Parts/Assembly Environment >> Drawing Area >> **DA Toolbar**

Parts/Assembly Environment >> **View Manager**

## 3.3.8 Assembly Cut Optimization

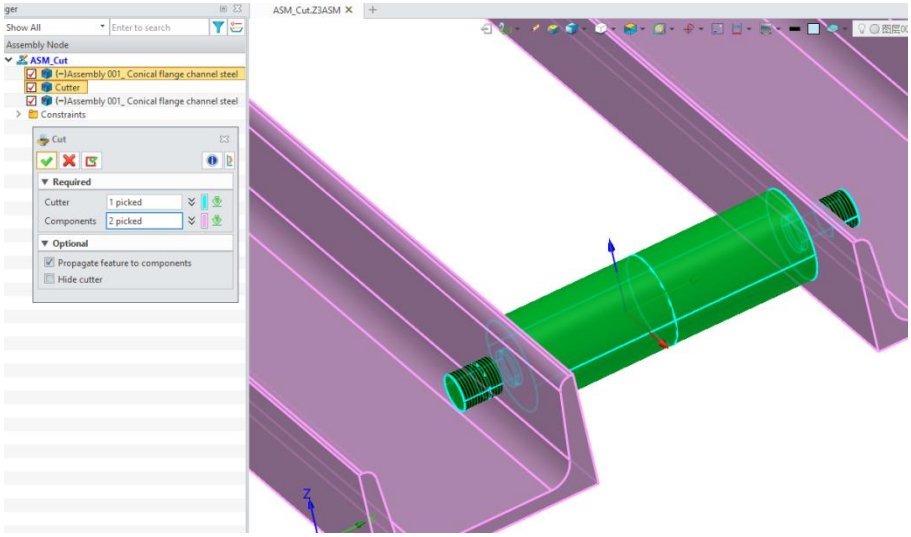
In assembly design, the same structural part may be installed in multiple locations. For practical production needs, one cut part can be used to perform a single cut on a structural part at multiple positions. It allows for a single part to be cut at different component locations all at once.

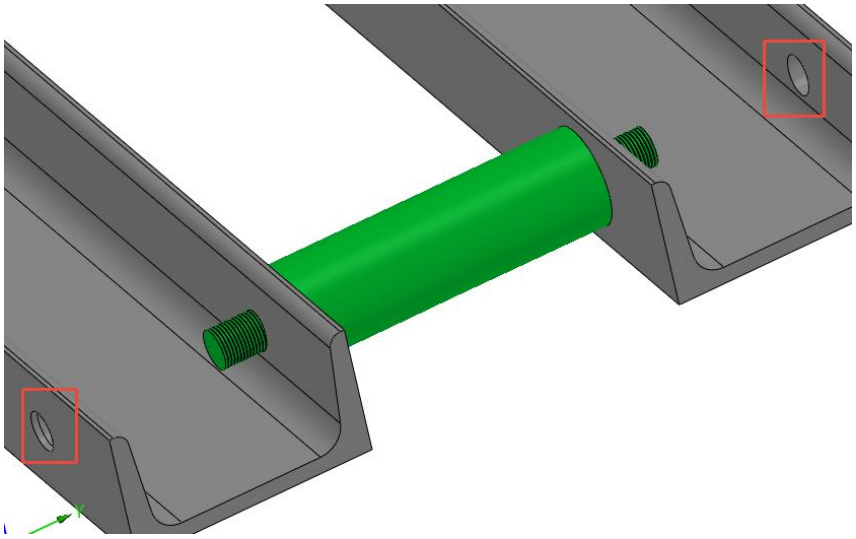
**【What users can do】**

✓ A single command operation can simultaneously cut different component instances of the same part and propagate the cutting feature into the part's historical features.

**【Example】**

When cutting multiple instances of the same part in an assembly, selecting the components to propagate this feature allows for the creation of cut results at different locations on the part.





【Where is it】

Assembly Environment >> **Cut**

## 3.3.9 ★Smart Fastener Optimization

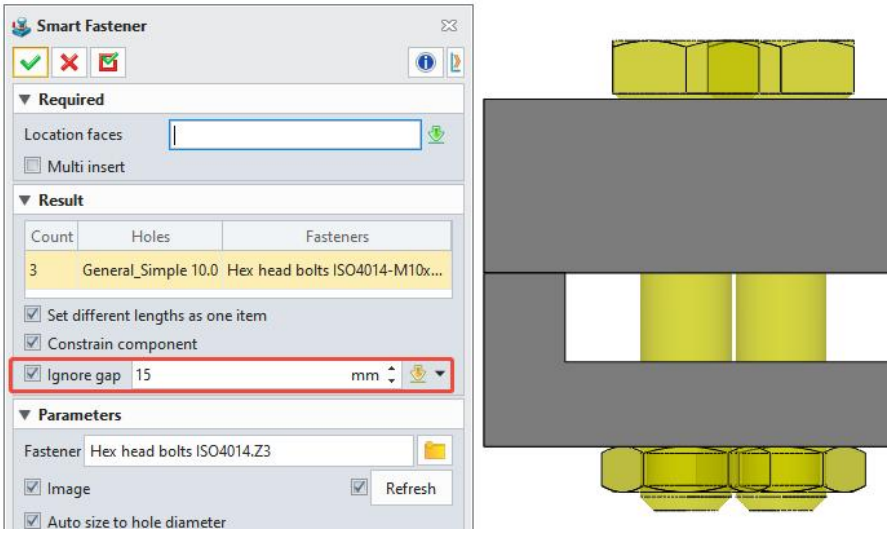
In assembly design, there are often intermittent installation scenarios where fasteners need to be installed on two locating plates that have a gap between them. The smart fastener can recognize the installation gap and correctly install the nut at the other end of the gap.

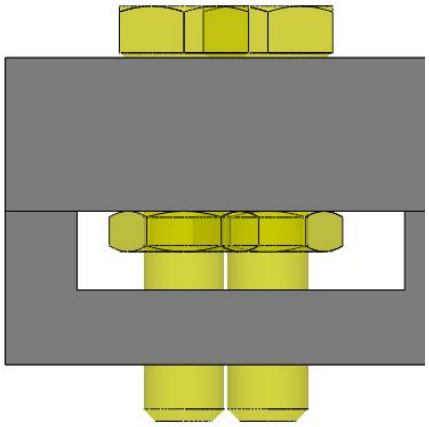
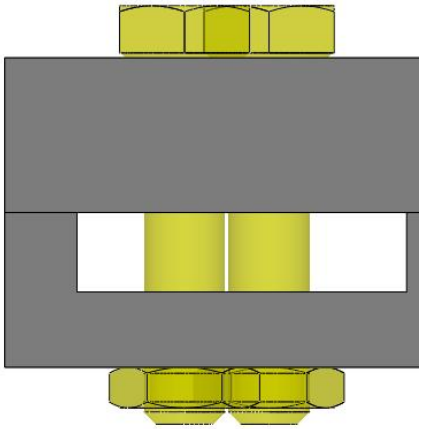
**【What users can do】**

✓ Add mounting positions for nuts while ignoring the intermediate layers based on specified gaps.

**【Example】**

When adding fasteners to two mounting plates with gaps, the system can intelligently determine bolt and nut placements at both ends while ignoring gaps within the set value.




| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|  |  |

**【Where is it】**

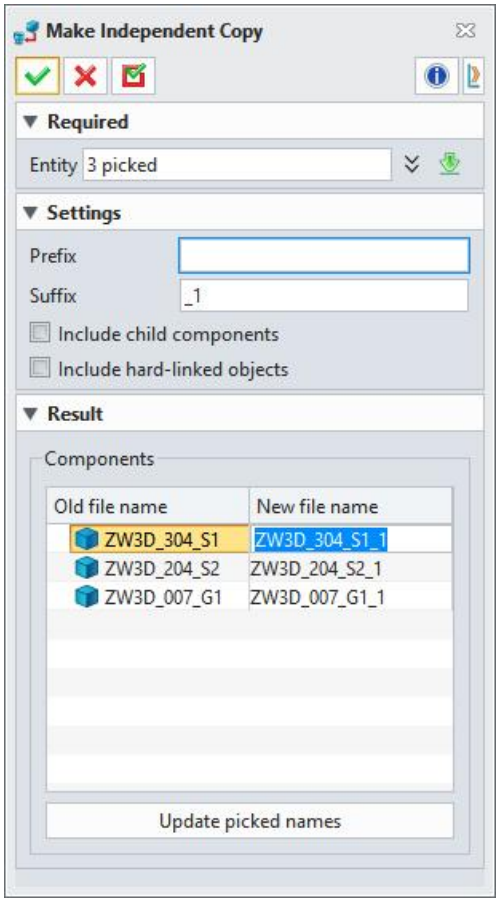
Assembly Environment >> **Smart Fastener**

## 3.3.10 Make Independent Copy Optimization

Organizations often have personalized naming standards for parts and assemblies. Make Independent Copy allows for direct renaming of new files, ensuring that the newly generated files also comply with the company's naming standards.

**【What users can do】**

✓ Generating an independent copy supports renaming.




**【Where is it】**

Assembly Environment >> Assembly Manager >> **Right click menu**

## 3.3.11 Assembly Tree Interaction Optimization

In assembly design, interactions between different levels are common. The new version has streamlined the convenience of interactions among multiple levels within an assembly, enhancing the flexibility of assembly design.

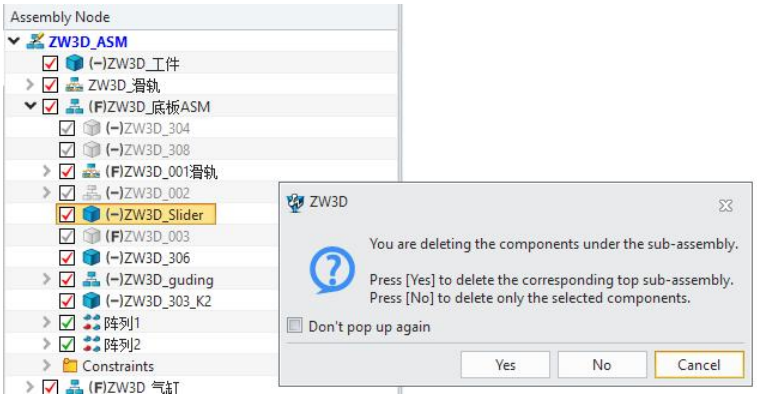
**【What users can do】**

✓ To delete lower level components, you can choose to delete the corresponding top-level assembly.

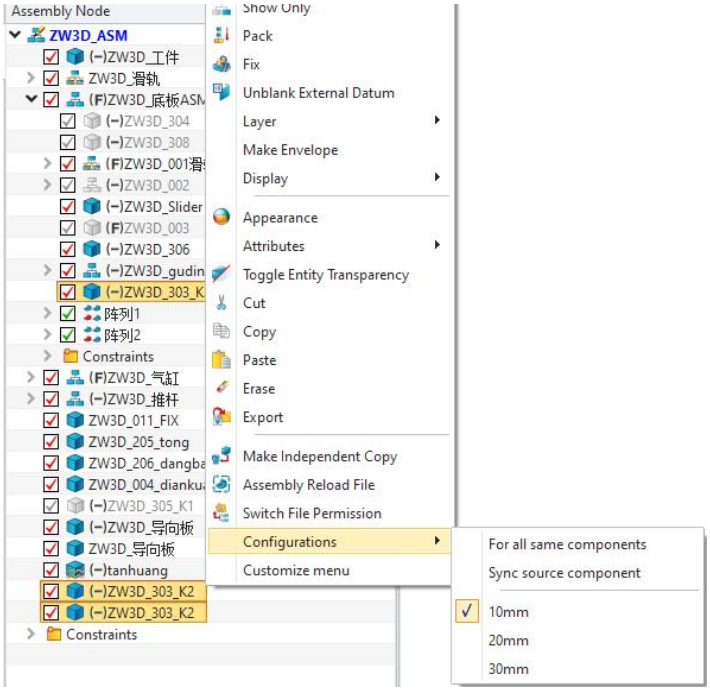
✓ Support batch switching of configurations for the same components.

**【Example】**

1) Select the components within the sub assembly on the assembly tree, delete the components, and a prompt will pop up asking whether to delete the top-level assembly or only the selected components.




2) Multiple selection of identical parts at different levels and switching of component configurations can achieve batch configuration switching of selected components.



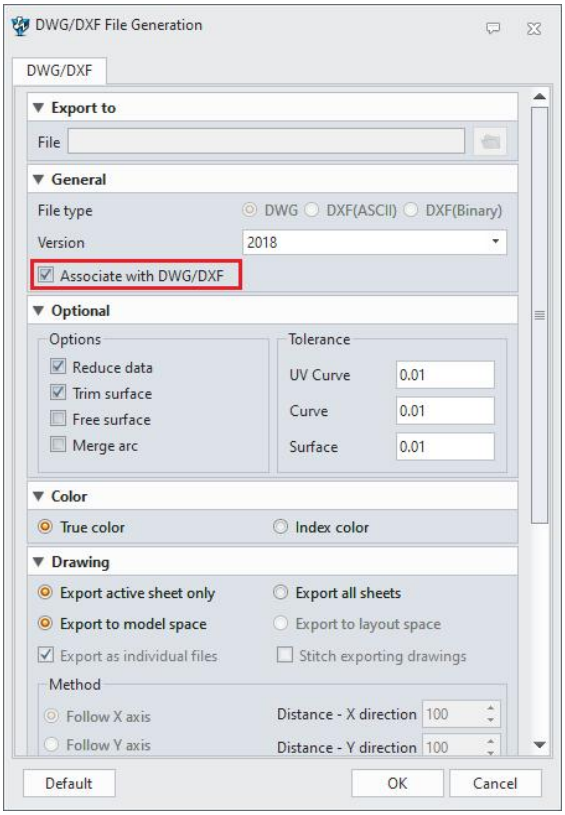
【Where is it】

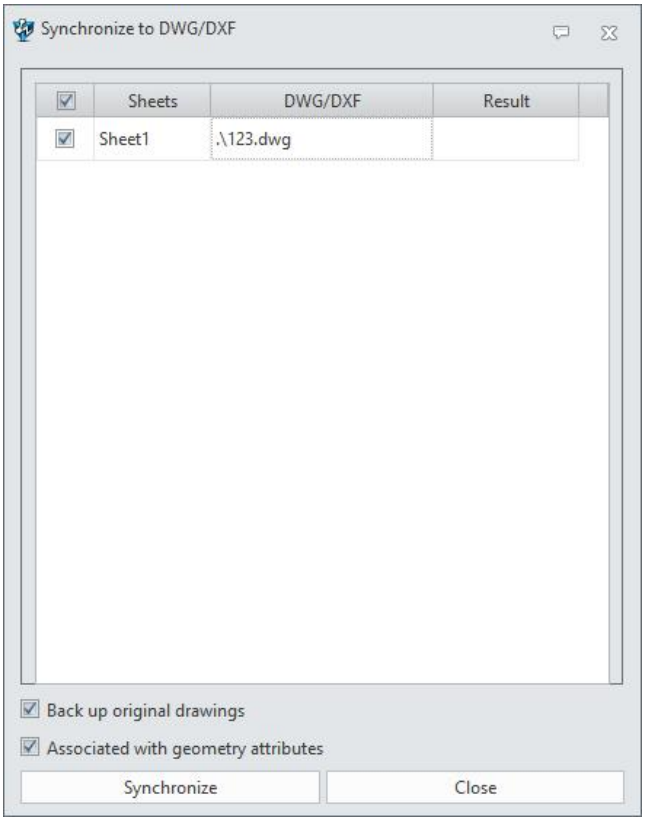
Assembly Environment >> Assembly Manager >> **Right click menu**

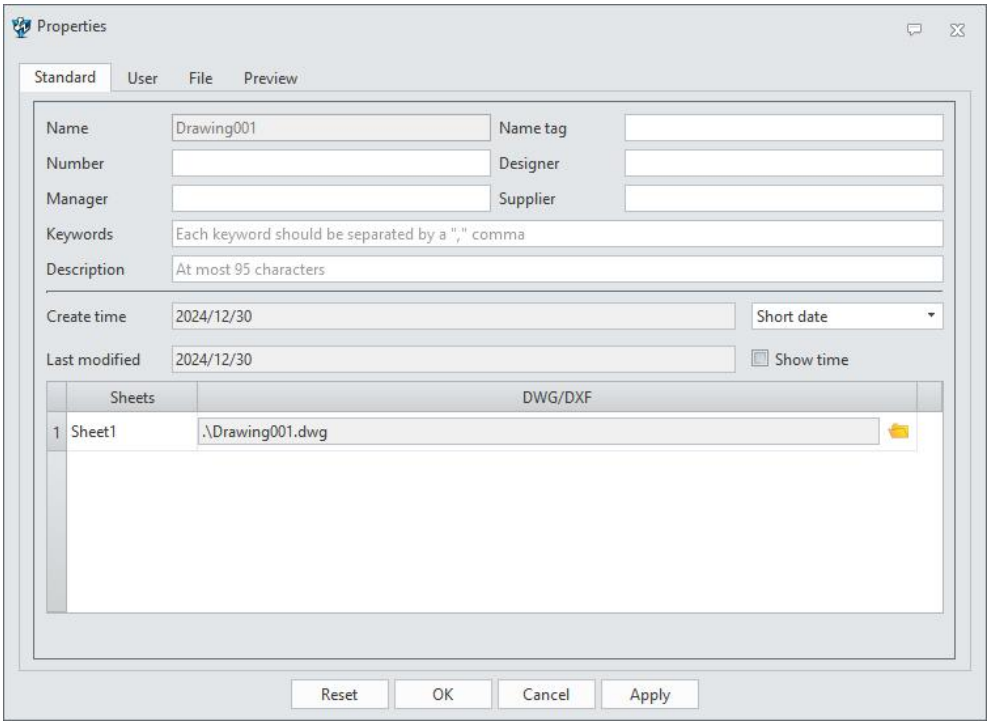
## 3.4 Drawing Sheet

### 3.4.1 ★ 2D/3D Support for Association Updates

In the drawing environment, ZWCAD Annotation allows you to convert 3D DRW drawings directly to DWG/DXF format and open them in ZWCAD. During this conversion process, it supports the establishment of association between DRW drawing files and DWG/DXF files. When the 3D DRW drawing is changed, through the new function "Synchronize to DWG/DXF", the DWG/DXF file associated with it can be automatically updated, without the need to manually redraw or modify the 2D drawing, ensuring the consistency of the 2D and 3D data and greatly saving the design time.






**【What users can do】**

✓ Establish association between the drawing and DWG/DXF exported.

✓ Keep the associated unbroken unless the relative path between the drawing and DWG/DXF changes.

✓ Establish association between the drawing and DWG/DXF existed manually in the property page of drawing or attributes page of sheet.

✓ Synchronize the modification of drawing to the DWG/DXF associated.

✓ Finish the process of synchronization when the DWG/DXF associated is opened by ZWCAD or ZWCADM.

✓ Associate several sheets of one drawing with one DWG/DXF file.

✓ Retain the change of attributes towards views and other curves in the DWG/DXF associated.

**【Notes】**

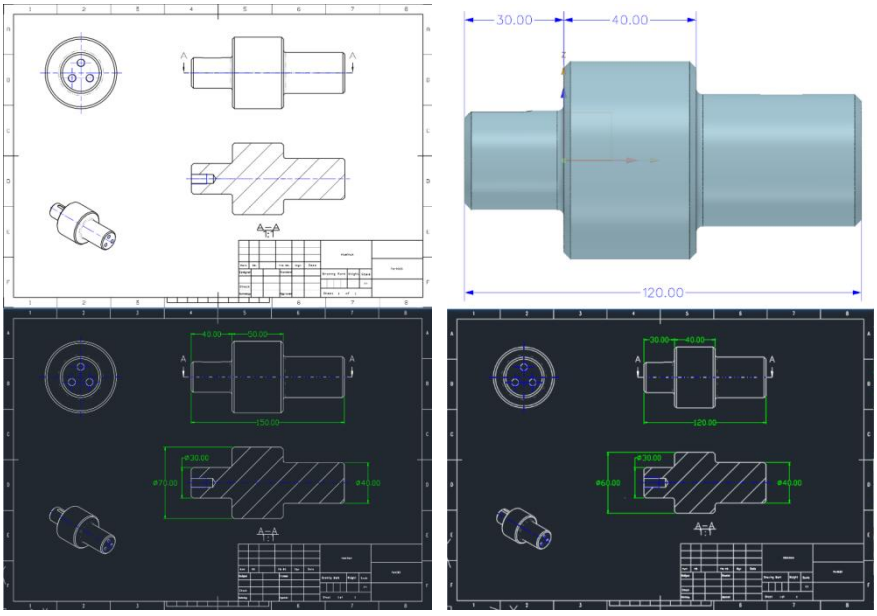
- DWG/DXF exported does not support to be associated with more than one drawing.

- The association cannot be established if views, hatches and texts are exported as one single line.
- The association cannot be established if the drawing is exported to layout space of DWG/DXF.
- The drawing has to be saved before it is exported and associated with a DWG/DXF file.
- The association will be broken when the relative path between drawing and DWG/DXF file changes.
- The process of synchronization could be completed in just one direction currently.

**【Example】**

1) By the function “Dimension in ZWCAD”, the current drawing can be exported as DWG/DXF file. Meanwhile, through checking the option "Associate with DWG/DXF" in the page "DWG/DXF File Generation", the association between two files will be established.

2) Modify the 3D model associated after finishing dimensioning in the DWG/DXF file. Through the new function "Synchronize to DWG/DXF", the modification of the 3D model will be synchronized to the DWG/DXF file associated.




**【Where is it】**

Drawing Sheet Environment >> Dimension >> Collaboration >> **Dimension in ZWCAD**

## 3.4.2 ★ Optimization of Projection Efficiency

The projection efficiency, drawing sheet regeneration efficiency, and file size are optimized in the new version.

**【What users can do】**

✓ Without opening the hidden line and centerline, the projection efficiency of discrete projection is increased by about 60%, the regeneration efficiency of marked discrete projection drawing sheets is increased by about 80%; the geometric projection efficiency is increased by about 40%, and the regeneration efficiency of marked geometric projection drawing sheets is about 60%.

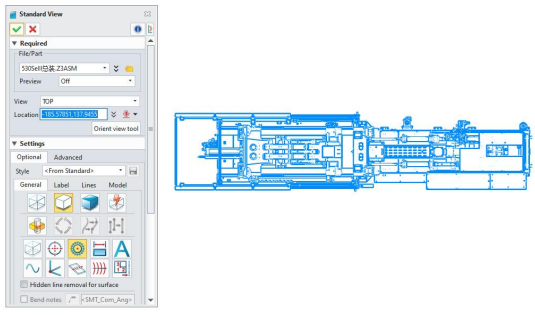
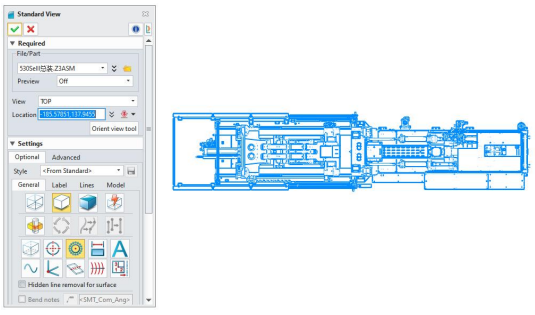
✓ The file size is reduced by about 85% without opening the hidden line and the center line.

✓ The efficiency of drawing sheet radio selection elements is increased by about 90%, and the efficiency of box selection elements is increased by about 70%.

✓ The lines of the discrete projection view support setting layers, and layer settings take effect in high-quality views.

**【Example】**

Taking the projection efficiency of the injection molding machine model as an example, the discrete projection time has been reduced from about 7s to about 2s, and the geometric projection has been reduced from about 33s to about 15s.

| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|  |  |

**【Where is it】**

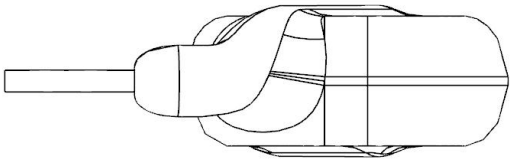
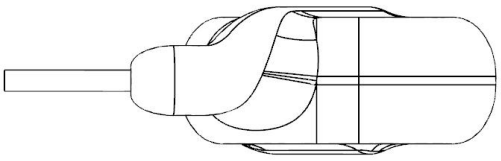
✓ Drawing > > Standard view

## 3.4.3 View Optimization

### 3.4.3.1 Optimization of Display Effect of Discrete Projection

In discrete projection mode, the high-quality edge's function can be enabled to improve the display accuracy of the discrete views.

**【What users can do】**

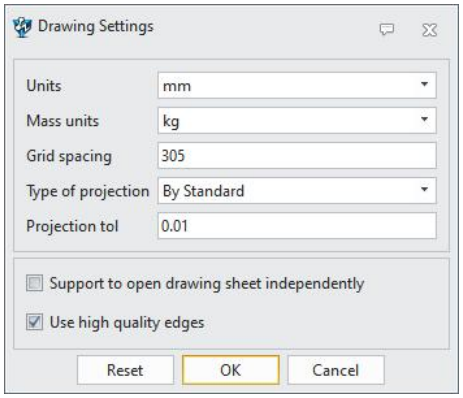
| Don't use high quality edges | Use high quality edges. |
| --- | --- |
|  |  |

**【Notes】**

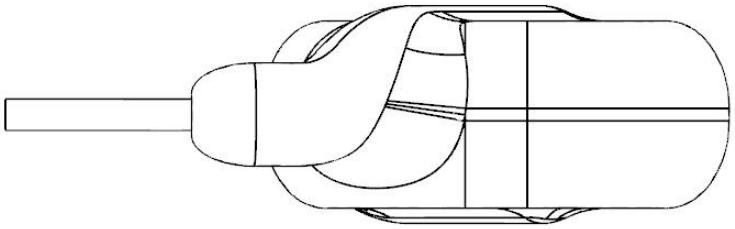
- It can only improve the display effect of discrete view under discrete projection.
- When this function is turned on, the projection efficiency, the drawing regeneration efficiency and the file opening efficiency will be reduced.

**【Example】**

1) Turn on the use of high-quality edges in the environmental parameter setting of drawing sheets.




2) The edges generated by projection will be smoother.

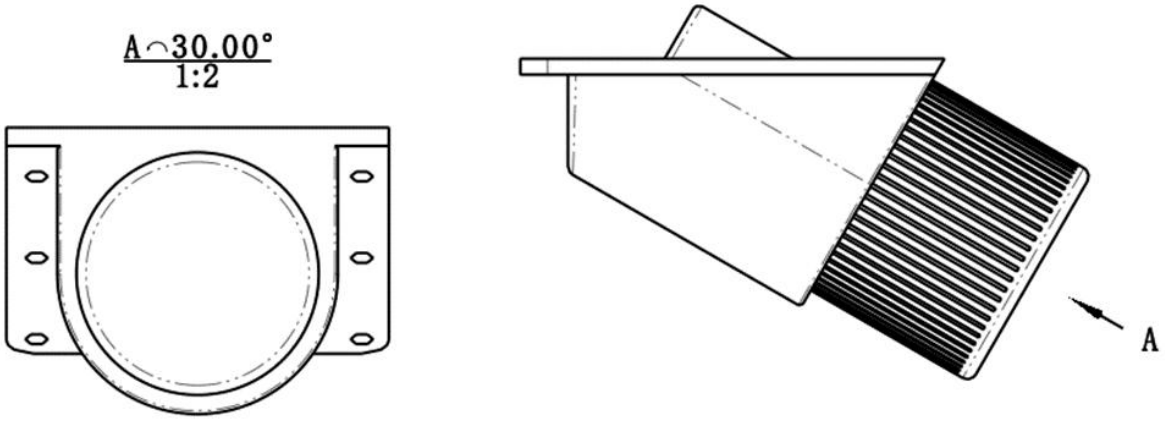


【Where is it】

Drawing > > Tools > > Parameter Settings > > **Use High Quality Edges**

## 3.4.3.2 ★ View Rotation Optimization

The rotation angle function can realize that the projection view, auxiliary view, and cross-sectional view rotate around the drawing plane while maintaining the association with the parent view and automatically generating the view rotation label. It also supports automatic clockwise rotation of the view settings to realize the automatic alignment of the view angle.



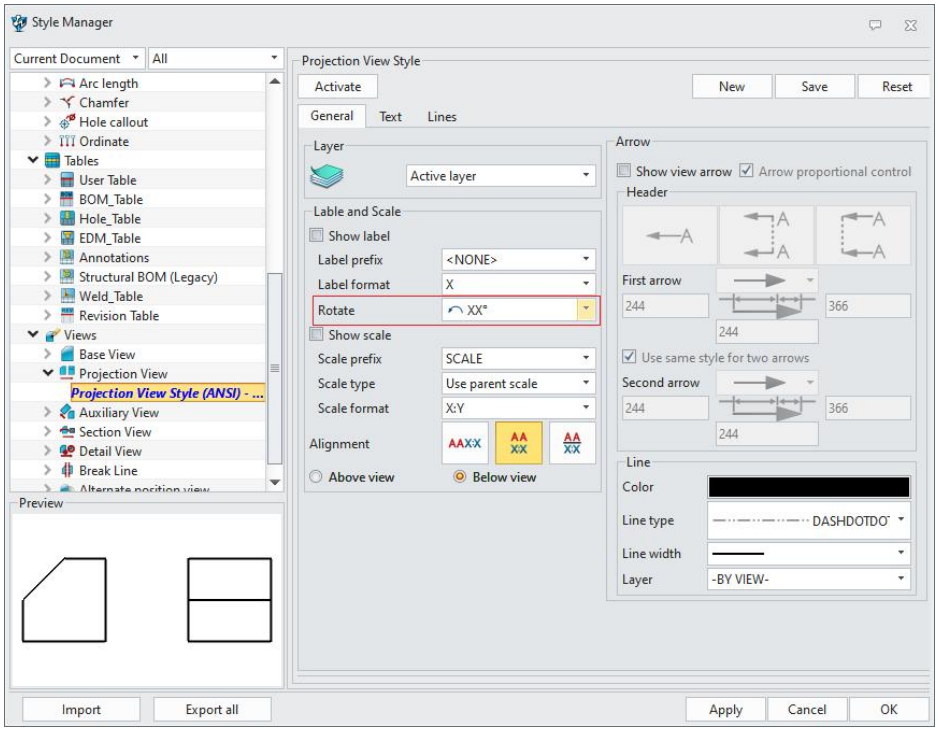

**【What users can do】**

✓ Support users to manually set the view rotation angle for projection view, auxiliary view and section view, and automatically generate the rotation angle label.

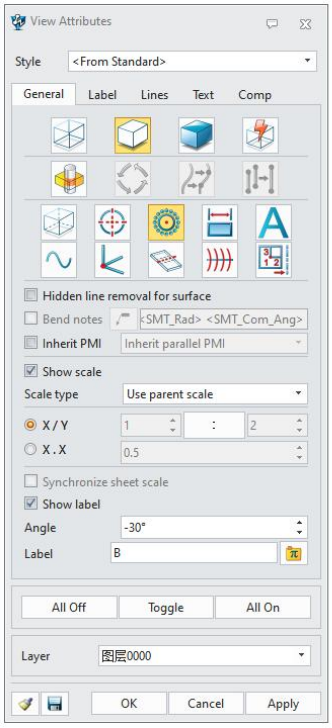
✓ Support the setting of clockwise and counterclockwise rotation for auxiliary views and sectional views and realize automatic view alignment.

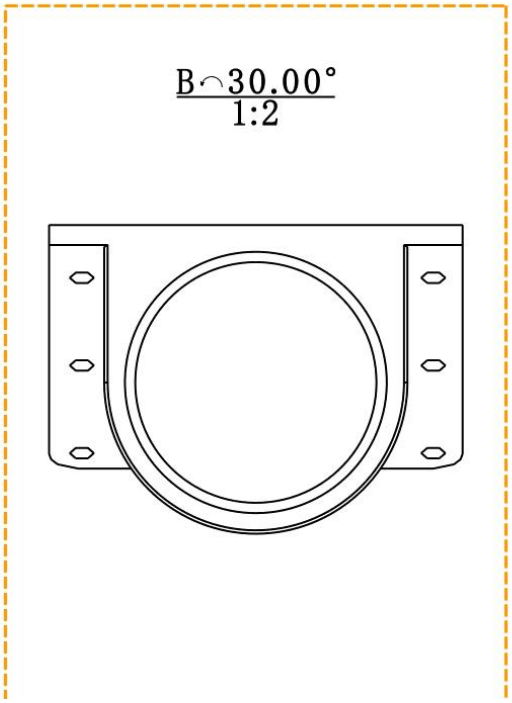
**【Example】**

1) Set the style of the view rotation label in the Style Manager.

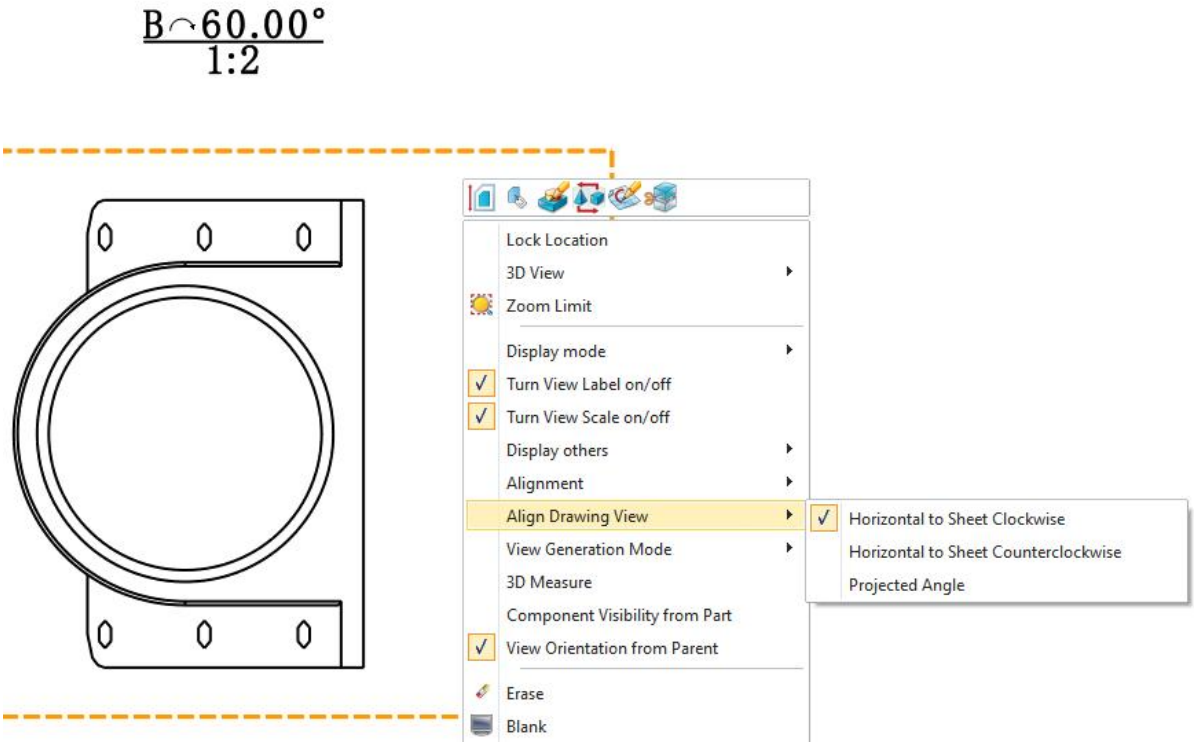



2) Change the view rotation angle, straighten the position of the view, and automatically generate the view rotation angle label.



3) Setting【 Horizontal to Clockwise 】or【 Horizontal to Counterclockwise 】for the view can realize the automatic alignment of the view.




**【Where is it】**

Drawing > > View > > **View Properties**

## 3.4.3.3 Curved Section View Supporting Arc Curve Type.

The sketch drawing section line in the bent section view command supports the inclusion of arcs, and when the section method selects "sect curve", the arc segment will be processed in the unfolded way, and the section result will be flattened to the section direction.

**【What users can do】**

✓ Support the flattening of the cutting results generated by the arc cutting line to the cutting direction.

**【Notes】**

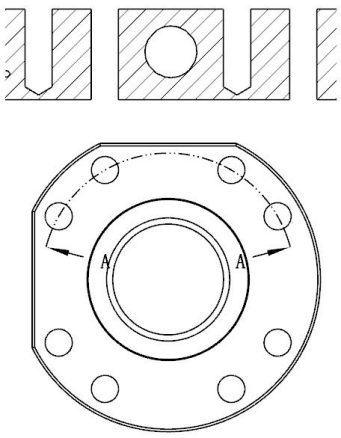
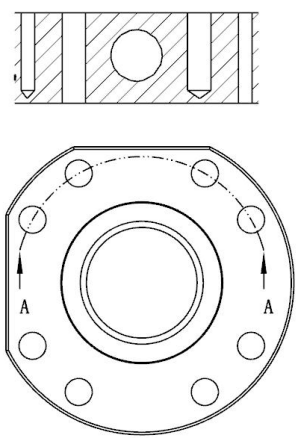
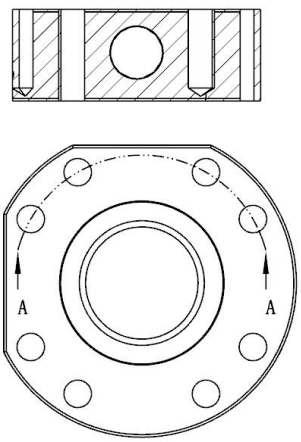
- The cutting line can be a pure arc line, or it can start or end with a straight line with an arc

segment in the middle.

- The cutting line is not closed and does not cross.
- When an arc segment is included, it is a legal arc when the arc angle is less than or equal to 180, and an illegal arc when the arc angle is greater than 180.

**【Example】**

Take the circular arc bent section of the bearing as an example, and the results are as follows.

| Profile curve | Trim parts | Trim surface |
|---|---|---|
|  |  |  |

**【Where is it】**

Drawing > > View > > Bent Section

## 3.4.3.4 Optimization of Label Movement in Detail View

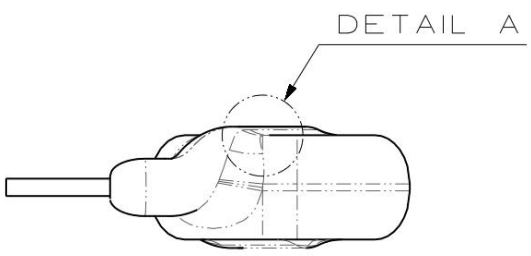
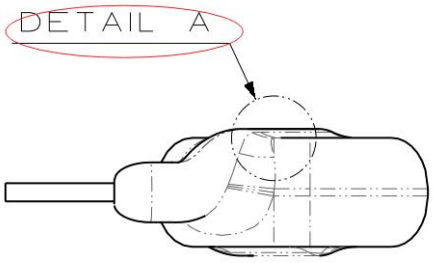
The label movement of the detail view has been optimized in the 2026 version, and the annotation text can be selected to realize the movement of the label.

**【What users can do】**

- ✓ Directly select the label text to realize the label movement of the detail view.

**【Example】**

Directly select the label text to realize the label movement of the detail view.

| Before moving | After moving |
| --- | --- |
|  |  |

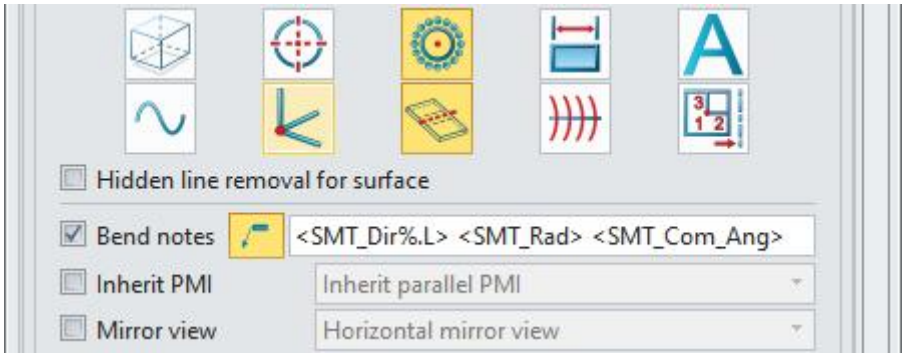


**【Where is it】**

Drawing > > View > > Detail view

## 3.4.3.5 Optimization of Sheet Metal Bending Tips

Sheet metal bending is optimized in version 2026, supporting the supplementary angle showing the bending angle, and adjusting the sheet metal bend notes field of the original configuration panel to the drawing sheet template file.




**【What users can do】**

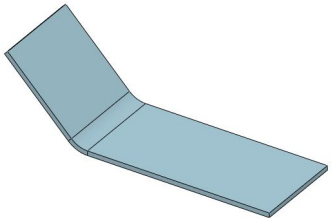
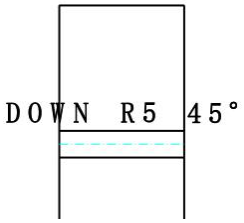
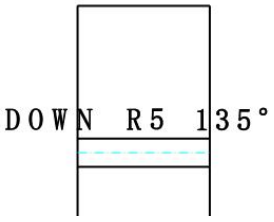
- ✓ Adjust the sheet metal bend notes to follow the drawing template.
- ✓ Support to display the bend angle as complementary angle or supplementary angle.

【Notes】

- The original "Sheet metal bend callout" was changed to "Bend notes".
- The original prompt "With leader" is changed to the button control, and the leader is created when the button is lit, but not when the button is not lit.
- The "Bend notes" input box keeps <SMT_Dir%. L> <SMT_Rad> <SMT_Com_Ang>, where the original <SMT_Ang> is subdivided into the following two angle types.
- <SMT_Com_Ang>: complementary angle-complementary angle
- <SMT_Sup_Ang>: supplementary angle-complementary angle
- Original Layout > > 2D > > Sheet metal bend callout Removal

【Example】

Take the bending angle of sheet metal as 45°, and project the flattened top view as an example, and the result is as follows.

| Case | Display complementary angle | Display complementary angle |
|---|---|---|
|  |  |  |

【Where is it】

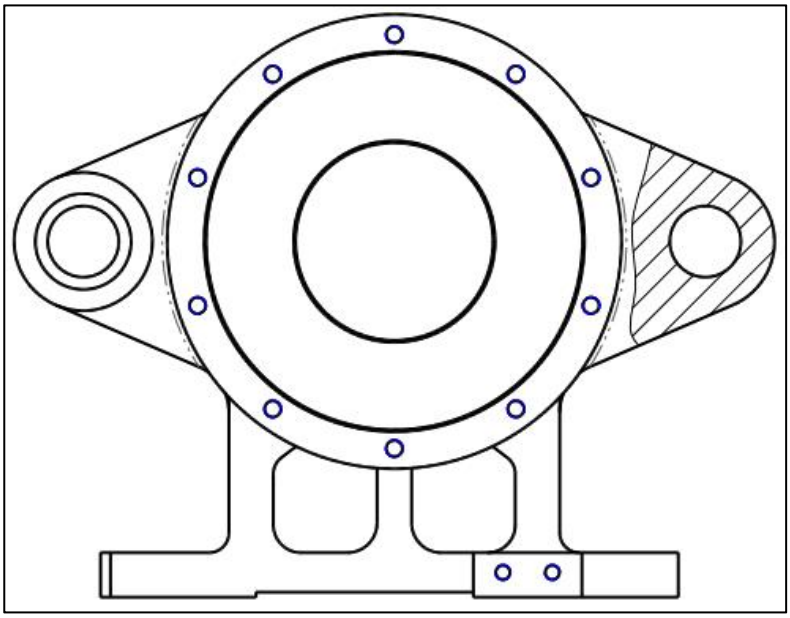
Drawing > > Tools > > Style Manager > > View > > **Bend notes**

## 3.4.3.6 New Spline Boundary to Broken Section View/Detail View

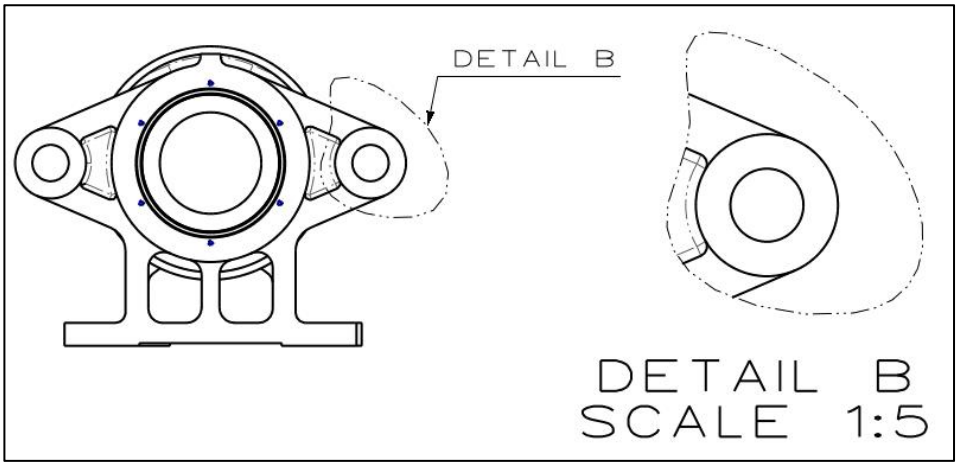
The boundary type of broken section view and detail view expands. Based on the original circle, rectangle, and polyline type, a new spline boundary type is added.

**【What users can do】**

- ✓ Create a broken section view using spline boundaries.



- ✓ Create a detail view using spline boundaries.



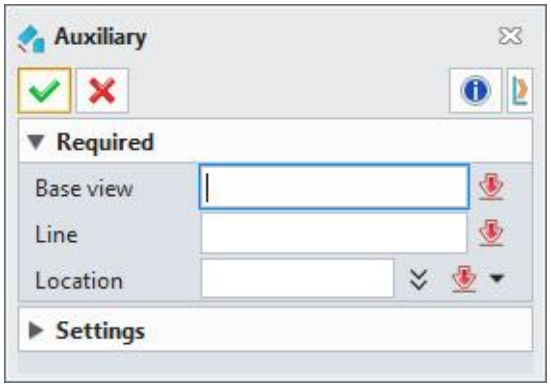

**【Where is it】**

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

Drawing Sheet Environment >> Layout >> **Detail View**

### 3.4.3.7 Optimized Auxiliary View

Auxiliary view has been optimized. You are able to choose center line as projection line.

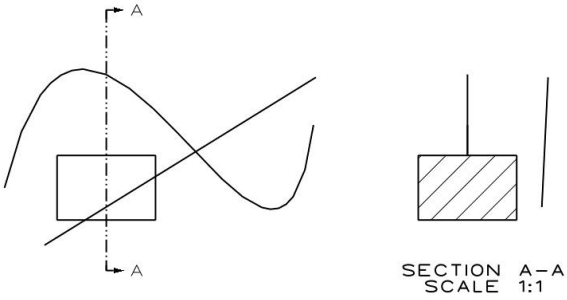



**【Where is it】**

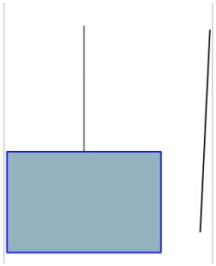
Drawing Sheet Environment >> View >> **Auxiliary**

### 3.4.3.8 Optimization of Section View（ZW3D 2026 SP1）

In drawing sheet, when you perform sectioning on views, you can actually section the projected 3D wires, such as 3D curves and objects in sketches and 3D sketches.




Sectional effect of the engineering drawing



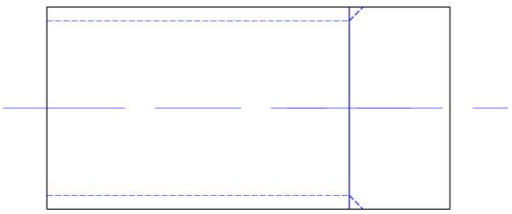
3D environmental profile effect

**【Where is it】**

Drawing Sheet >>Layout>>View>>**Full Section/Aligned Section /3D Named Section / Bent Section / Isometric Section**

### 3.4.3.9 External thread projection optimization（ZW3D2026 SP1）

When projecting the external thread, it can generate the standard termination lines.



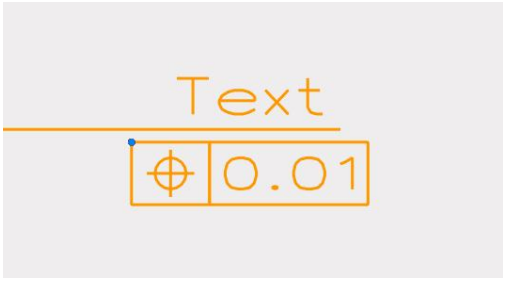
**【Where is it】**

Drawing Sheet >>Layout>>**View**

## 3.4.4 ★Annotation Optimization

### 3.4.4.1 Feature Control Optimization

Current attaching behavior of the geometric tolerance has been optimized. When you create or drag the geometric tolerance, you can attach it with label dimension. Later label and geometric tolerance can be dragged together or be unattached.

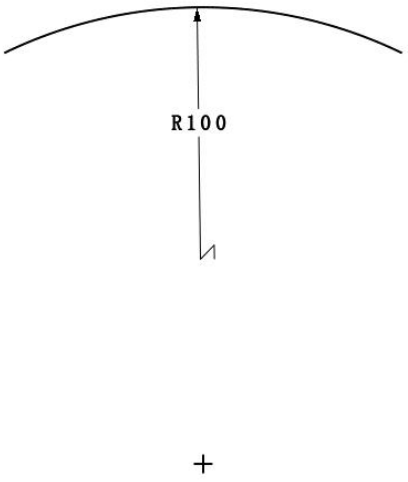
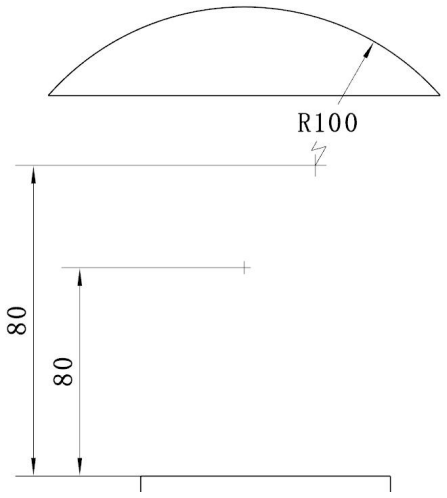



**【Where is it】**

Drawing Sheet Environment >> Dimension >> Annotation >> **Feature Control**

### 3.4.4.2 Broken Radial Dimension Optimization

The current broken radial dimension has been optimized. The center will be attached to the tail end, as you create a broken radial dimension. Later, the dimensions including this center and dimensions including the actual center will be the same.

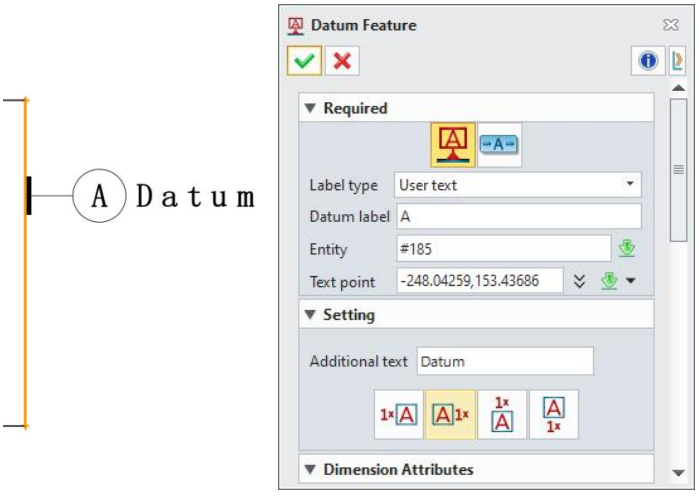
| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|   |   |

【Where is it】

Drawing Sheet Environment >> Dimension >> Dimension >> Radial/Diametric Dimension >> **Broken Radial Dimension**

### 3.4.4.3 New Additional Text of Datum Feature

Additional text functions have been added. When you create or edit datum feature, the additional text can be appended to the datum feature, and you can select the placement direction.

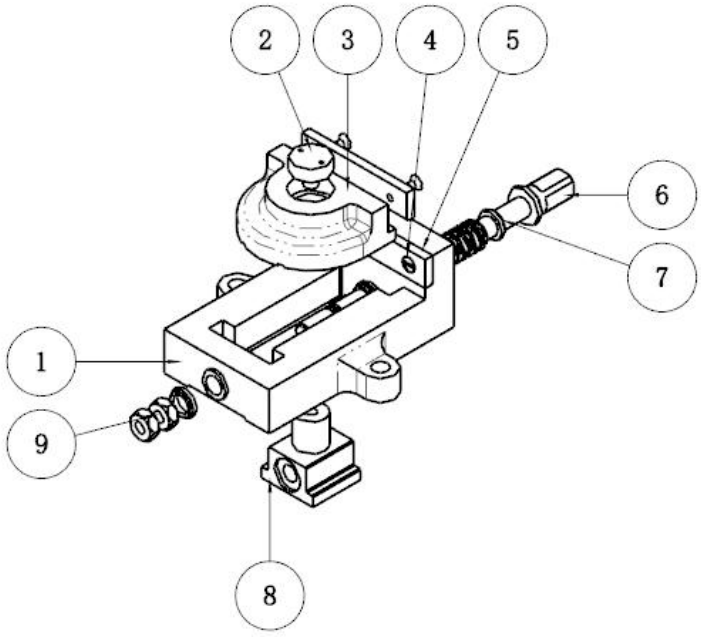



【Where is it】

Drawing Sheet Environment >> Dimension >> **Datum Feature**

### 3.4.4.4 Auto Balloon Optimization

The current auto balloon function has been optimized. A proper offset value will be calculated automatically when you use the auto balloon function, which will make the placement of the balloon more proper.

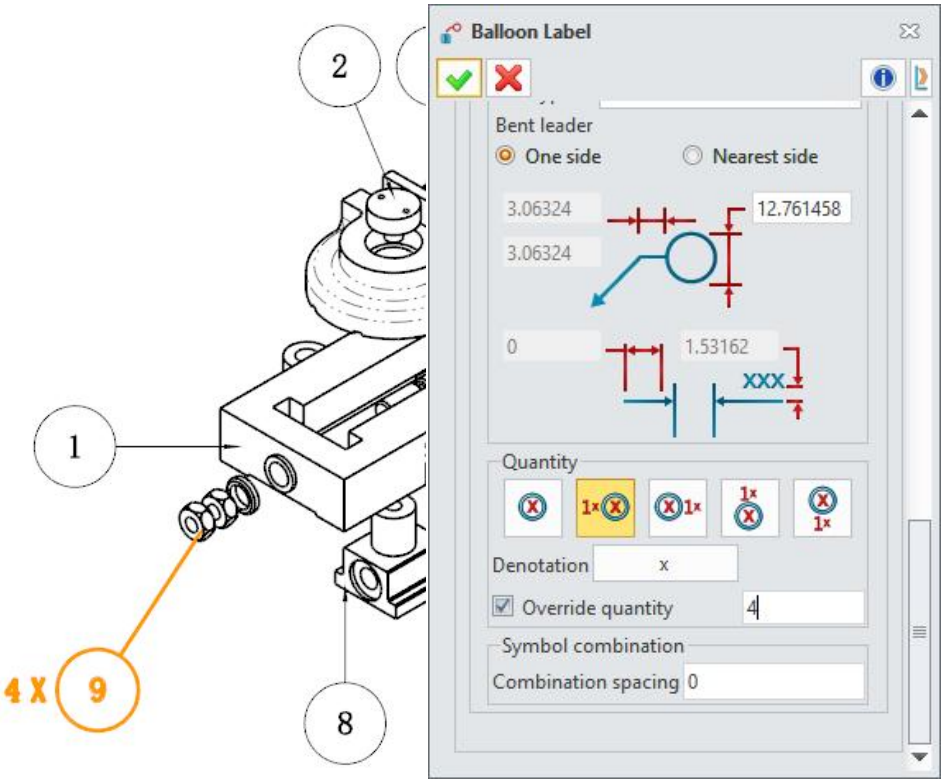



**【Where is it】**

Drawing Sheet Environment >> Dimension >> Annotation >> **Auto Balloon**

### 3.4.4.5 Balloon Optimization

An override quantity function has been added to balloon label. When you create or edit balloon label, some texts can be used to override the balloon's quantity, and the actual quantity of this part won't be shown.

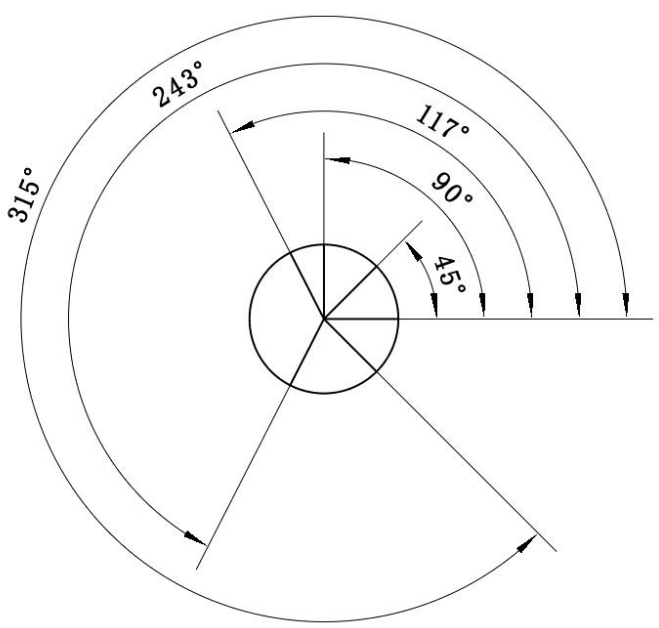


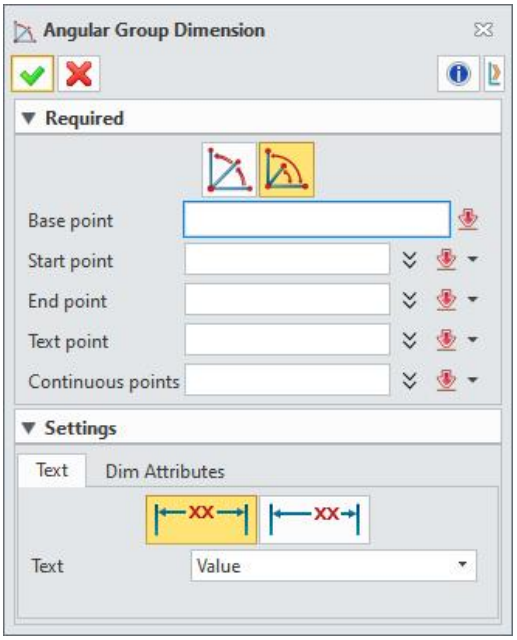

【Where is it】

Drawing Sheet Environment >> Dimension >> Annotation >> **Balloon Label**

## 3.4.4.6 Angular Group Dimension

The name of the original angular ordinate function has been changed to angular group dimension. Meanwhile, angular group dimension contains angular baseline mode, the new function. You can create the angular dimensions as follows.




**【Where is it】**

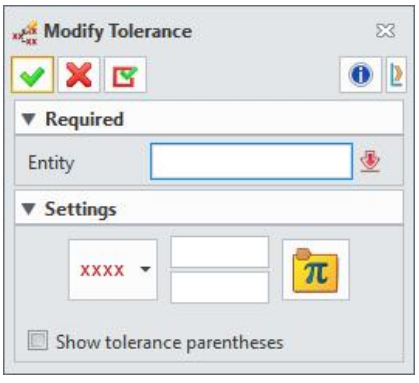
Drawing Sheet Environment >> Dimension >> Dimension >> **Angular Group Dimension**

## 3.4.4.7 Tolerance Optimization

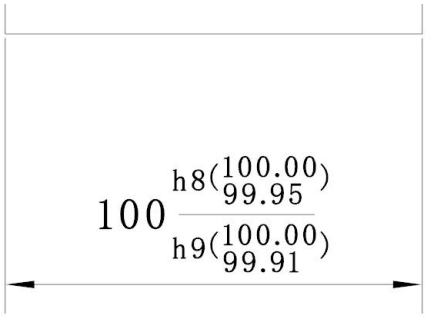
Tolerance function has been optimized, which contains “Modify Tolerance” in the PMI module, a new tolerance style and optimization in tolerance inquiry.

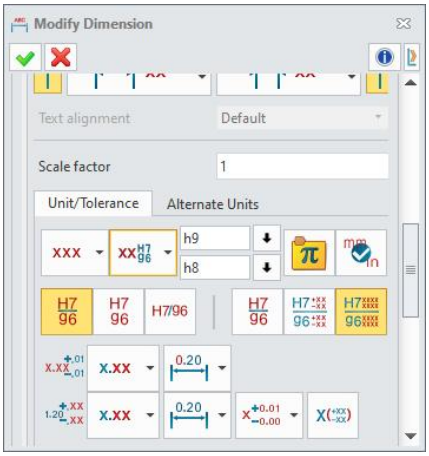
**【What users can do】**

- ✓ Edit tolerance information of the dimension in the PMI module.

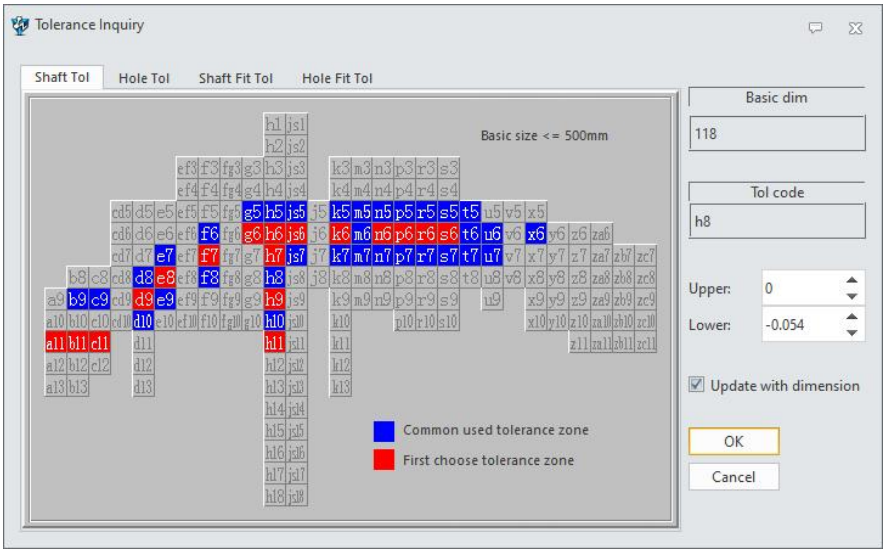



- ✓ Display the tolerance in a new style.

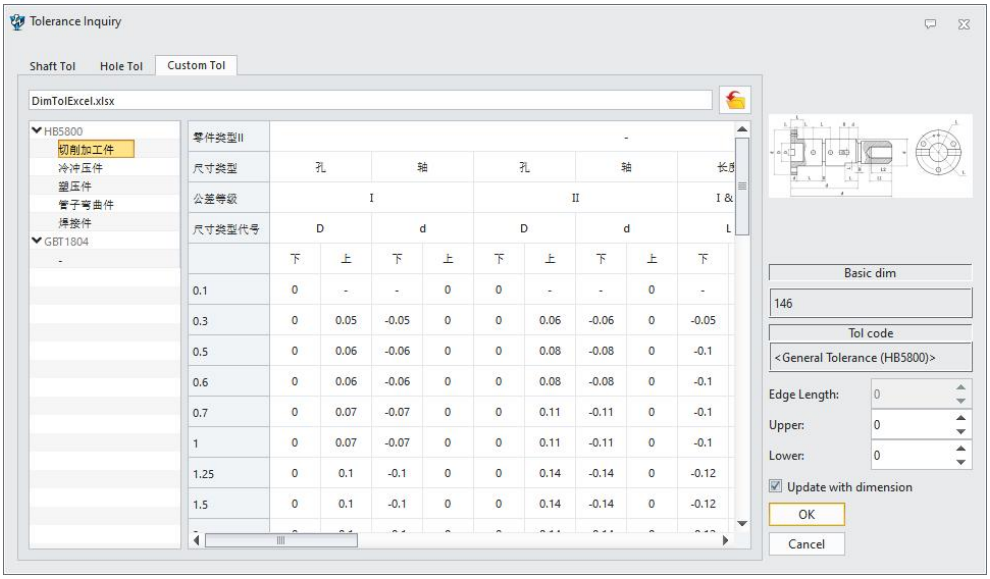



✓ Tolerance will be able to be updated with its basic dimension.




✓ Tolerance standards can be imported via Excel tables, supporting the HB 5800 and GB/T 1804 standards. The symmetrical tolerance query interface is as shown below.




**【Where is it】**

Part/Assembly Environment >> PMI >> **Edit Dimension**

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

## 3.4.4.8 Optimization of Labeling Occlusion Effect

Linear dimension, angle dimension and other dimension objects have added the filling and occlusion effect optimization, which can realize occlusion of dimension, graphic elements and other objects.

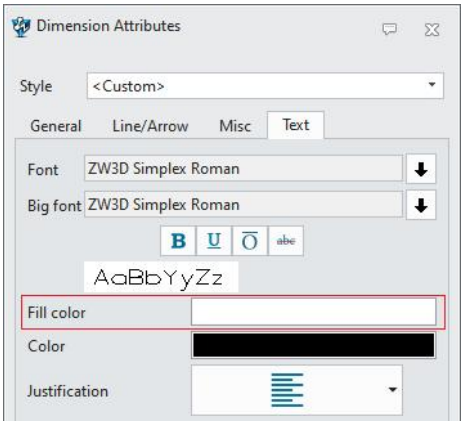
**【What users can do】**

✓ You can set the text fill color for the dimension object, and the fill color can set any color in the palette or follow the paper color.

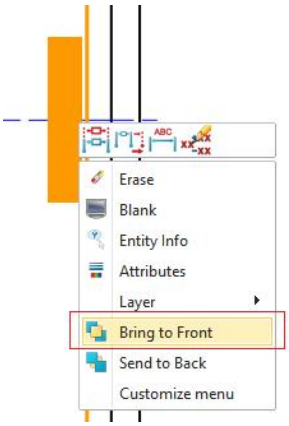
✓ Supports pre-and post-labeled objects, so that dimensional objects can block other objects and primitives.

**【Example】**

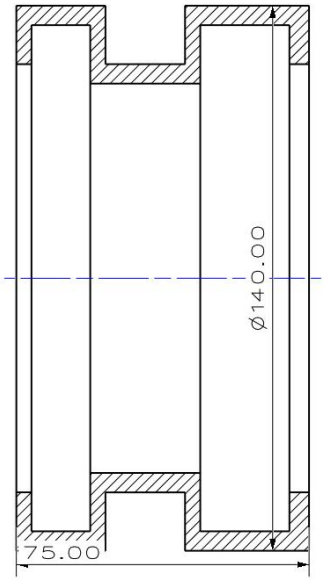
1) Set a color fill for dimension.




2) Bring the object to front




✓ You can realize the occlusion of dimension objects to other objects and primitives.



【Where is it】

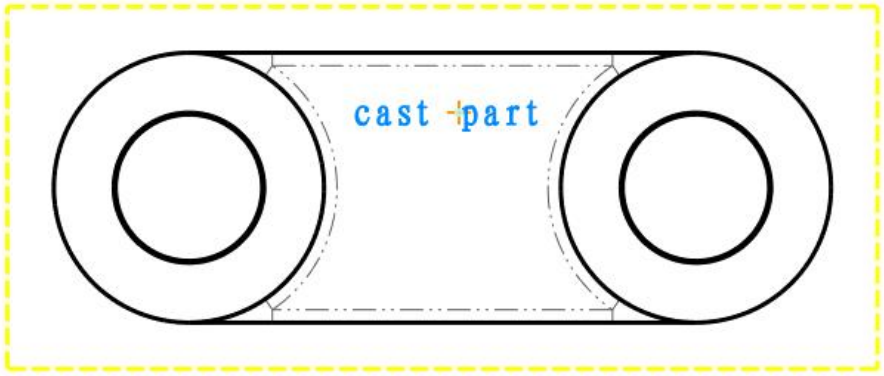
Drawing Environment > > Dimension > > **Properties**

## 3.4.4.9 Leadless Label Supporting Dragging Associated Views

The label function supports directly dragging leadless label to the view position to achieve association with the view and keep the position relatively fixed with the view.

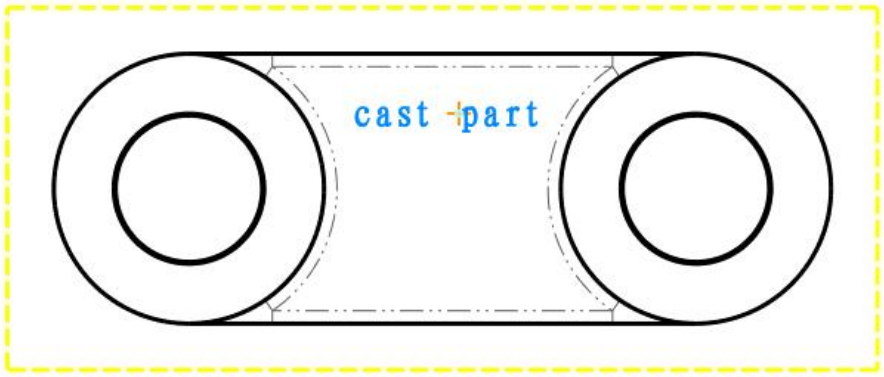
**【What users can do】**

- ✓ Associate a leaderless label with a view, keeping the position relatively fixed.




**【Example】**

- ✓ Create a label without a leader outside the view.
- ✓ Drag the label into the view to realize the association between the label and the view, and keep the position fixed first.



【Where is it】

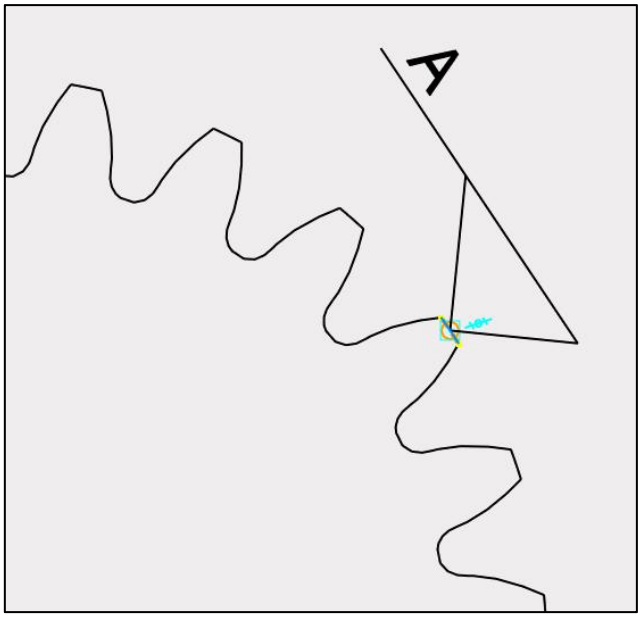
Drawing Environment > > Annotation > > Label

### 3.4.4.10 Insertion Block with Orientation Judgment

Intelligent direction judgment has been added when inserting the drawing sheet block. When the insert block is attached to a geometric object, the orientation of the block will be adjusted automatically according to the orientation of the geometry, and the user can flip the block as needed. This feature can reduce the number of user adjustments when inserting a block and improve the design efficiency

**【What users can do】**

✓ When inserting the block, intelligently judge the direction of the block, and after the insertion, support to flip the direction of the block.




【Where is it】

Drawing Sheet Environment >> Dimension >> Annotation >> **Insert Block**

### 3.4.4.11 Block Expression Display Optimization

Display of block has been optimized in the drawing. When the block refers to the expression and the value of the expression is empty, no expression is displayed in the drawing, and when the value of

the expression is not empty, the value is displayed in the drawing.

**【What users can do】**

✓ The drawing insert block does not display a blank expression.

**【Where is it】**

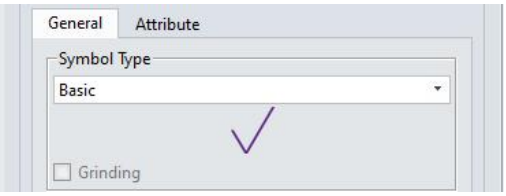
Drawing Sheet Environment >> Dimension >> Annotation >> **Insert Block**

### 3.4.4.12 Surface Finish Interface Optimization

The interface of the surface finish command has been optimized, and the type of roughness will be shown as individual icons, which will enhance convenience of selection.

**【What users can do】**

✓ Tile display, more intuitive selection of roughness type.

| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|  | |





**【Where is it】**

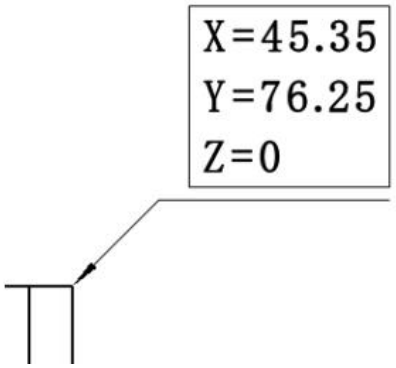
Drawing Sheet Environment >> Dimension >> Annotation >> **Surface Finish**

### 3.4.4.13 Label Coordinate Optimization

Function "Label Coordinate" has added rectangle, circle, right mark, left mark, long circle, square, scored circle, diamond, and triangle border styles, for further design demands.

**【What users can do】**

✓ Label Coordinate supports displaying border styles.




【Where is it】

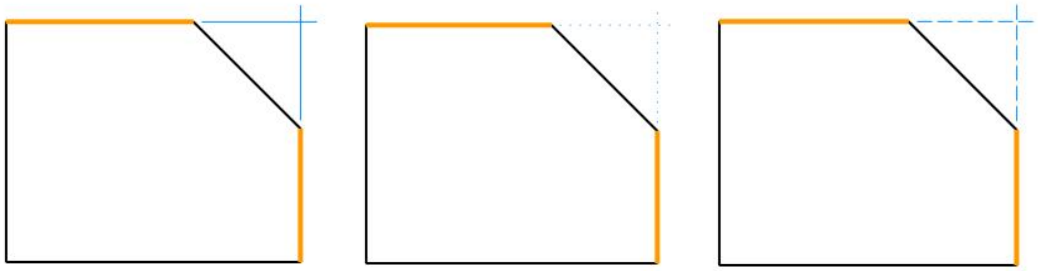
✓ Drawing Sheet Environment >> Dimension >> Annotation >> Label Coordinate

## 3.4.4.14 Intersection Symbol Line Type Extension

The intersection symbols of ZW3D are originally solid lines, but in actual scenarios, the intersection symbols need more line types. Therefore, the line types of intersecting symbols have been expanded to solid, dots, dash, dash dot, dash dot dot and long dash dot.

**【What users can do】**

✓ The line types of intersecting symbols expand to solid, dots, dash, dash dot, dash dot dot and long dash dot.



Solid Dot Dash

Dash Dot　　Dash Dot Dot　　Long Dash Dot

**【Where is it】**

Drawing Sheet Environment >> Dimension >> Annotation >> **Intersection Symbol**

## 3.4.4.15 Auto Drafting Optimization

The auto drafting function is optimized for sheet metal and complex models, allowing users to save the settings in the auto drafting, and optimize the automatic drafting results for sheet metal model characteristics and complex models, reducing the number of dimension objects for non-horizontal vertical objects, and marking out more accurate drawing sheets.

**【What users can do】**

- ✓ Supports saving the settings of tab pages such as views, labels and tables in configuration files.
- ✓ Tab page of drawings supports setting different retention configurations for different models.
- ✓ The view tab page adds two functions: inheriting PMI and displaying bent notes.
- ✓ Dimension tab page supports setting dimension settings for each view separately.
- ✓ Auto drafting supports saving settings in a configuration file.
- ✓ Auto-dimensioning dimension objects add such objects as dimensioning arc position dimension, dimensioning non-horizontal vertical straight line, bending inner diameter, bending outer diameter, bending position dimension, non-90 bending angle and near flange plane.
- ✓ Mark the position dimension of arc. If it is not checked, the position information of fillet will not be marked, that is, the dimensions in X and Y directions will not be marked.
- ✓ "Mark non-horizontal and vertical lines"; if it is not checked, the lines that are not vertical in X

and Y directions will not be marked.

✓ Bend Inner Diameter and Bend Outer Diameter control whether to dimension the values of the inner and outer diameters of sheet metal bending.

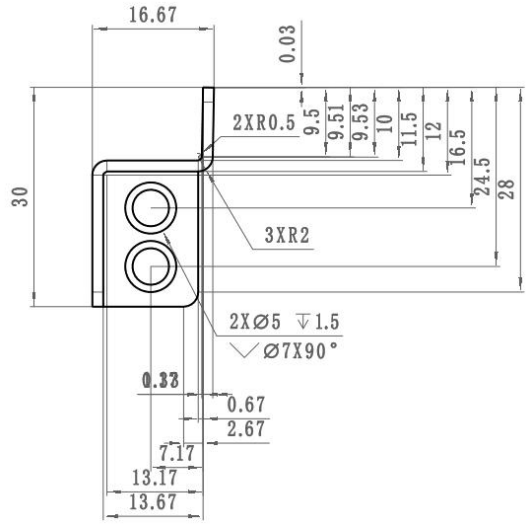
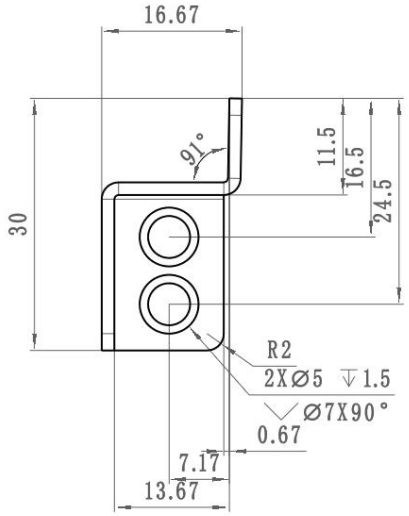
✓ The Bend Position Dimension controls whether to dimension the value of the sheet metal bending plane.

✓ "Non-90 bend angle" controls whether to dimension the value of non-90 bend angle of sheet metal.

✓ The "Proximal Flange Plane" controls whether to dimension the value of the sheet metal bending plane closest to the dimension far point.

**【Example】**

1) You can quickly select the desired plot configuration through the drop-down options.
2) The automatic labeling results are as follows

| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|  |  |

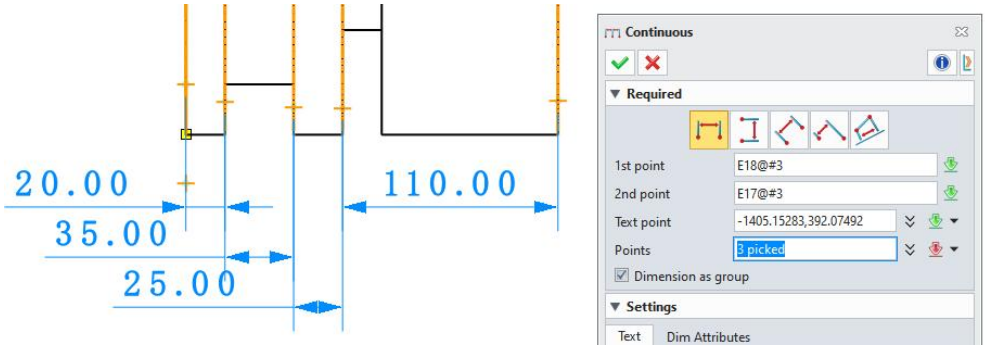


**【Where is it】**

Drawing Environment > > Labeling > > Labeling > > **Automatic Labeling**

## 3.4.4.16 Optimization of Continuous Labeling Function

The continuous dimension function automatically places the dimension. Due to the small size value, the dimension will be offset to the dimension direction to ensure that the dimension results do not overlap.




**【What users can do】**

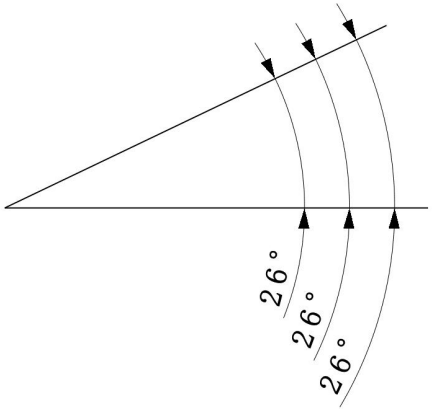
✓ It solves the problem of overlapping dimensions when the dimension value is too small and placed on a horizontal line.

**【Where is it】**

Drawing Environment > > Dimension > > **Continuous**

## 3.4.4.17 Angular Dimension Optimization

ZW3D 2026 has an optimized extension line of angular dimension. You can adjust the extension line to just one side which will make the angular dimension more clear and meet the standard.



【Where is it】

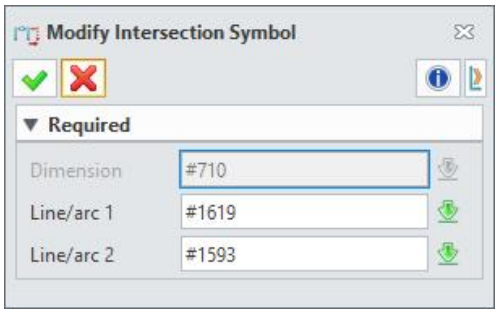
Drawing Sheet Environment >> Dimension >> Dimension >> **Angular Dimension**

## 3.4.4.18 Modify Dimension Points Optimization

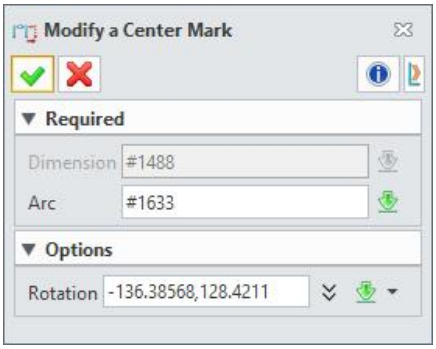
The support scope of the modification point command has been expanded to modify the intersection symbol and center mark.

**【What users can do】**

✓ Modify the intersection symbol.




✓ Modify the center mark.




【Where is it】

Drawing Sheet Environment >> Dimension >> Edit Dimension >> **Modify Dimension Points**

## 3.4.4.19 Hole Callout Supporting Reverse Identification.

Hole Callout function for the reverse face in the projection perspective supports the dimension result and adds the reverse identification.

**【What users can do】**

- ✓ Users can customize the dimension results of positive and negative holes.

**【Example】**

1) Open HoleCallFormat.txt in the supp folder under the installation directory.
2) Text labels specific to holes that define the positive and negative directions.

```
{Depth from feature}
#This configuration will read the feature depth instead of
#<H_DH1>, <H_H2>, <H_TH>
##          quantity          ##
QUANTITY=<H_Q>X
##          Depth1            ##
DEPTH1=<H_H1>
##          Depth2            ##
DEPTH2=<H_H2>
##          Thread depth      ##
THREAD_DEPTH=<H_TH>
##          through-hole      ##
THRUHOLE=THRU
##          hole-direction    ##
NEGATIVE_DIR=反面
POSITIVE_DIR=
```

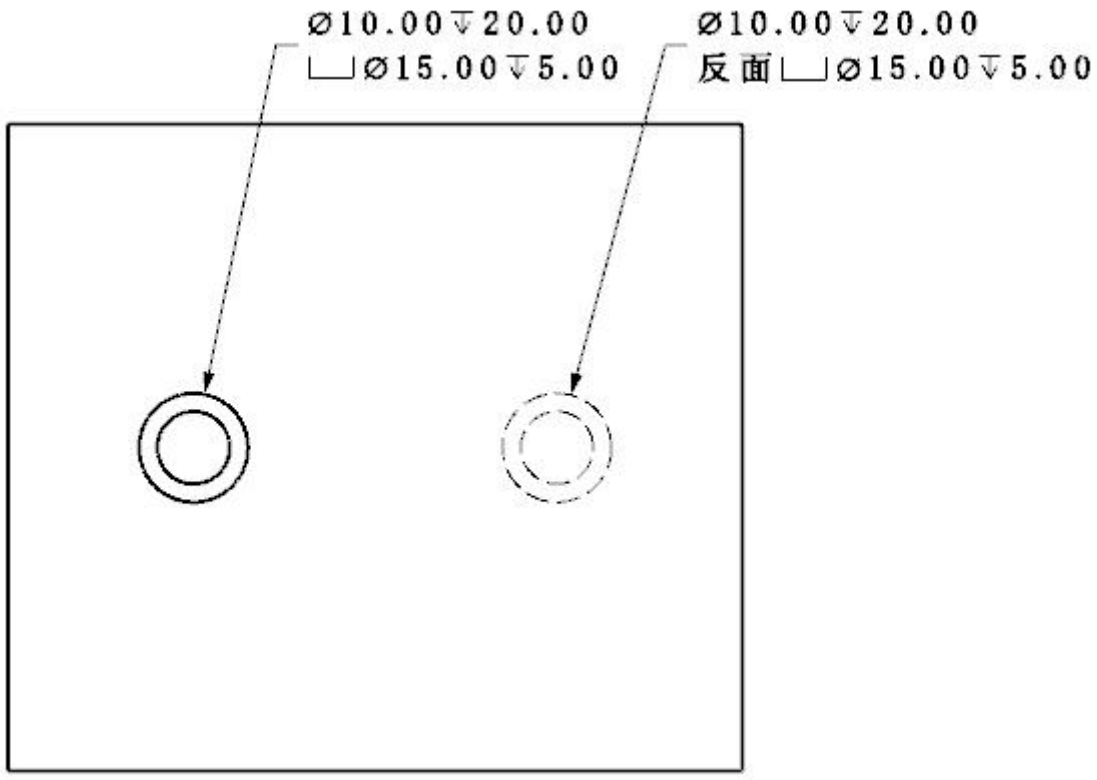
3) Add a reverse hole type

```
#General counter-bore hole#
GENERAL_COUNTER-BORE=[VxX]c<H_D1>[VxX]{<H_DH1><H_L>;\
                [VxX]y[VxX]c<H_D2>[VxX]{<H_DH2>
GENERAL_COUNTER-BORE_SCHAM=[VxX]c<H_D1> [VxX]{<H_DH1> <H_L>;\
                    [VxX]y[VxX]c<H_D2> [VxX]{<H_DH2>;\
                    [VxX]z[VxX]c<H_CD>X<H_CA>[VxX]b
REV_GENERAL_COUNTER-BORE=[VxX]c<H_D1>[VxX]{<H_DH1><H_L>;\
                <H_DIR>[VxX]y[VxX]c<H_D2>[VxX]{<H_DH2>
```

4) Determine the position of the text label

```
#General counter-bore hole#
GENERAL_COUNTER-BORE=[VxX]c<H_D1>[VxX]{<H_DH1><H_L>;\
                [VxX]y[VxX]c<H_D2>[VxX]{<H_DH2>
GENERAL_COUNTER-BORE_SCHAM=[VxX]c<H_D1> [VxX]{<H_DH1> <H_L>;\
                    [VxX]y[VxX]c<H_D2> [VxX]{<H_DH2>;\
                    [VxX]z[VxX]c<H_CD>X<H_CA>[VxX]b
REV_GENERAL_COUNTER-BORE=[VxX]c<H_D1>[VxX]{<H_DH1><H_L>;\
                <H_DIR>[VxX]y[VxX]c<H_D2>[VxX]{<H_DH2>
```

5) After restarting the software, mark the holes on the reverse side.

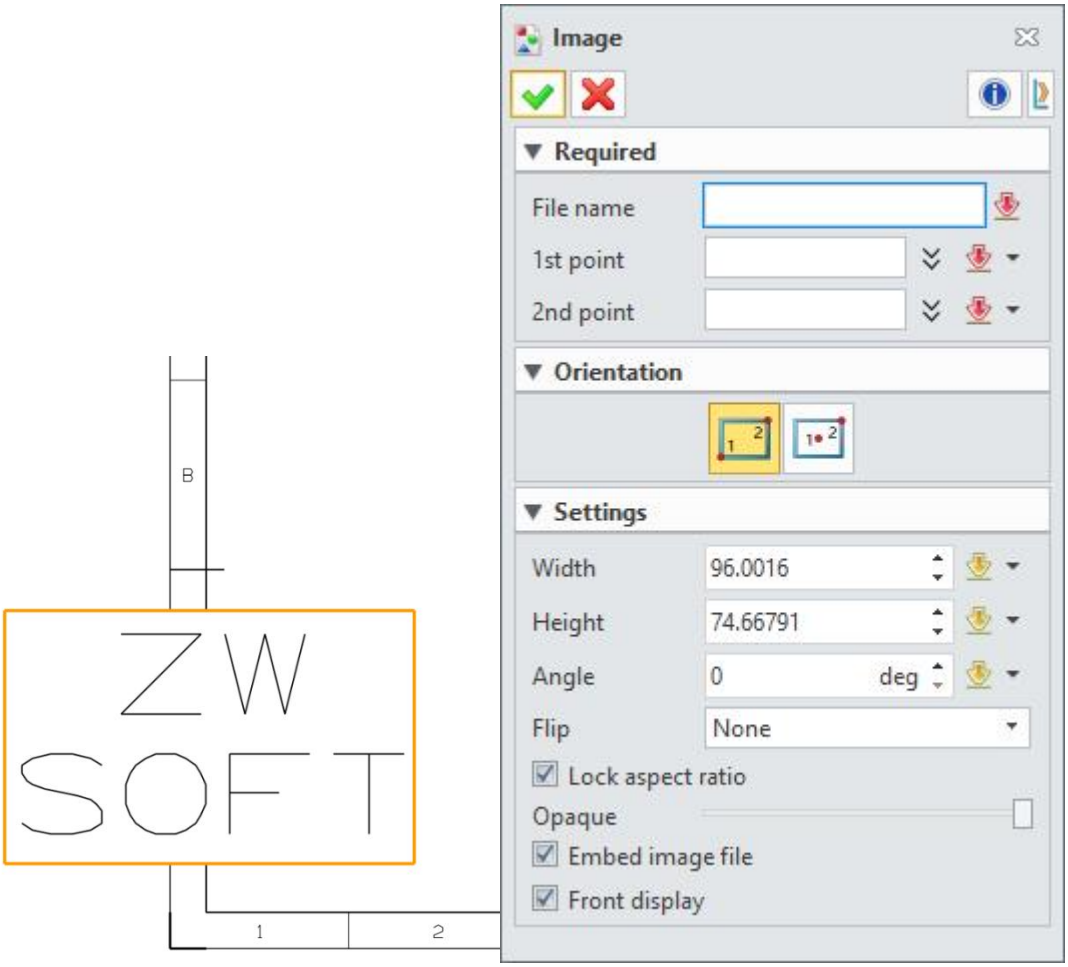



**【Where is it】**

Drawing Environment > > Dimension > > **Hole Callout**

## 3.4.4.20 Image Setting Optimization

Function "front display" has been added for the image inserted. When you want to hide some information or add a watermark, the image can be displayed on the top through this function.

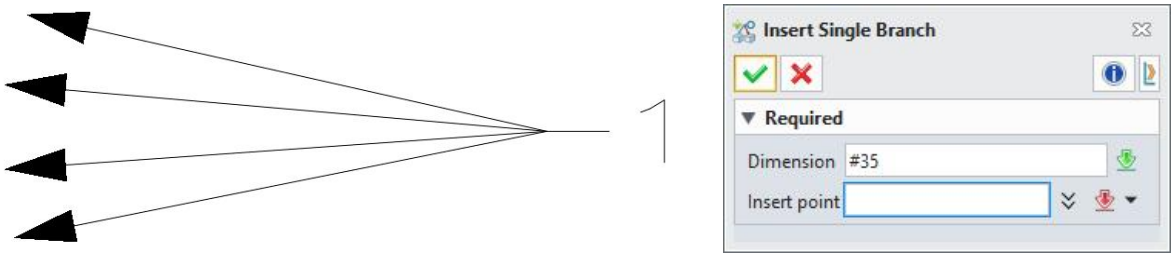



【Where is it】

Drawing Sheet Environment >> Dimension >> Symbol >> **Image**

## 3.4.4.21 Branch Optimization

Function "insert single branch" has been added and function "delete branch" has been optimized. You can insert or delete branches continuously by these two functions.



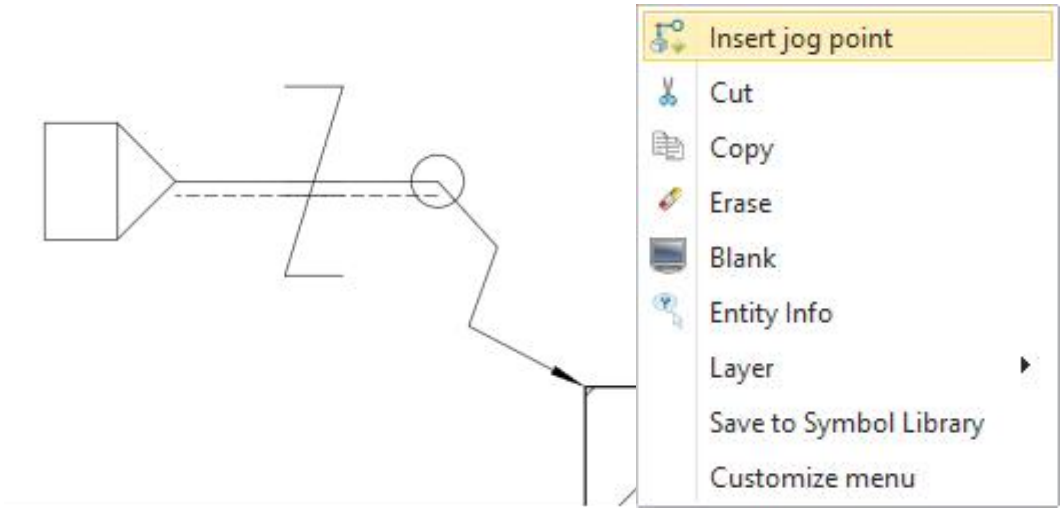
【Where is it】

Drawing Sheet Environment >> Branch Dimension >> Right-Click Menu >> **Insert Single Branch**

Drawing Sheet Environment >> Branch Dimension >> Right-Click Menu >> **Delete Branch**

### 3.4.4.22 Weld Symbol Optimization

Function "Insert jog point" has been added to the weld symbol. Users can insert jog point in the branch of the weld symbol, achieving the bending of branch.

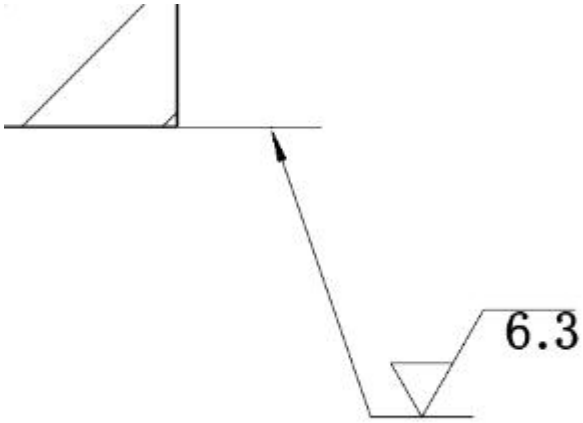


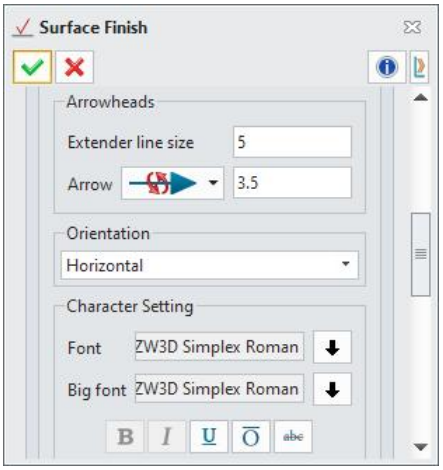

【Where is it】

Drawing Sheet Environment >> Dimension >> Symbol >> **Weld Symbol**

### 3.4.4.23 Arrowhead Extension Line Optimization

Arrowhead extension line has been added to note, balloon, feature control weld symbol and surface finish. When the arrowhead is dragged to the region outside entity, an extension line will be generated automatically.



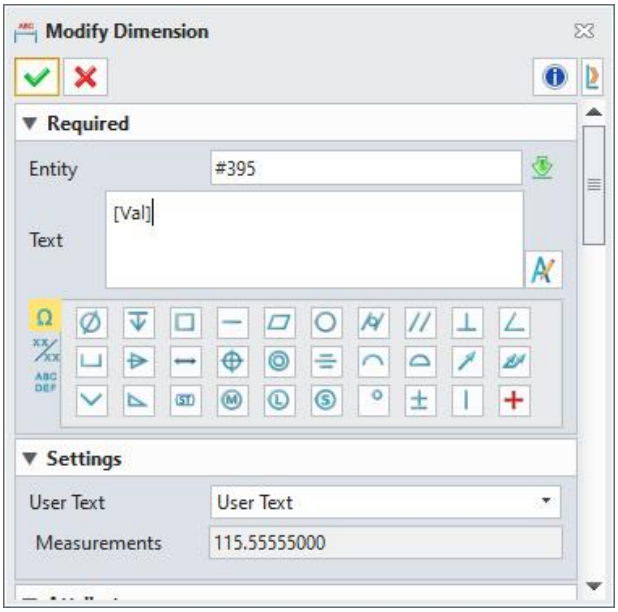




**【Where is it】**

Drawing Sheet Environment >> Dimension >> Annotation

## 3.4.4.24 New Measurements

Function "measurements" has been added to the modify dimension panel. Through this function, you will be able to check the precise value of the dimension.

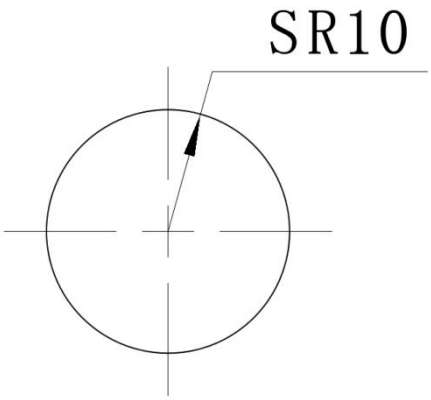



**【Where is it】**

Drawing Sheet Environment >> Modify Dimension >> **Measurements**

### 3.4.4.25 New Sphere Feature Dimension

Sphere feature's dimension has been added. The sphere feature will be able to be recognized automatically in the dimension and radial/diametric function.




**【Where is it】**

Drawing Sheet Environment >> Dimension >> Dimension >> **Dimension**

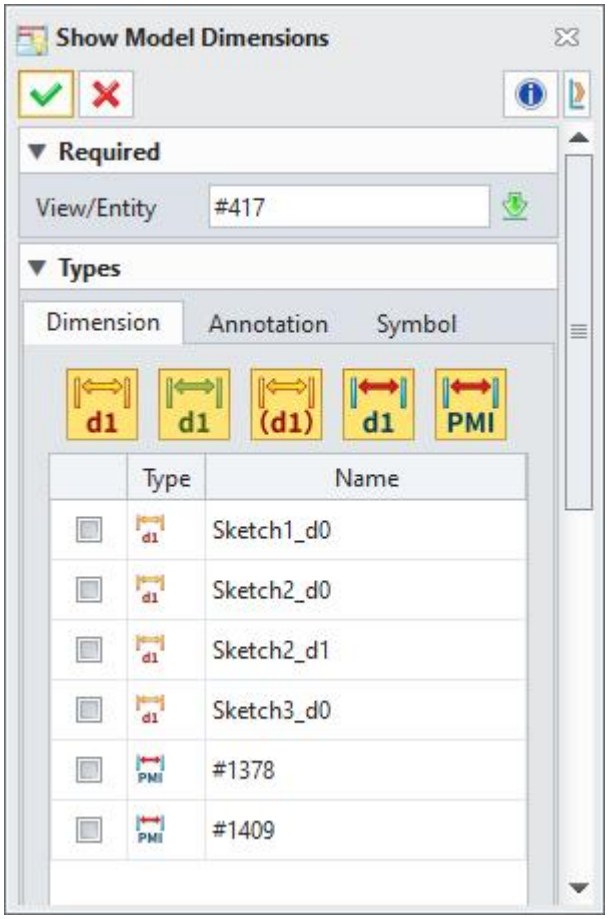
Drawing Sheet Environment >> Dimension >> Dimension >> **Radial/Diametric**

### 3.4.4.26 New Show Model Dimensions

Function " show model dimensions" has been added. Through this function, you can show the dimensions, annotations and symbols of the 3D model in the view selectively. Specifically, dimensions refer to strong dimension, weak dimension and reference dimension in the sketch. Annotations and symbols refers to all annotations and symbols created in the PMI.

**【What users can do】**

- ✓ Display dimensions, annotations and symbols in the views of the 3D model selectively.
- ✓ The dimensions, annotations and symbols inherit the position of corresponding entities in the sketch and PMI.
- ✓ Select in batch through the shortcut Shift or Ctrl.

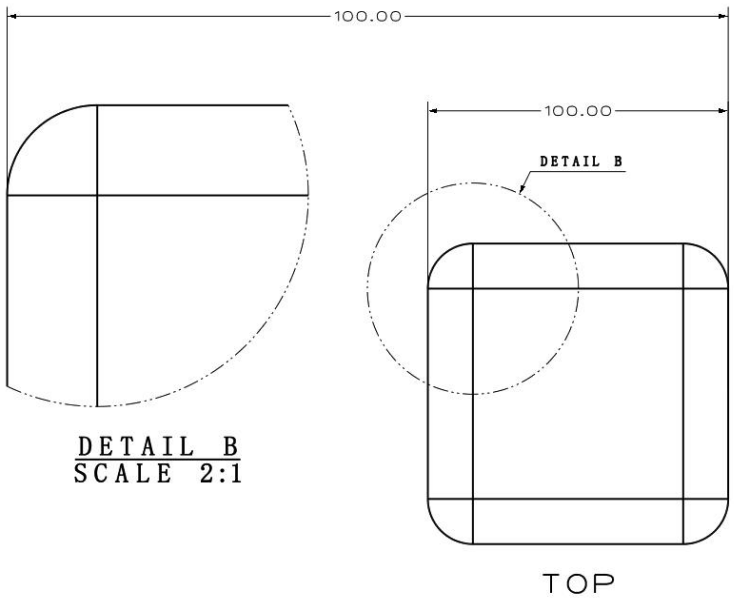



**【Where is it】**

Drawing Sheet Environment >> Dimension >> Dimension >> **Show Model Dimensions**

## 3.4.4.27 Cross-View Dimension Optimization

Cross-view dimension has been optimized. You will not be able to create cross view dimensions except cross-view dimensions between detail view, crop view etc. and its base view.




**【Where is it】**

Drawing Sheet Environment >> Dimension >> **Dimension**

## 3.4.4.28 Dimension Optimization

The dimension command is optimized in version 2026, which supports moving and editing existing dimensions during the process and supports setting tolerances.

**【What users can do】**

✓ When the dimension command is activated, pre-highlight appears when the mouse moves over the existing label.

✓ When the dimension command is activated, click the existing label with the left mouse button and hold it down to move the position of the existing label.

✓ When the dimension command is activated, double-click the existing annotation with the left mouse button to edit the existing annotation. When you exit the attribute editing through OK, Cancel or ESC, you should return to the annotation interface.

✓ The dimension command adds two types of dimension attribute settings of "Unit/Tolerance" and "Alternate Units".

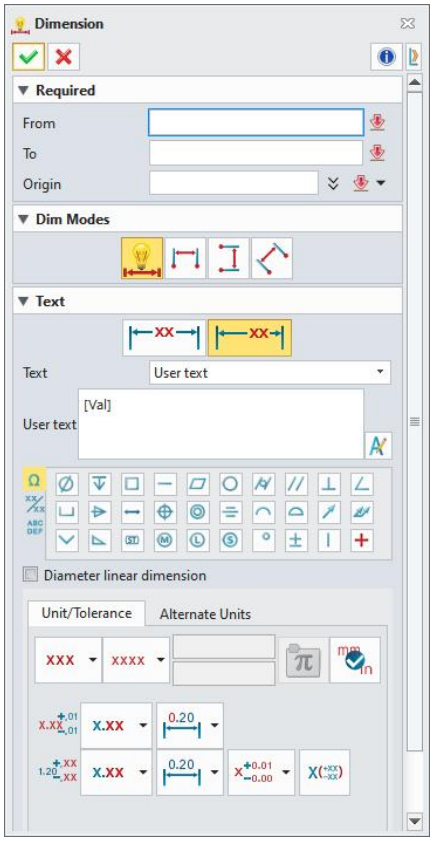
✓ After the user has entered the text and set the properties of unit/tolerance and replacement unit, the input value will not be reset until the user ends the command actively.

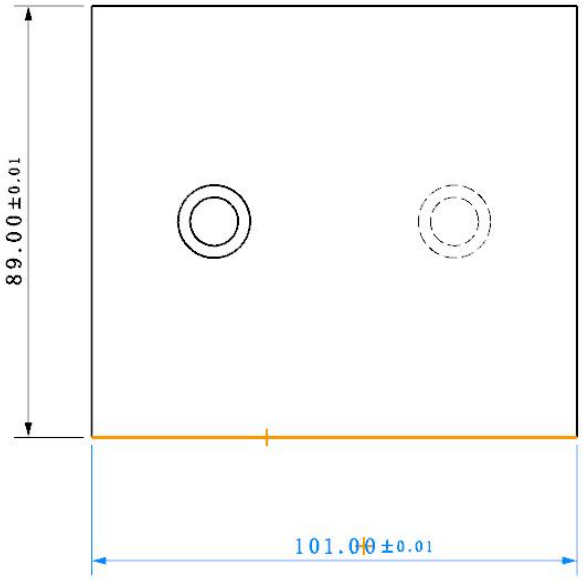
**【Notes】**

• Due to the lack of tolerance grade for angle dimension, and the main unit options are displayed in the form of fractional seconds instead of fractional ones, the angle dimension result will not take effect if the attribute of tolerance grade or the display form of fractional is set for the attribute of the dimension command.

**【Example】**

After the user enables the dimension command, you can add characters and change the tolerance in the user's text and complete the dimension creation.




**【Where is it】**

Drawing > > Dimension > > Dimension

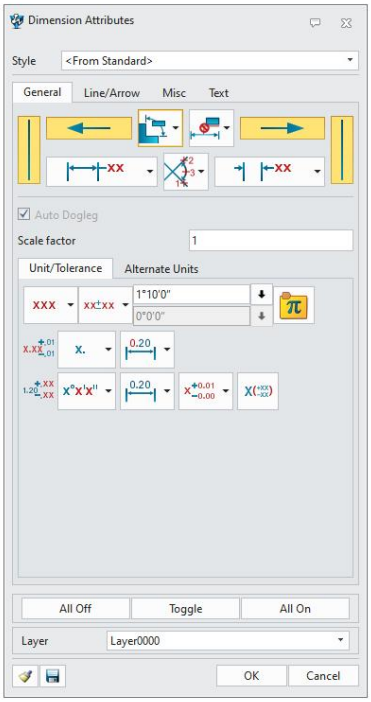
## 3.4.4.29 Optimization of Angular Dimension（ZW3D2026 SP1）

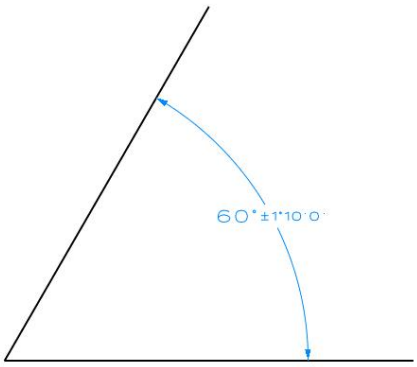
Angular annotations have been optimized. ZW3D 2026 SP supports entering angular tolerances in the format of XX°XX′XX″, that is, degrees, minutes, and seconds, eliminating the need for manual conversion.

**【What users can do】**

- ✓ Enter angular tolerance in the format of XX°XX′XX″.
- ✓ Convert inputs in other formats to this format automatically.

**【Example】**



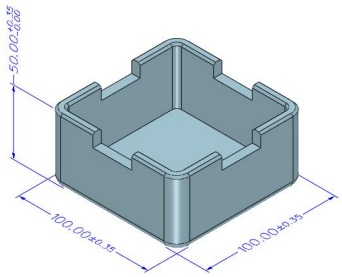
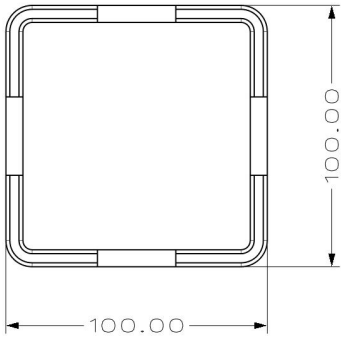
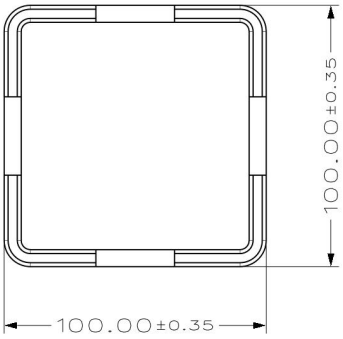

**【Where is it】**

Drawing >> Angular dimension >> **Dimension Attributes**

PMI >> Angular dimension >> **Dimension Attributes**

### 3.4.4.30 Optimization of Inheriting PMI（ZW3D 2026 SP1）

The drawing dimensions can inherit the units and tolerance attributes of PMI. Right-clicking on the annotation and using the option “Inherited from Part” can control the association status of these attributes.

| Model | ZW3D 2026 | ZW3D 2026 SP1 |
|---|---|---|
|  |  |  |



**【Where is it】**

View Attributes >> **Inherit PMI**

## 3.4.5 Table Optimization

### 3.4.5.1 New Order by Name and Auto Update IDs after Sort

Functions "Order by name" and "Auto update IDs after sort" have been added in the sorting functions of Hole Table, Electrode Table, Structural BOM Table, Weld Table and Annotation Table.

**【What users can do】**

- ✓ Reorder the table contents by name.
- ✓ After the user adjusts the sequence in the table, the software automatically updates the corresponding ID.

**【Where is it】**

Drawing Sheet Environment >> Layout >> Table >> **Hole Table/Electrode Table/Structural BOM Table/ Weld Table/Annotation Table**

## 3.4.5.2 New Show Unit to BOM

In practical scenarios, the BOM will have both display units and non-display units. The option of "Show unit" has been added to the BOM table, so that the users can switch flexibly based on the actual needs.

**【What users can do】**

✓ Control whether the units in the BOM are displayed or not.

| ID | Name | Cost | Number | Quantity | Material | Stock Size |
|---|---|---|---|---|---|---|
| 12 | W045-082 | | 10045-0008200 | 1 | Q235A | 25*28*28(mm) |
| 11 | W038-044 | | 10038-0004400 | 1 | Q235A | 120*135*11(mm) |
| 10 | W038-013A | | 10038-0001301 | 2 | Q235A | 625*525*14(mm) |
| 9 | W017-001 | | 10017-0000100 | 2 | Q235A | 35*25*35(mm) |
| 8 | W008-129C | | 10008-0012903 | 1 | Q235A | 80*105*52(mm) |
| 7 | 70F2-DT-02-17A | | 12303-7217001 | 1 | Q235A | 275*280*60(mm) |
| 6 | 500SeIII-D-17-05 | | 13183-1705000 | 1 | Q235A | 1033*1700*4(mm) |
| 5 | 500SeIII-D-17-04 | | 13183-1704000 | 1 | Q235A t6X534X1033 | 6*1033*534(mm) |
| 4 | 500SeIII-D-17-03 | | 13183-1703000 | 1 | Q235A | 3*1032*410(mm) |
| 3 | 500SeIII-D-17-02 | | 13183-1702000 | 1 | Q235A | 16*1033*534(mm) |
| 2 | 500SeIII-D-17-01 | | 13183-1701000 | 1 | Q235A | 1732*1045*540(mm) |
| 1 | 380SEIII-D-02-08 | | 13163-0208000 | 2 | Q235A | 625*525*48(mm) |

Display

| ID | Name | Cost | Number | Quantity | Material | Stock Size |
|---|---|---|---|---|---|---|
| 12 | W045-082 | | 10045-0008200 | 1 | Q235A | 25*28*28 |
| 11 | W038-044 | | 10038-0004400 | 1 | Q235A | 120*135*11 |
| 10 | W038-013A | | 10038-0001301 | 2 | Q235A | 625*525*14 |
| 9 | W017-001 | | 10017-0000100 | 2 | Q235A | 35*25*35 |
| 8 | W008-129C | | 10008-0012903 | 1 | Q235A | 80*105*52 |
| 7 | 70F2-DT-02-17A | | 12303-7217001 | 1 | Q235A | 275*280*60 |
| 6 | 500SeIII-D-17-05 | | 13183-1705000 | 1 | Q235A | 1033*1700*4 |
| 5 | 500SeIII-D-17-04 | | 13183-1704000 | 1 | Q235A t6X534X1033 | 6*1033*534 |
| 4 | 500SeIII-D-17-03 | | 13183-1703000 | 1 | Q235A | 3*1032*410 |
| 3 | 500SeIII-D-17-02 | | 13183-1702000 | 1 | Q235A | 16*1033*534 |
| 2 | 500SeIII-D-17-01 | | 13183-1701000 | 1 | Q235A | 1732*1045*540 |
| 1 | 380SEIII-D-02-08 | | 13163-0208000 | 2 | Q235A | 625*525*48 |

Blank

**【Where is it】**

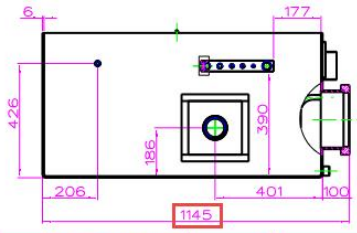
Drawing Sheet Environment >> Layout >> Table >> **BOM**

### 3.4.5.3 New Insert Dimension to BOM

Function "Insert Dimension" has been added in the BOM table, which is able to reference the dimensions in the drawing. When the dimensions change, the table data can be updated synchronously.

**【What users can do】**

✓ The BOM table can reference the dimensions in the drawing.




| ID | Name | Cost | Number | Quantity | Material | Length |
|---|---|---|---|---|---|---|
| 12 | W045-082 | | 10045-0008200 | 1 | Q235A | 25 |
| 11 | W038-044 | | 10038-0004400 | 1 | Q235A | 120 |
| 10 | W038-013A | | 10038-0001301 | 2 | Q235A | 625 |
| 9 | W017-001 | | 10017-0000100 | 2 | Q235A | 35 |
| 8 | W008-129C | | 10008-0012903 | 1 | Q235A | 80 |
| 7 | 70F2-DT-02-17A | | 12303-7217001 | 1 | Q235A | 275 |
| 6 | 500SeIII-D-17-05 | | 13183-1705000 | 1 | Q235A | 1145 |
| 5 | 500SeIII-D-17-04 | | 13183-1704000 | 1 | Q235A t6X534X1033 | 6 |
| 4 | 500SeIII-D-17-03 | | 13183-1703000 | 1 | Q235A | 3 |
| 3 | 500SeIII-D-17-02 | | 13183-1702000 | 1 | Q235A | 16 |
| 2 | 500SeIII-D-17-01 | | 13183-1701000 | 1 | Q235A | 1732 |
| 1 | 380SEIII-D-02-08 | | 13163-0208000 | 2 | Q235A | 625 |

**【Where is it】**

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

### 3.4.5.4 New Update Hole Features with View to Hole Table

To help users quickly add the holes in the view to the hole table, especially the hole table update after the model modification, function "Update hole features with view" has been added to the Hole Table.

**【What users can do】**

✓ When the "Update hole features with view" is checked, all the holes in the view are updated to

the hole table.

**【Where is it】**

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

### 3.4.5.5 New Format Text to Hole Table

Function “Format Text” has been added in the hole table, based on a variety style in practical application. So the users can customize the hole annotation style and apply it to the hole table.

**【What users can do】**

- ✓ When the Format Text selects the default, the hole table represents the hole size according to the default style.
- ✓ When Format Text selects the depth from feature, the hole table represents the hole size in the style of the HoleCalloutFormat.txt file configuration in the supp folder.

**【Where is it】**

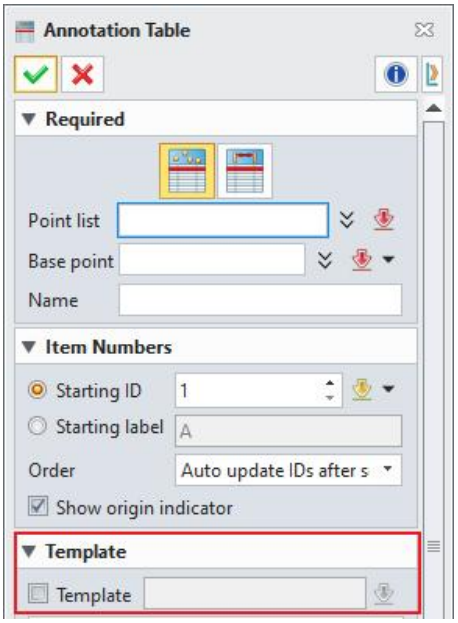
Drawing Sheet Environment >> Layout >> Table >> **Hole Table**

### 3.4.5.6 Optimization of the Annotation Table（ZW3D2026 SP1）

The annotation table function has been enhanced to include the ability to import templates.

**【What users can do】**

- ✓ Users can choose the customized templates to generate the annotation table scheme.

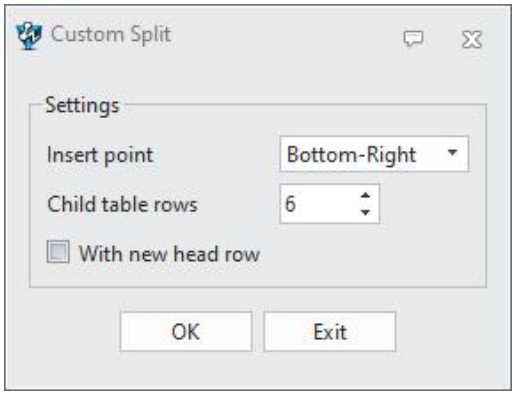



**【Where is it】**

Drawing Sheet>> Layout>> Table>> **Annotation Table**

## 3.4.5.7 Optimization of BOM Table Splitting Function（ZW3D2026 SP1）

Users can make custom settings when splitting the BOM table.




**【What users can do】**

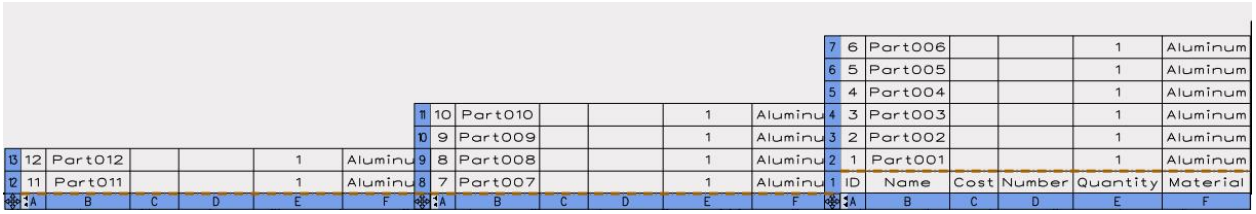
- ✓ BOM table can be split multiple times；
- ✓ You can set the number of rows to be split each time；
- ✓ You can manually specify the position of the table after each split.

**【Notes】**

The number of rows set for the sub-table is the number of rows for the next sub-table after splitting, not the number of rows of the current table.

**【Example】**

Users can set the number of rows for the next sub-table. This operation can be repeated to generate multiple sub-tables with the desired number of rows as per the user's requirements.



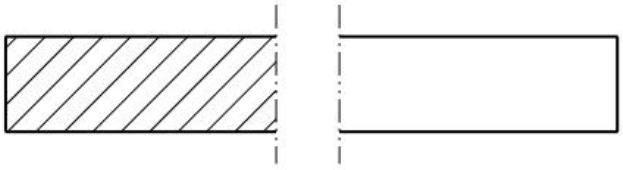

**【Where is it】**

Drawing Sheet>>Layout>> Table>> **BOM Table**

## 3.4.6 Optimized drawing function（ZW3D2026 SP1）

### 3.4.6.1 Optimization of the Cross Hatch

The optimization of the Cross Hatch enables using the breaking lines of the broken view as the filling boundary.



**【What users can do】**

✓ When activating the Cross Hatch and move your mouse over the breaking lines , a pre-highlight appears.

✓ When activating the Cross Hatch, you can select the breaking line as the input for the Entities.

**【Notes】**

When selecting the breaking lines, two breaking lines will be selected simultaneously. However, only the enclosed area designated as internal will generate the section lines.

**【Where is it】**

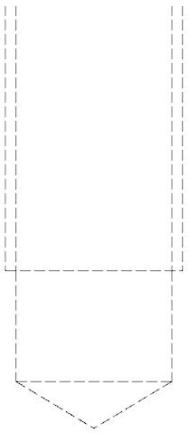
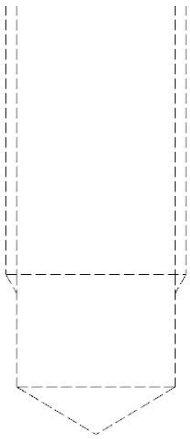
Drawing Sheet>> Drawing >> **Cross Hatch**

## 3.4.7 Style Manager Optimization

### 3.4.7.1 New Thread Closing Line

The generation of thread ending line has been supported when projecting.

**【What users can do】**

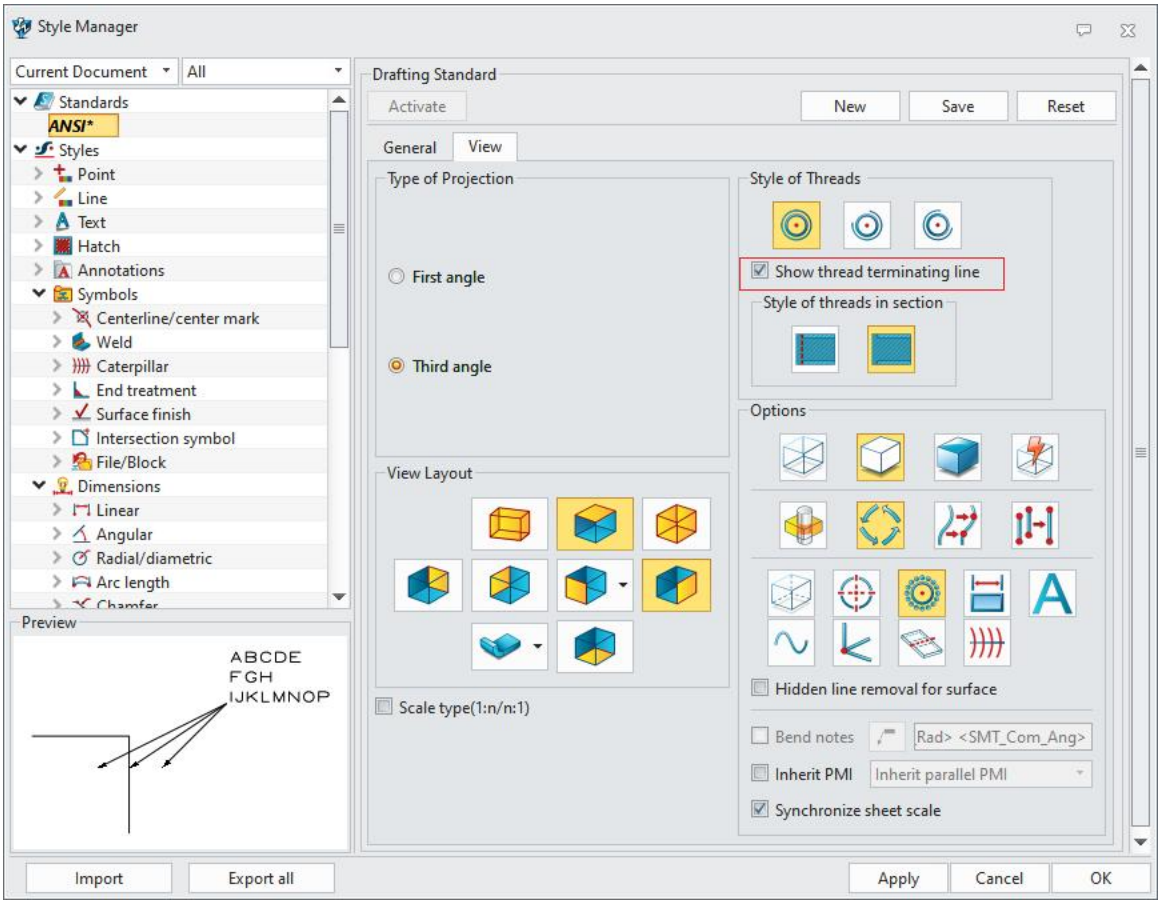
| No thread ending line | Threaded ending line |
| --- | --- |
|  |  |

**【Notes】**

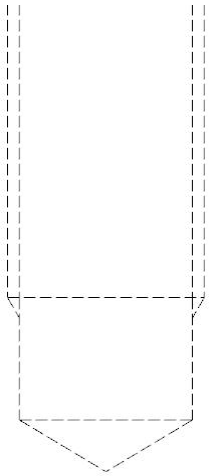
- When making a projection view, when the axial direction of the threaded hole is not parallel to the projection plane, the closing line will not be generated.
- When making a sectional view, the closing line will not be generated when the cutting position is not on the axis of the threaded hole.

**【Example】**

1) The Style Manager sets the thread terminating line.




2) Thread hole projection automatically generates thread terminating line.

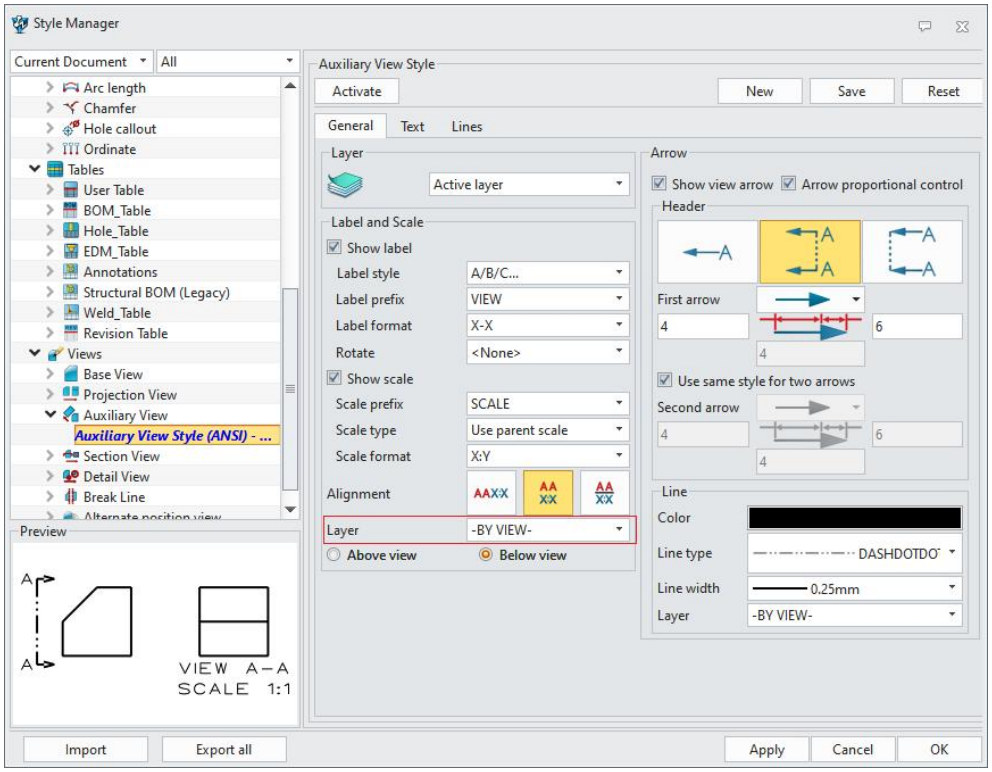


## 【Where is it】

Drawing > > Style Manager > > View > > **Show thread terminating line**

### 3.4.7.2 View Arrow, Label Supporting Setting Layer.

Auxiliary view, detail view, section view arrow symbol, label text set the layer separately.




**【Notes】**

- When the layer value is-BY VIEW-, the layer properties of label and arrow change with the layer properties of the view.
- When the layer value is a specific value (such as layer 0001, etc.), the layer property changes of the view will not affect the layer property values of label and arrow.

**【Where is it】**

Drawing > > Tools > > Style Manager > > View > > **Label and Scale**

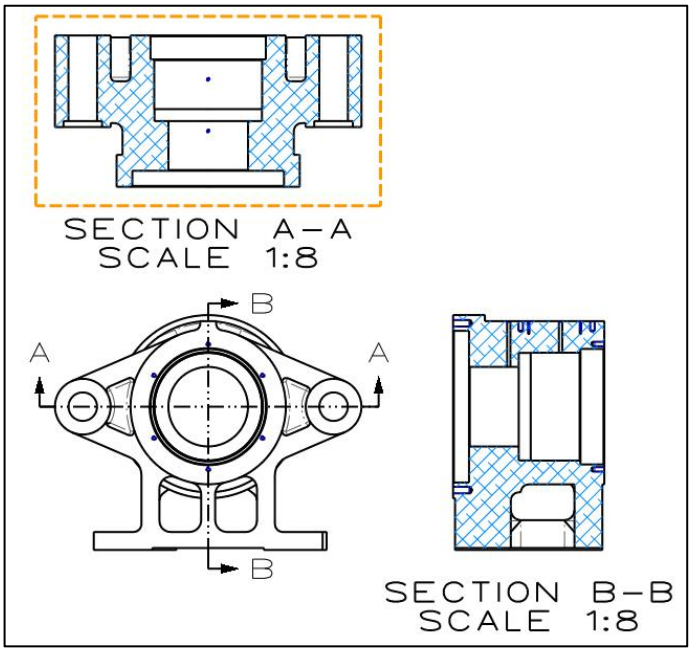
### 3.4.7.3 New "Apply to all from same component" to Section View

The drawing sheet specification requires that the section lines of the same components in the same drawing be consistent. For this application scenario, option "Apply to all from same component" has been added. When checked the section line style of one of the section views in the drawing will be

changed, and the changes will be automatically applied to other section views of the same component in the drawing.

**【What users can do】**

- ✓ Modify the hatch style of all section views of the component in the drawing at once.




**【Where is it】**

Drawing Sheet Environment >> Tool >> Style Manager >> Section View >> Hatch Attributes >> **Apply to all from same component**

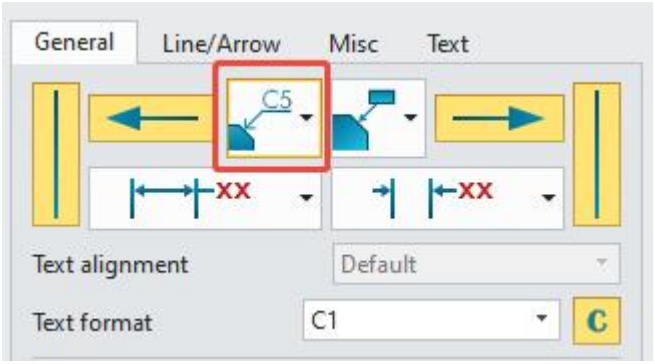
### 3.4.7.4 Text Direction Optimization for Linear Chamfer Dimensions

Optimization of attribute settings for chamfer dimensions in drawing sheets. In ZW3D 2026, the three types of "Normal", "Always perpendicular", and "Along the model line" for leader chamfer dimensions will be adjusted to the dimension attributes, known as text direction. The text direction option is also added to the dimension attributes page.

**【What users can do】**

- ✓ Set the text direction of the chamfer dimensions in the Style Manager.

✓ Set the text direction of the chamfer dimension in the dimension attribute page.




**【Where is it】**

Drawing Sheet Environment >> Tool >> Style Manager >> Chamfer >> **Text Direction**

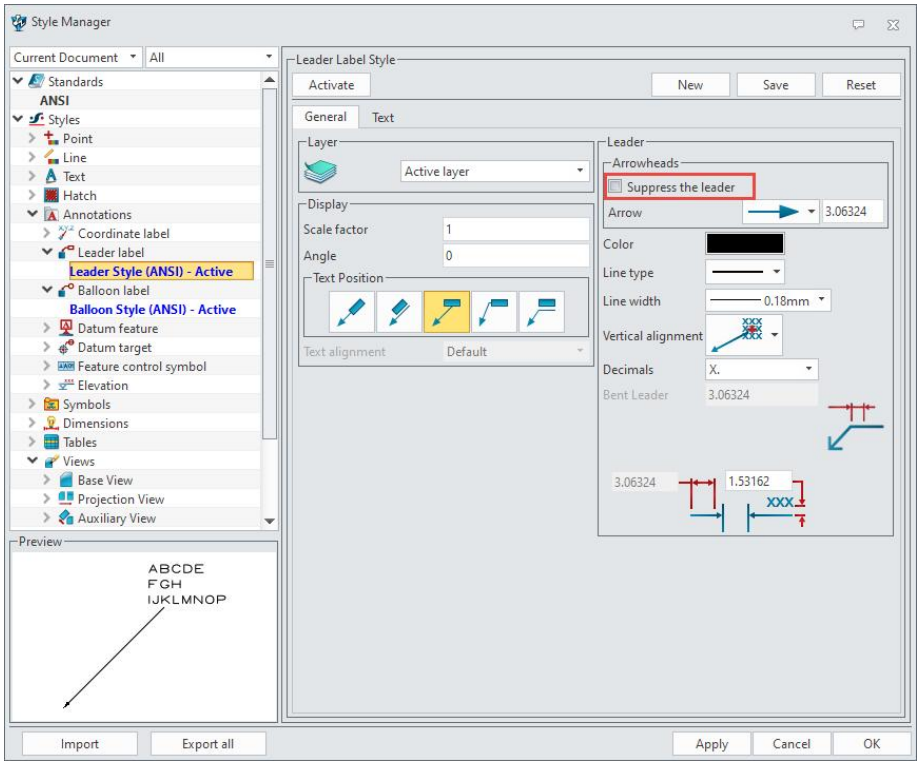
Drawing Sheet Environment >> Dimension >> Linear Chamfer Dimension>> Attribute >> **Text direction**

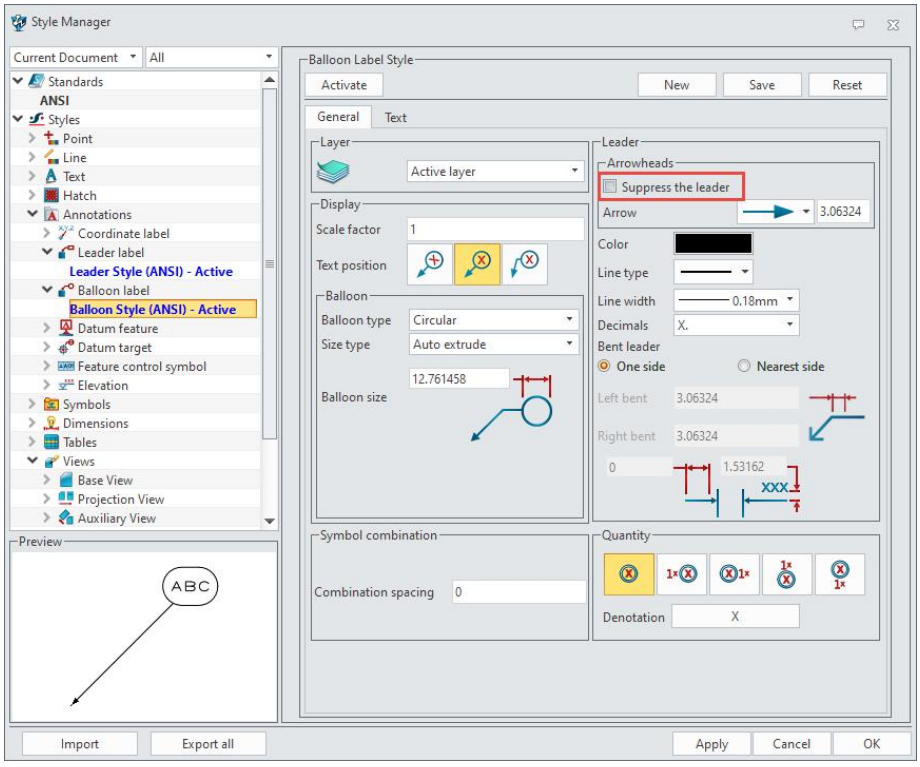
### 3.4.7.5 New "Suppress the lead" Option to Leader Label / Balloon Label

In the drawing specification, there are two styles of lead and bubble annotation with lead or without lead. Option "Suppress the leader" has been added in the drawing style manager. And the user uniformly configures the lead style and saves the style to the drawing template.

**【What users can do】**

✓ Set whether the Leader Label and the Balloon Label suppress the lead in the style manager.




✓ Supports saving the style of the suppressed lead to the drawing template.

**【Where is it】**

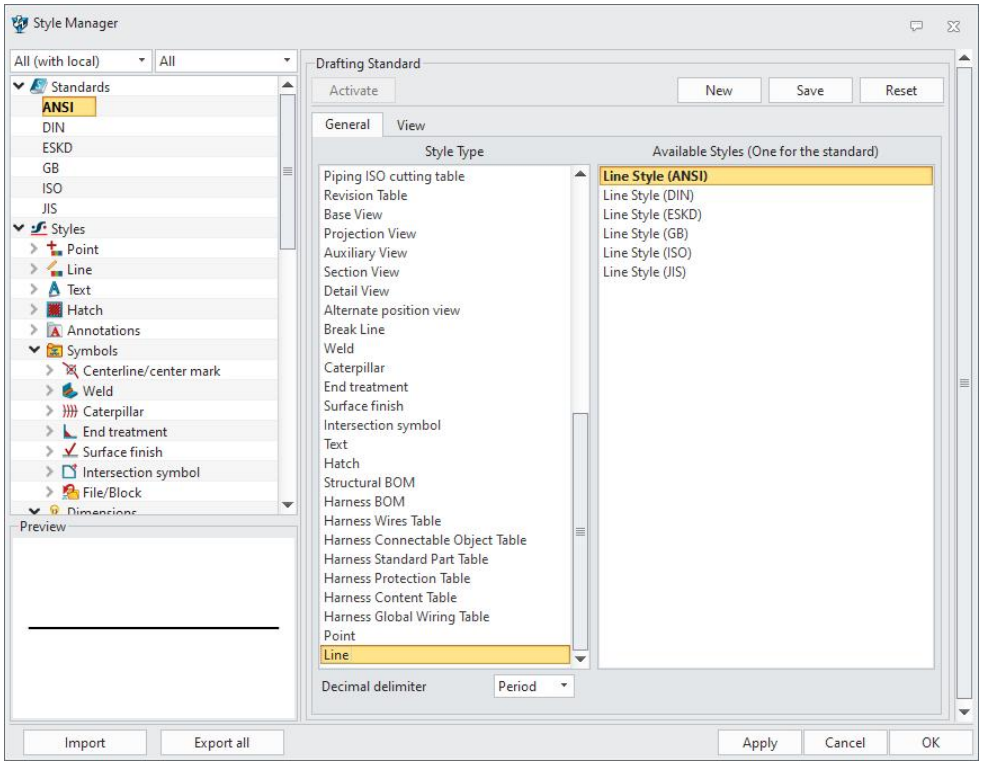
Drawing Sheet Environment >> Tool >> Style Manager >> Leader Label/**Balloon Label**

## 3.4.7.6 Point/Line Styles Controlled by Standards in the Style Manager

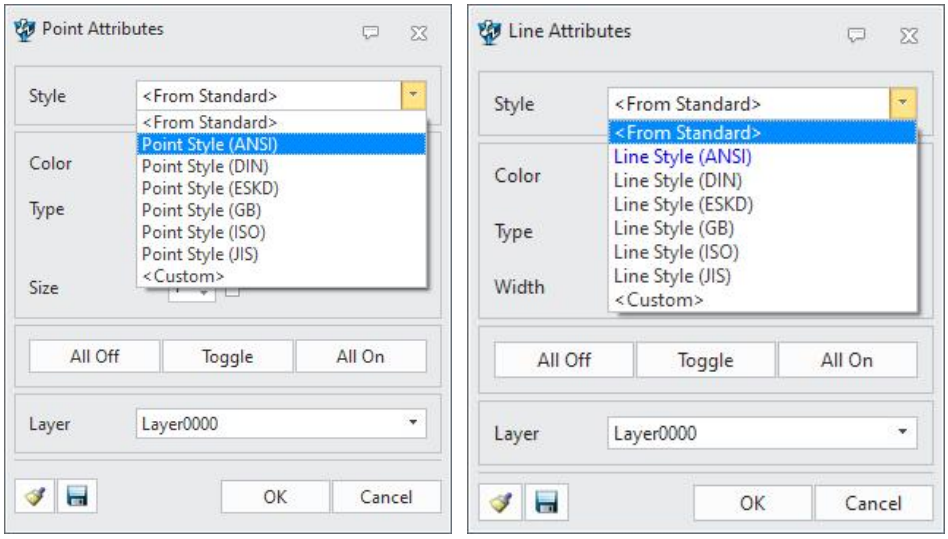
The range of objects controlled by standards in the drawing sheet style manager has been optimized, including both point and line objects within the standard style control range.

**【What users can do】**

✓ In the standards of the Style Manager, it supports setting styles for both point and line objects.




✓ In the attribute interface of points and lines, standard styles can be selected.

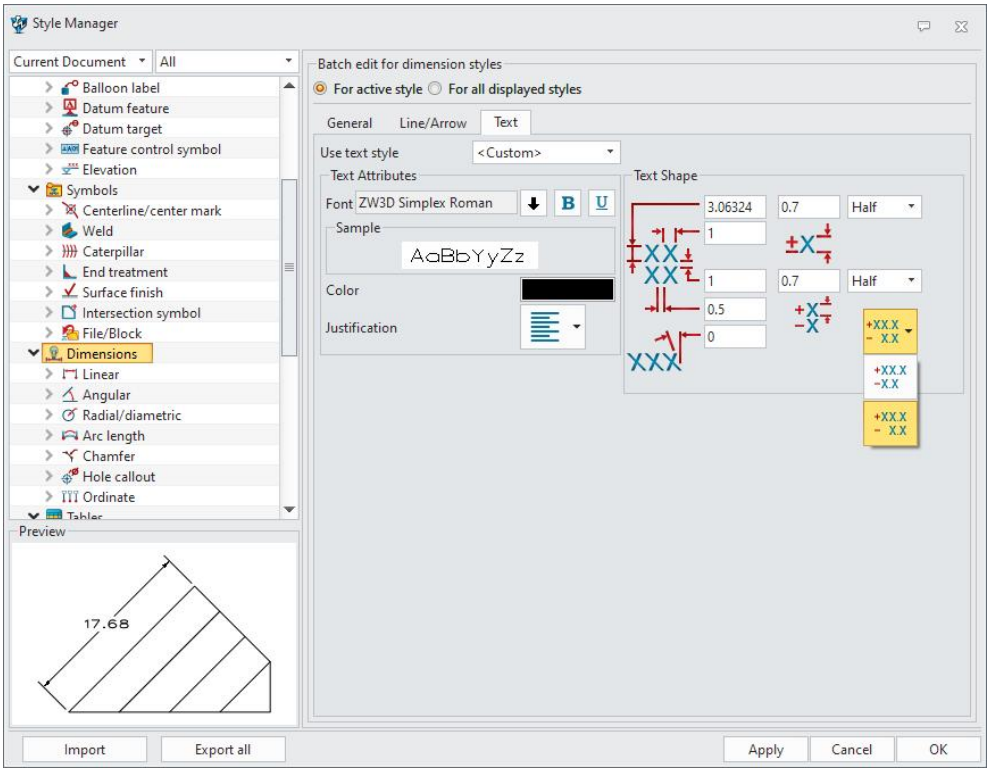


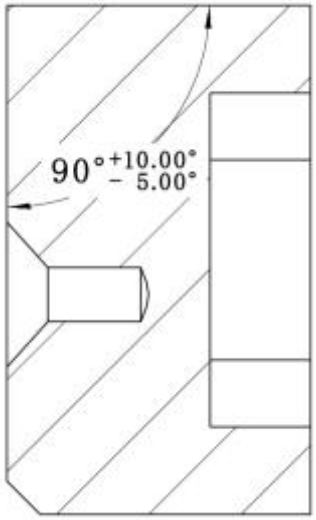

**【Where is it】**

Drawing Sheet Environment >> Tool >> Attributes >> **Style Manager**

## 3.4.7.7 New Decimal Point Alignment

New style "Decimal Point Alignment" has been added in the dimension tolerance. Tolerance alignment style can be customized in the style manager, to achieve alignment with the decimal point of tolerance.




**【Where is it】**

Drawing Sheet Environment >> Tool >> Attributes >> **Style Manager**

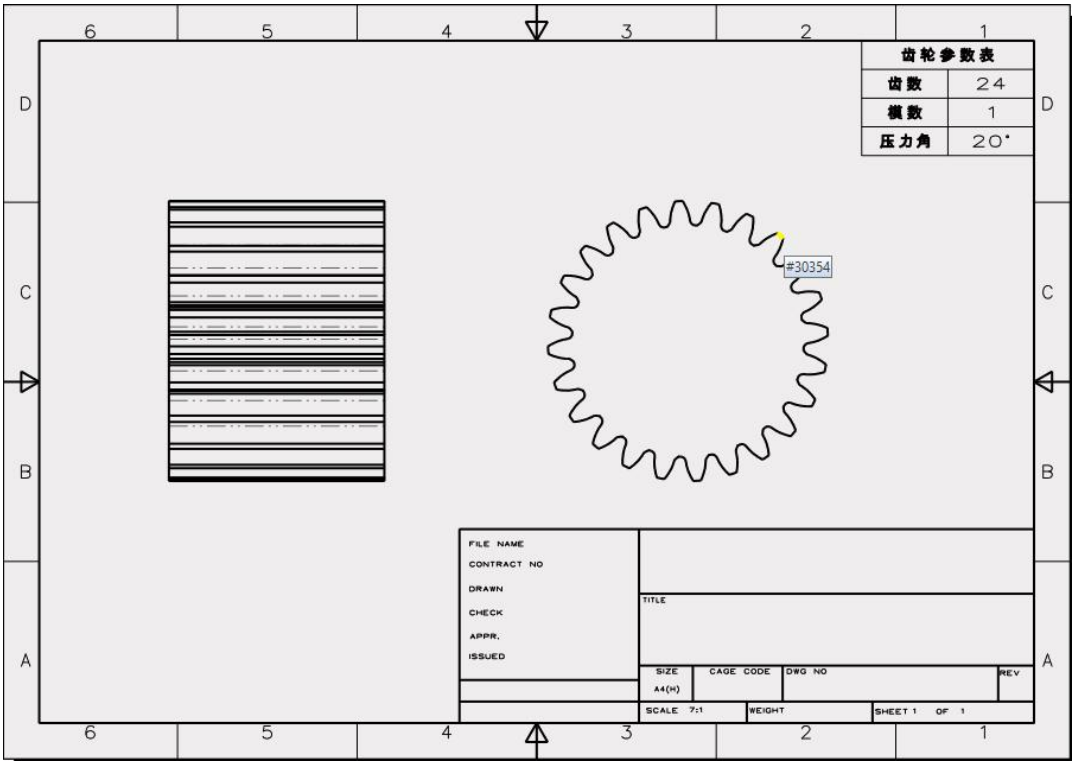
## 3.4.8 New Parametric

In industrial applications, it is often necessary to add some parameter information to the drawing

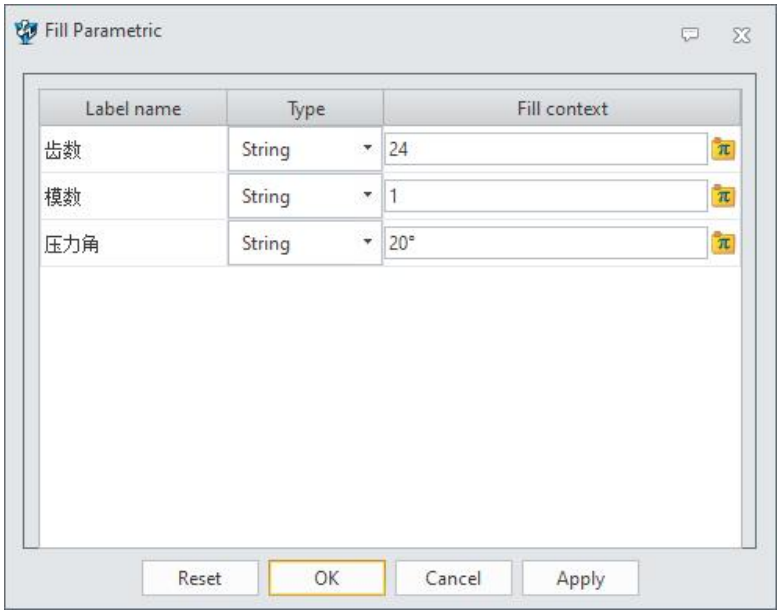
sheet drawings, such as gear parameter table, technical parameter table, etc., and ZW3D 2026 new parametric of the drawing sheet to meet this scenario.

**【What users can do】**

- ✓ Insert the parameter table in the drawing to support the calling of the expression.




- ✓ After inserting the parameter table, support filling the contents of the table.




✓ Drawing format attributes and batch modification attributes page synchronous add parametric.

**【Where is it】**

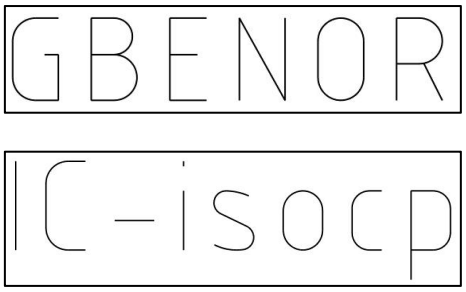
Drawing Sheet Environment >> Manager >> Right-click menu >> **Add Parametric**

## 3.4.9 Single Line Fonts "GBENOR. SHX" and "IC isoscp. shx"

Two new single line fonts "GBENOR.SHX" and "IC-isocp.shx" has been added.

**【What users can do】**

✓ In the drawing sheet, support the creation of font styles for "GBENOR.SHX" and "IC-isocp.shx".




**【Where is it】**

Drawing Sheet Environment >> Drawings >> **Text**

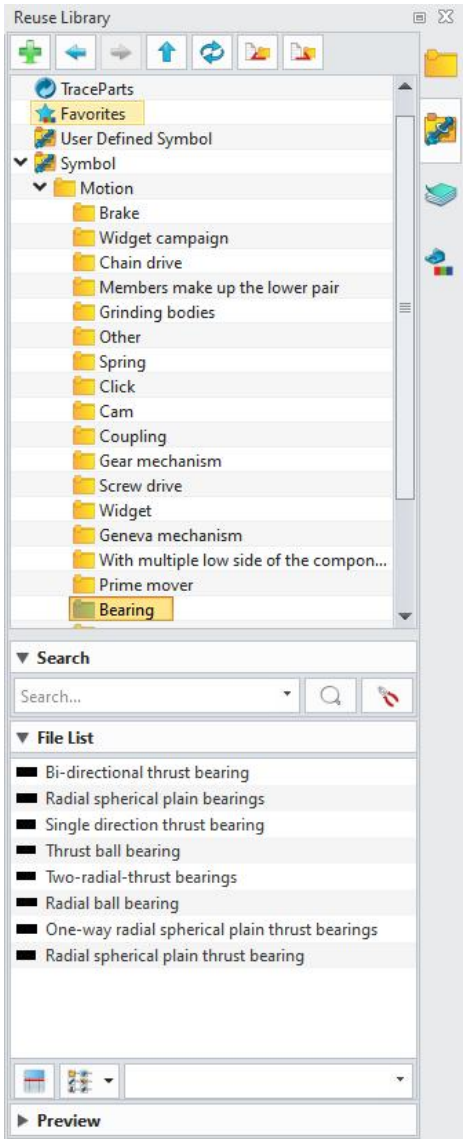
## 3.4.10 New Symbol Library

A symbol library has been added in the drawing sheet component, through which various symbols can be quickly reused,

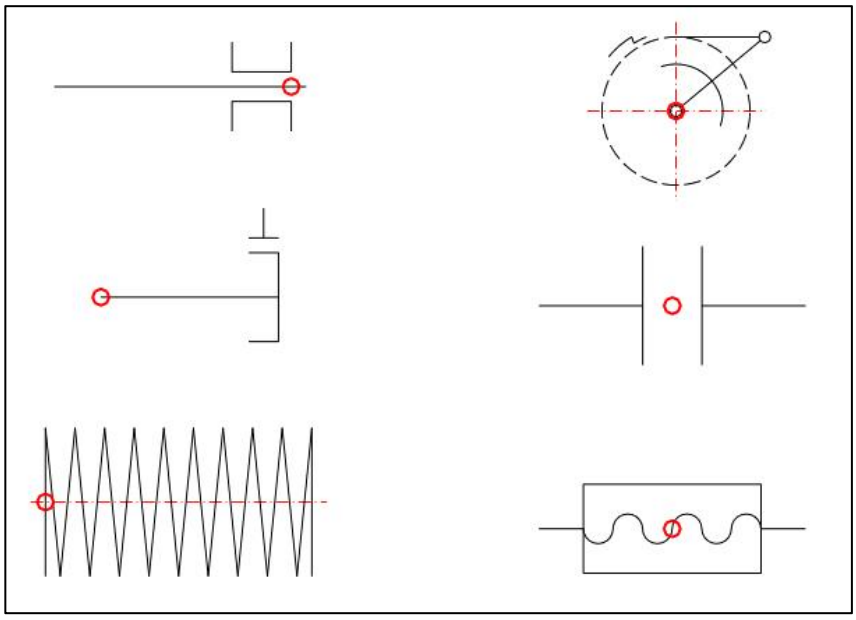
**【What users can do】**

✓ Support the annotation, balloon, feature control, datum feature, weld symbol, surface finish saves to the symbol library.

✓ Supports reading the symbols in the DWG format.



✓ Support quick insertion of symbols into the drawings through symbol library.



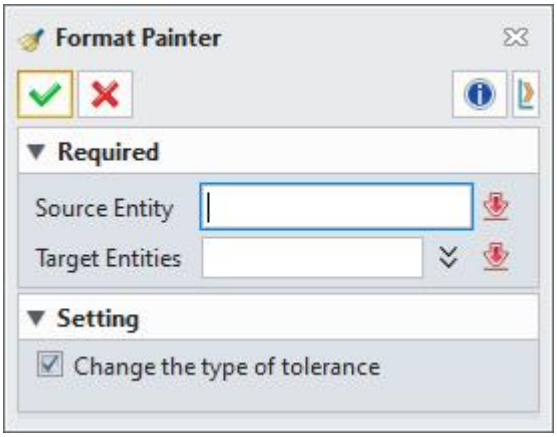
【Where is it】

Drawing Sheet Environment >> Reuse Library >> **Symbol Library**

## 3.4.11 Format Painter Optimization

Function "Change the type of tolerance" has been added in the format painter. Through checking this option, you will be able to control whether to paste tolerance information or not.

ZW3D 2026 has added change the type of tolerance function in the format painter. Through checking this option, you will be able to control whether to paste tolerance information or not.



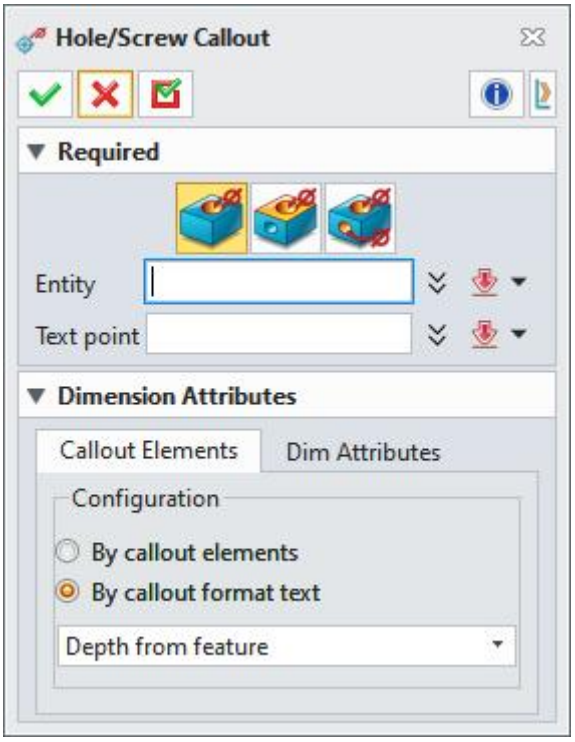
【Where is it】

Drawing Sheet Environment >> Tool >> Attributes >> **Format Painter**

## 3.5 PMI

### 3.5.1 Hole/Thread Labeling Supporting Text Labeling According to Labeling Format.

PMI hole/Screw Callout supports counting the number of the same feature, the same surface, and the same individual holes, and supports the dimension method according to the callout format text. The function usage method is consistent with the environmental hole callout function of the drawing sheet.




【What users can do】

✓ Support counting the number of the same feature or the same surface or the same hole in the same individual.

✓ Share the same hole labeling configuration file with the drawing.

✓ Support custom hole labeling format.

**【Example】**

1) You can change the HoleCallFormat.txt of the supp folder under the installation directory to customize the annotation format of various hole types.

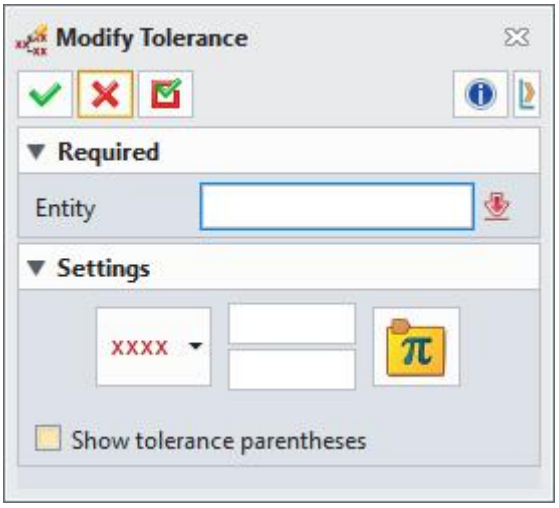
2) Select the corresponding hole marking format text in the drawing sheet to mark the required hole marking results.

**【Where is it】**

PMI > > **Hole/Screw Callout**

## 3.5.2 Modify Tolerance

Function "Modify Tolerance" has been added in the PMI component, which bahaves in the same way as the function in the drawing component.

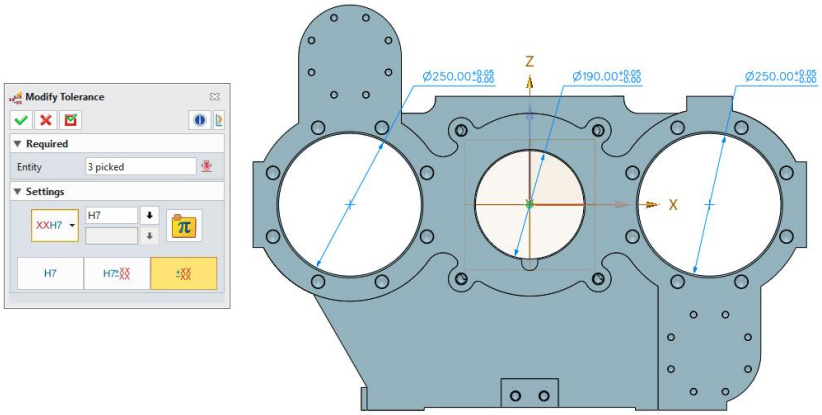



**【What users can do】**

✓ Support selecting multiple PMI for tolerance modification.

**【Example】**

As shown in the figure below, you can select multiple diameter dimensions and modify the tolerance of diameter dimensions in batches.

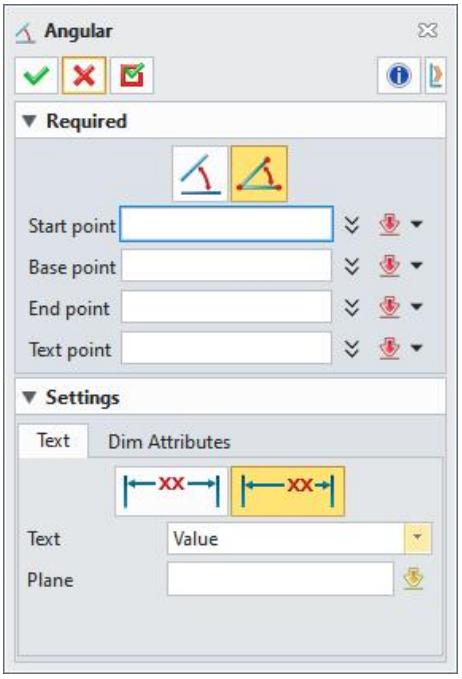



**【Where is it】**

PMI > > **Modify Tolerance**

## 3.5.3 New 3 Point Angular Marking Function.

Function "3 point angular marking" has been added to PMI component. whose usage is consistent with the use of the 3 point angle marking function in the drawing sheet environment.

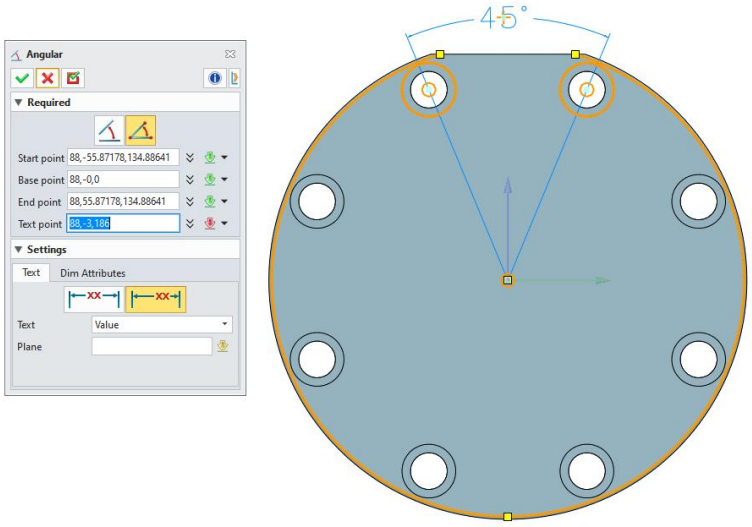



**【What users can do】**

- ✓ Support selection of starting point, base point and end point to create angle dimension.

【Example】

Support selection of start point, base point and end point to create angle dimension.

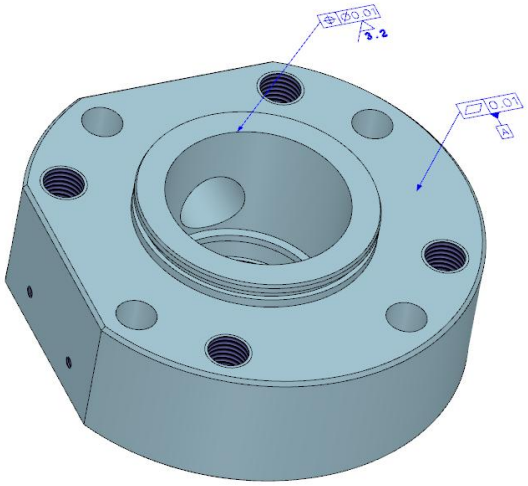



【Where is it】

PMI > > **Angle**

## 3.5.4 Symbol Adsorption Effect Optimization

Absorption of feature control to dimension has been supported, so as to the absorption of surface finish and datum feature to the feature control, whose usage is consistent with the drawing sheet environment.



**【What users can do】**

- ✓ Feature control supports snap to dimension.
- ✓ Surface finish supports adsorption to feature control.
- ✓ Datum feature supports snapping to feature control.

**【Where is it】**

PMI > > **Drawing Area**

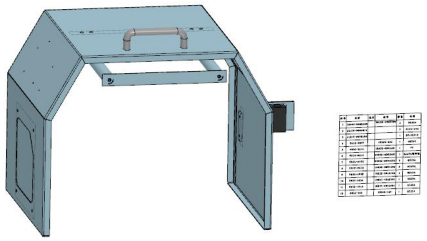
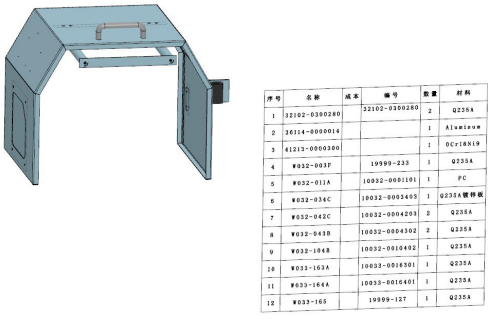
## 3.5.5 BOM Supporting Resizing

"Resize PMI" supports resizing the BOM table according to the current model scaling ratio.

**【What users can do】**

- ✓ Resize the BOM table according to the current zoom ratio.

**【Example】**

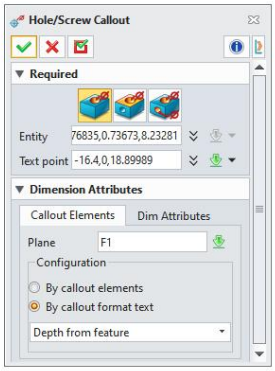
| Do not resize PMI | Resize PMI size |
| --- | --- |
|  |  |

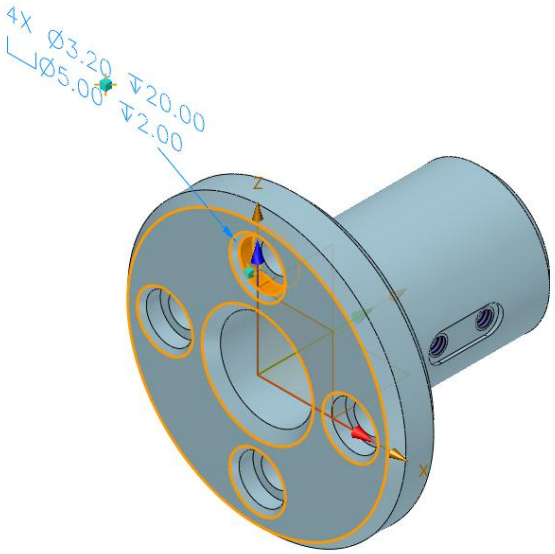
**【Where is it】**

PMI > > **Resize PMI**

## 3.5.6 Optimization of Hole/Screw Callout（ZW3D2026 SP1）

The Hole/Screw Callout of PMI has added a placement plane. ZW3D 2026 SP1 supports placing hole annotations on a specific plane.




**【What users can do】**

- ✓ Select placement plane when creating or modifying the Hole/Screw Callout of PMI.
- ✓ Right-click the Hole/Screw Callout of PMI to modify it's placement plane.

## 【Where is it】

PMI >> Dimension >> **Hole/Screw Callout**

# 4 Industry Applications

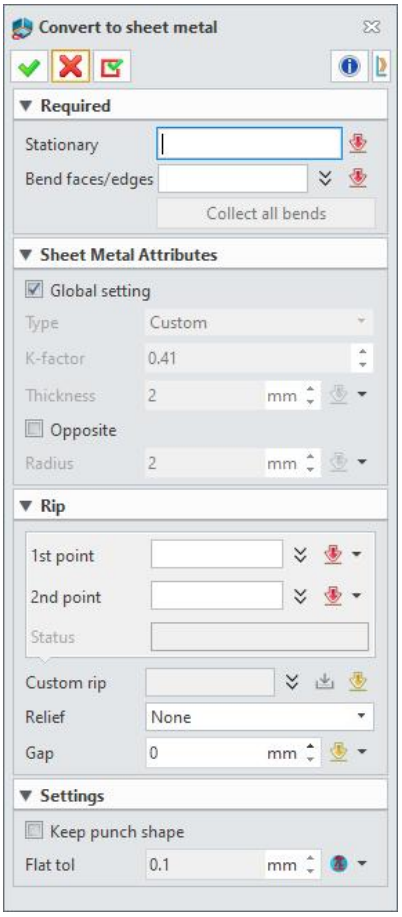
## 4.1 Sheet Metal Design

### 4.1.1 ★ Convert to Sheet Metal Improvement

The "Convert to Sheet Metal" function is improved, which allows for the convenient conversion of solids into sheet metal. This function can effectively improve the design efficiency for reverse-engineering complex sheet metal parts.

**【What users can do】**

- ✓ Support selecting "bend edges" and defining the bending radius corresponding to the bend edges.
- ✓ Support the automatic addition of rip and custom rip.
- ✓ Support defining the types and parameters of close corner
- ✓ Support retaining punch shape for conversion

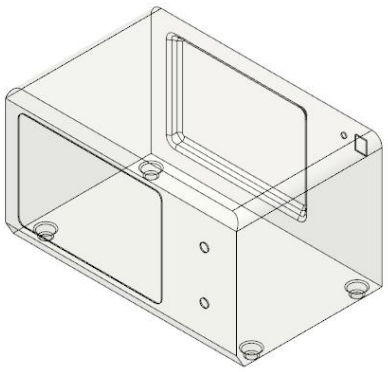
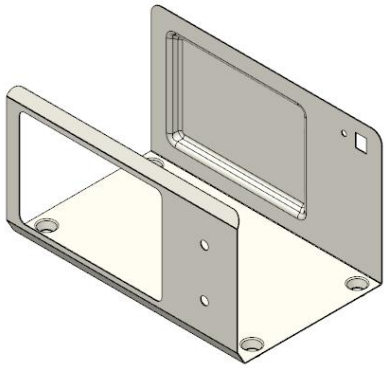



**【Notes】**

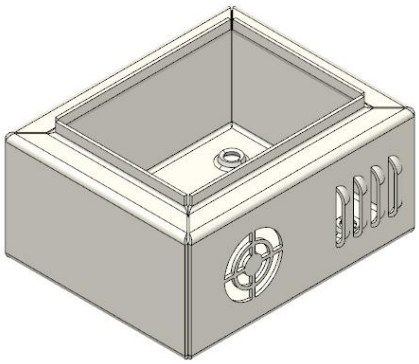
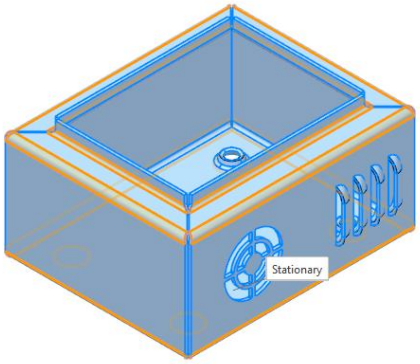
- The required option “Collect all bends” only supports collecting tangent bending faces and does not support collecting circular arc faces of punch shape and bending edges.
- When the sheet metal parts are unevenly thick before conversion, they will be repaired according to the set thickness value during conversion to ensure consistent sheet metal thickness

**【Example】**

1) Easy conversion of appearance, shape, and solid to sheet metal.

| Before conversion | After conversion |
| --- | --- |
|  |  |

2) Fast conversion of third-party format sheet metal parts and preservation of punch shape.

| Before conversion | After conversion |
| --- | --- |
|  |  |

**【Where is it】**

Sheet Metal Environment >> Convert >> Convert to sheet metal

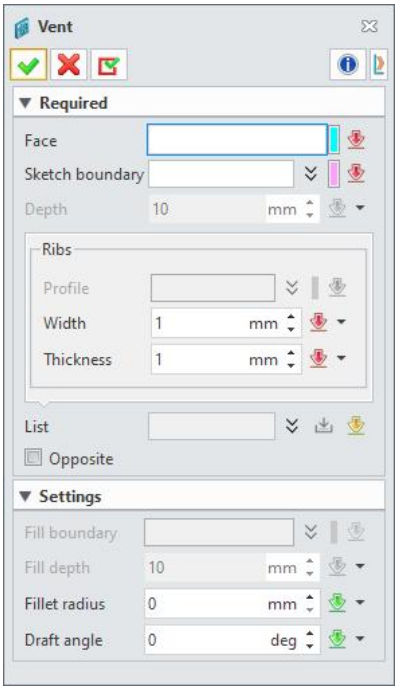
## 4.1.2 New Vent

The "vent " function can quickly create sheet metal heat dissipation structures for electrical

enclosures. This function supports defining the boundaries and ribs of ventilation ports using sketch contours, setting different parameter values for ribs, and also supports setting draft and fillet values. Facilitating design modifications, effectively reducing the number of command operations, and improving design efficiency.

**【What users can do】**

- ✓ Support sketch definition of ventilation areas, rib contours, and filling boundary.
- ✓ Support setting parameter values such as width and thickness for multiple sets of ribs.
- ✓ Support side draft and fillet for ventilation openings

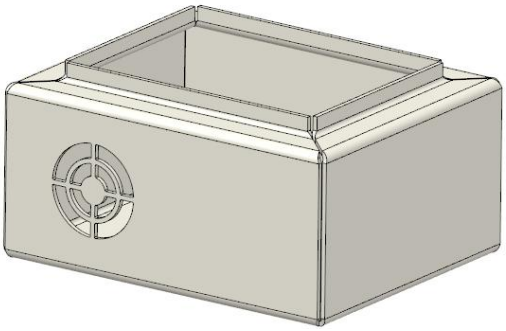



**【Notes】**

- The boundaries and ribs in the vent command are limited to the same sketch for selection, and multiple sketch definitions are not supported.

**【Example】**

Sheet metal box ventilation vent heat dissipation structure.

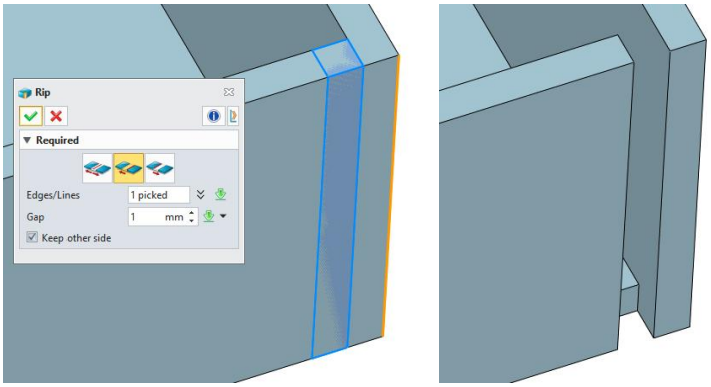


【Where is it】

Sheet Metal Environment >> Form >> **Vent**

## 4.1.3 Optimization of the Rip（ZW3D2026 SP1）

The Rip supports the removal of the other side of the wall thickness.




**【What users can do】**

✓ In "left side" and "right side" mode, the wall thickness of the other side can be retained by checking the "Keep other side" option.

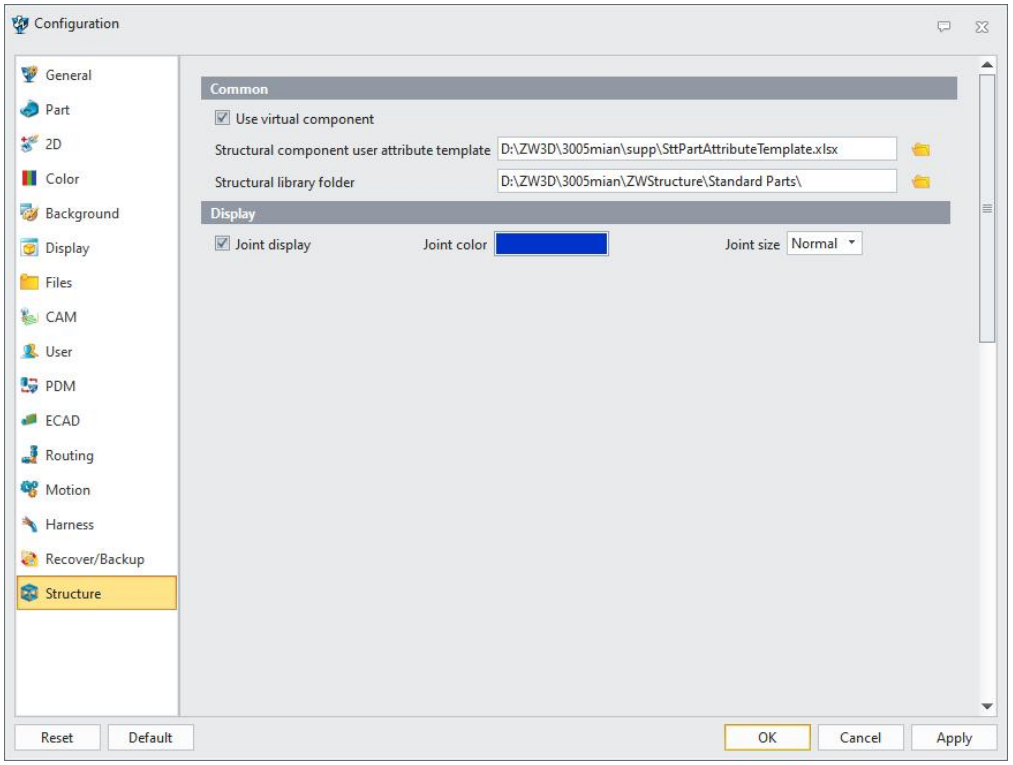
【Where is it】

Sheet Metal >> Convert >> **Rip**

## 4.2 Structure

### 4.2.1 New Structure Configuration （ZW3D 2026 SP1）

Newly added Structural Component Configuration Item allows setting configurations such as profile creation methods, structural component user attribute templates, structural libraries, and joint display.




**【What users can do】**

✓ Control the profile generation method, specifically, whether to generate in the form of virtual components.

✓ Specify user attribute files for structural component.

✓ Specify structural component profile libraries.

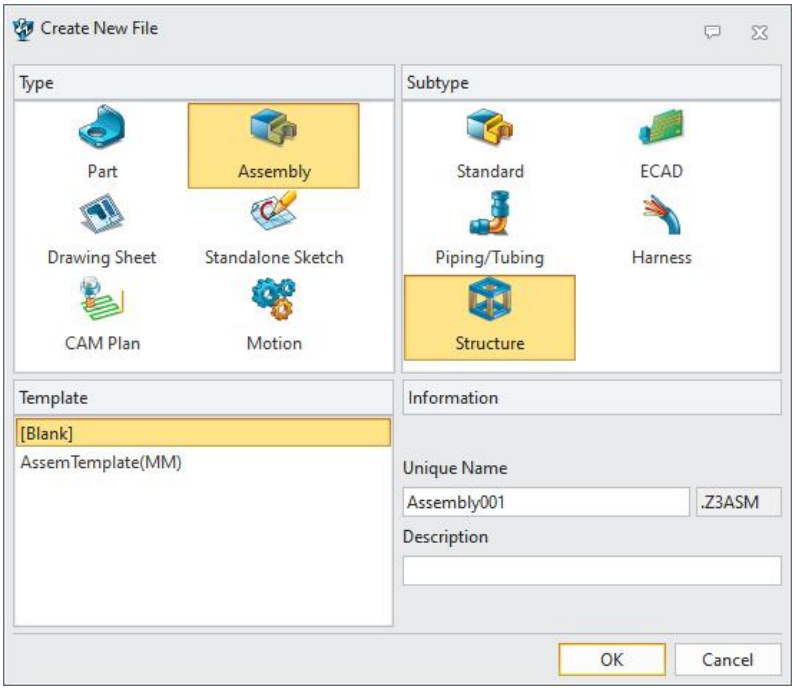
✓ Modify whether joints are displayed in the drawing area, as well as their display color and size.

**【Where is it】**

Utilities >> Configuration>> **Structure**

### 4.2.2 New Structural Component Design Entrance （ZW3D 2026 SP1）

When creating a file, you can select "Assembly >> Structure" to complete the file creation. Once the file is created, the software will directly enter the Structural Component Design Module, allowing you to start the next step of structural component model design.




**【What users can do】**

- ✓ Quick access to the Structural component design module.

**【Where is it】**

File >> New>> Assembly>> **Structure**

### 4.2.3 Joint Create and Edit

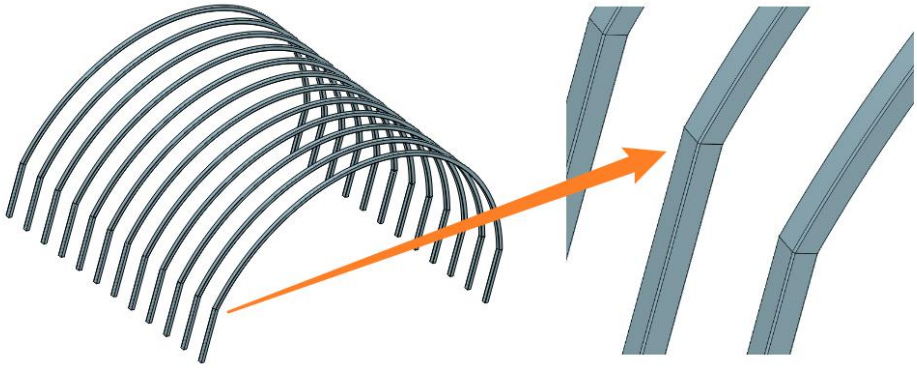
ZW3D 2026 SP1 has comprehensively upgraded and optimized the joint creation capabilities. It not only enables joint creation for curved profiles but also merges and upgrades the face trimming and trim-to-face functions. Newly add the command for the cross joint, allowing for the rapid creation of various typical design structures such as cross shapes and grid shapes. Moreover, the joint editing capabilities have been improved and optimized. It not only supports direct selection of a single joint for

parameter modification but also allows selection of multiple joints for batch modification, significantly enhancing the design efficiency of designers.

### 4.2.3.1 Joint Creation （ZW3D 2026 SP1）

#### 4. 2. 3. 1. 1 Creation of Curved Profile Joints

In structural component design, curved profiles are widely used due to considerations of structural stability and aesthetics. Examples include dome structures of roofs, conveyor contour skeletons, and sofa frames. ZW3D 2026 SP1 has enhanced its joint processing capabilities, enabling the handling of joints for common curved profiles.



**【What users can do】**

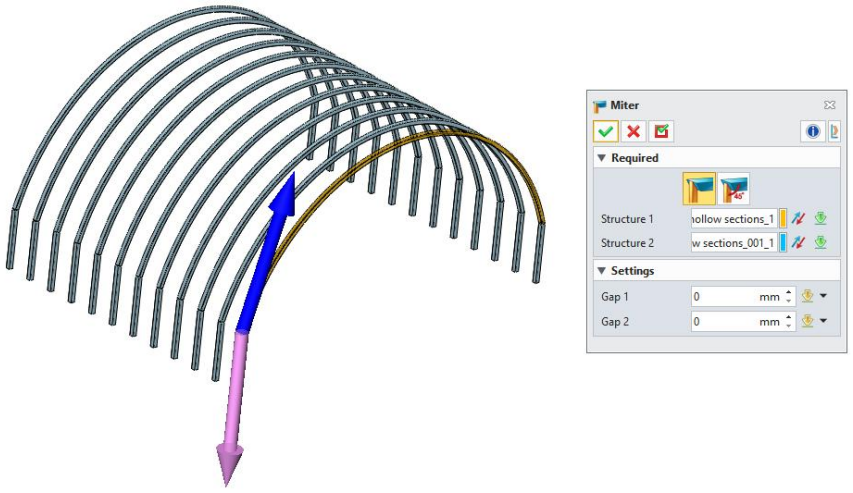
- ✓ Complete the joint creation for curved profiles.

**【Notes】**

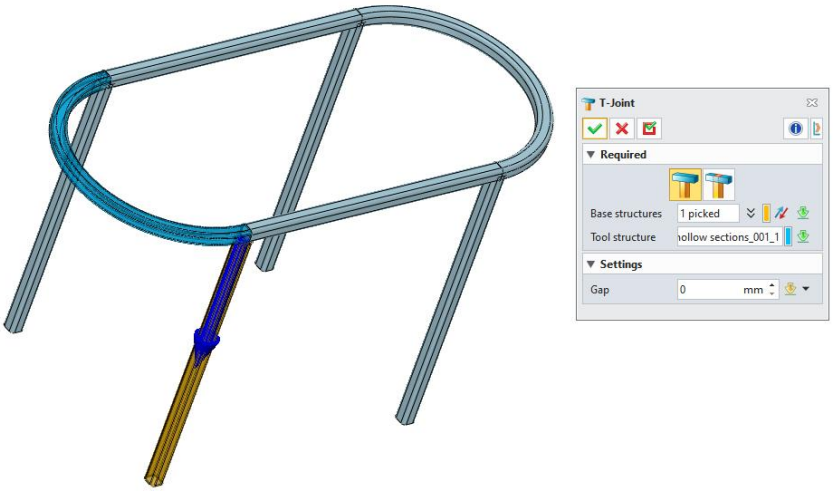
- For curved profiles to be processed, their skeleton center-lines must have exactly one intersection point to support joint creation.

**【Example】**

1) When designing the dome or shed roof structure of a roof, a miter joint can be created at the joint between the curved roof cover and the column.




2) When designing furniture products such as tables and sofas, T-joint processing can be applied to structures containing curved profiles.

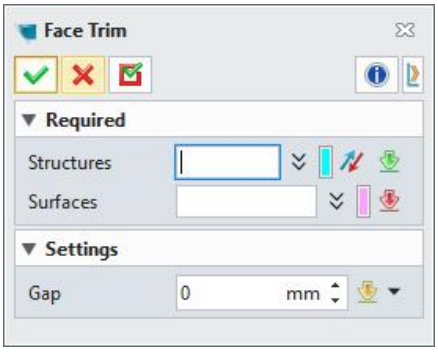
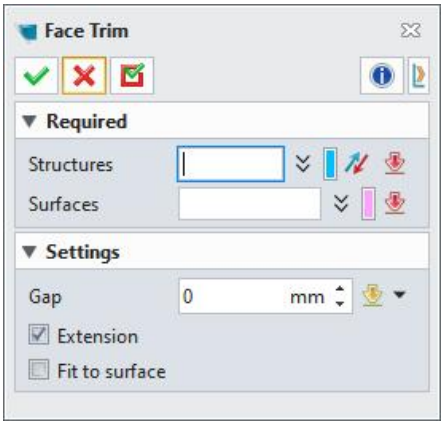



**【Where is it】**

Structure >> **Basic Joint**

### 4. 2. 3. 1. 2 Optimization of Face Trim & End To Face

ZW3D 2026 SP1 has merged and optimized the "Face Trim" and "End To Face" functions. It retains the Face Trim function and integrates End To Face, simplifying the functions in the structural component design module. You can quickly achieve both design effects through a single command.

| ZW3D 2026 | ZW3D 2026 SP1 |
| --- | --- |
|  |  |





**【What users can do】**

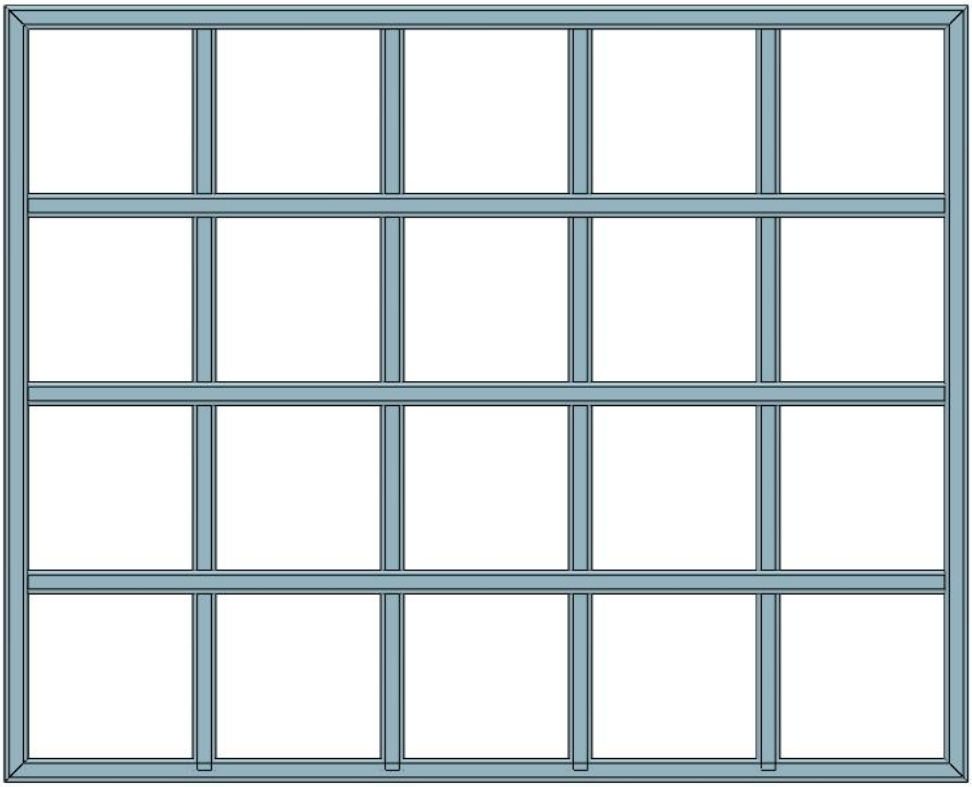
- ✓ Quickly achieve both face trimming and trim-to-face design effects through a single command.

**【Where is it】**

Structure >> Advanced Joint>>**Face Trim**

### 4. 2. 3. 1. 3 Cross Joint

In the structural design of aluminum profile frames, fences, and similar structures, structural components are widely used. These components intersect with one another. Processing the intersection joints individually would lead to extremely low work efficiency. With the Cross Joint command, multiple profile intersection positions can be processed in batches quickly, which significantly improves the user's design efficiency.



**【What users can do】**

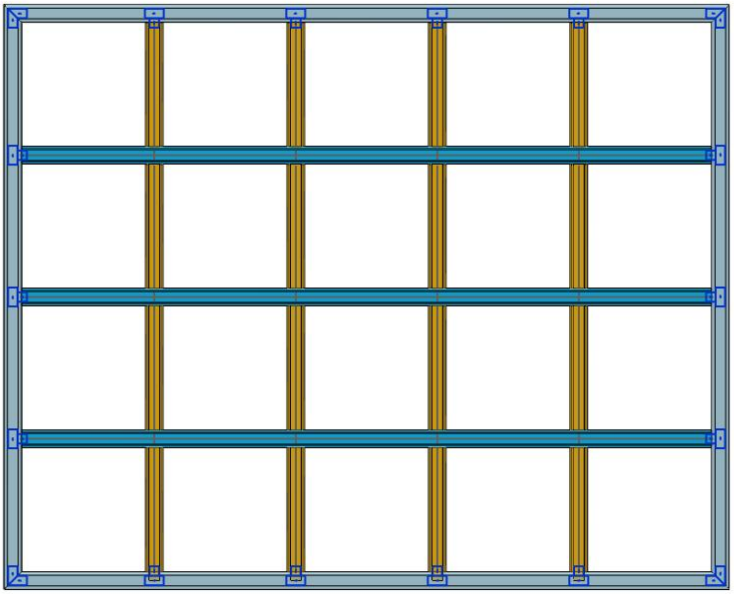
✓ Quickly complete joint processing at the intersection of two or more crossing profiles.

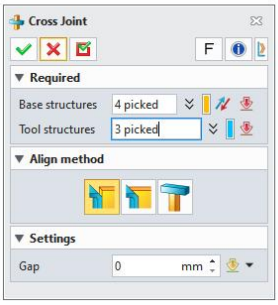
**【Notes】**

• For duplicated profiles where the skeleton curve cannot be found, cross joint processing is not supported.

**【Example】**

1) When designing the outer frame of a window, the Cross Joint command can be used to quickly complete the design of the intermediate crossing parts.






**【Where is it】**

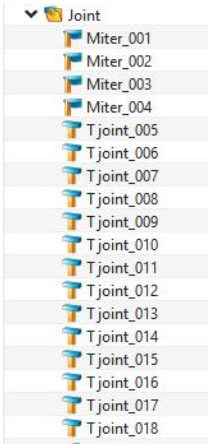
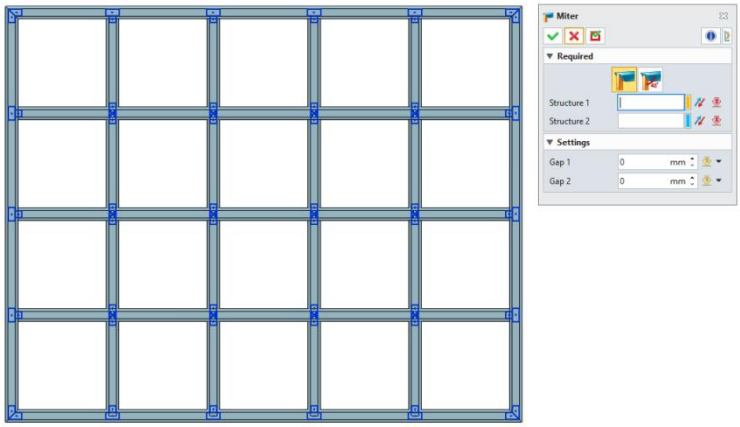
Structure >> Advanced Joint>>**Cross Joint**

## 4.2.3.2 Joint Editing

In the process of structural component design, it is common to re-edit joints. In ZW3D 2026 SP1, the joint editing method has been greatly improved. You can quickly select one or more joints for editing and modification, significantly enhancing design efficiency.

### 4. 2. 3. 2. 1 Joint Management and Visualization

Joints created by users are uniformly managed at the bottom of the Assembly Node Manager, and arranged in the "Joint Folder" in the order of their creation. In addition, when the joint command is activated, the already completed joints will be displayed with a schematic indication in the drawing area. This makes it more convenient for users to quickly determine whether joints have been created and whether the joint type is appropriate.

| Joint Nodes | Schematic Display of Joints in the Drawing Area |
| --- | --- |
|  |  |



**【What users can do】**

✓ Quickly edit corresponding joints through joint nodes.

✓ When creating a new joint, quickly determine whether a joint has already been created at the current position and whether the corresponding joint type is appropriate.

**【Notes】**

- Joints already created for the current object will only be displayed in the drawing area when the joint command is activated or when a profile is selected via the joint editing command.
- For profiles obtained through the profile reuse method, if the original profile contains joints, no new joints will be added after reuse.

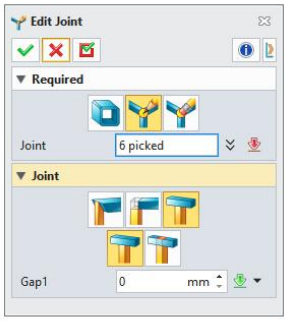
**【Where is it】**

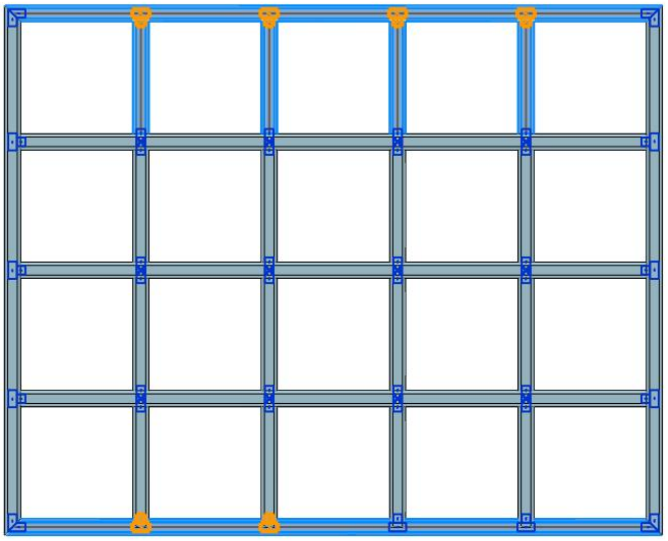
Structure >> Assembly Node Manager >>**Joint**

### 4. 2. 3. 2. 2 Optimization of Edit Joint

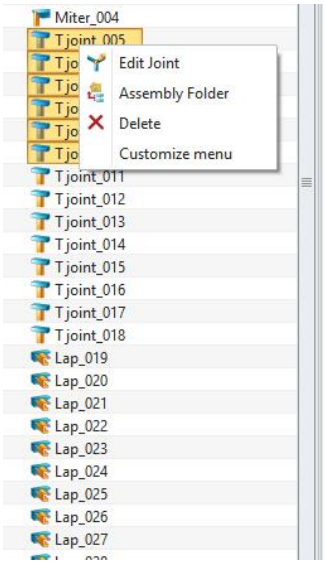
ZW3D 2026 SP supports selecting one or more joints directly for editing or deletion. You can also right-click directly on a joint node and select the edit command from the right-click menu to complete joint editing.

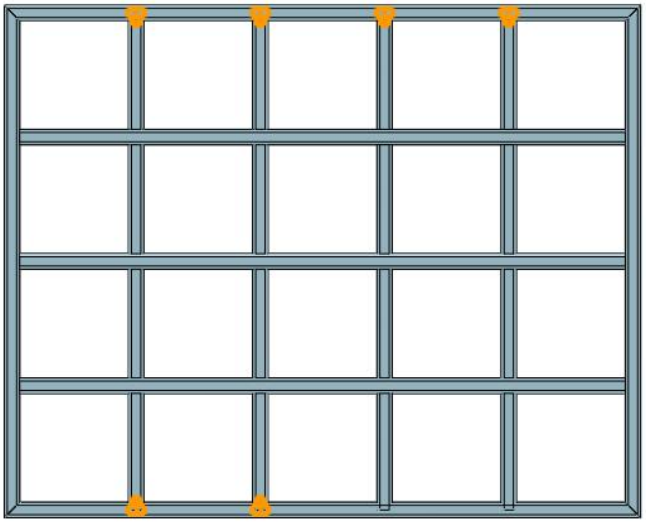
**Selecting Multiple Joints for Editing**



**Right-Click Editing of Joints**



**【What users can do】**

- ✓ Select Multiple Joints for Simultaneous Editing.
- ✓ Supports editing via the right-click menu at the joint node.

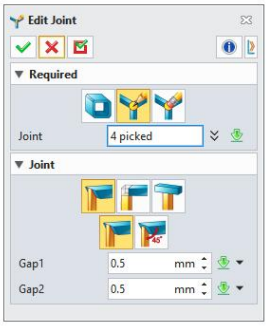
**【Notes】**

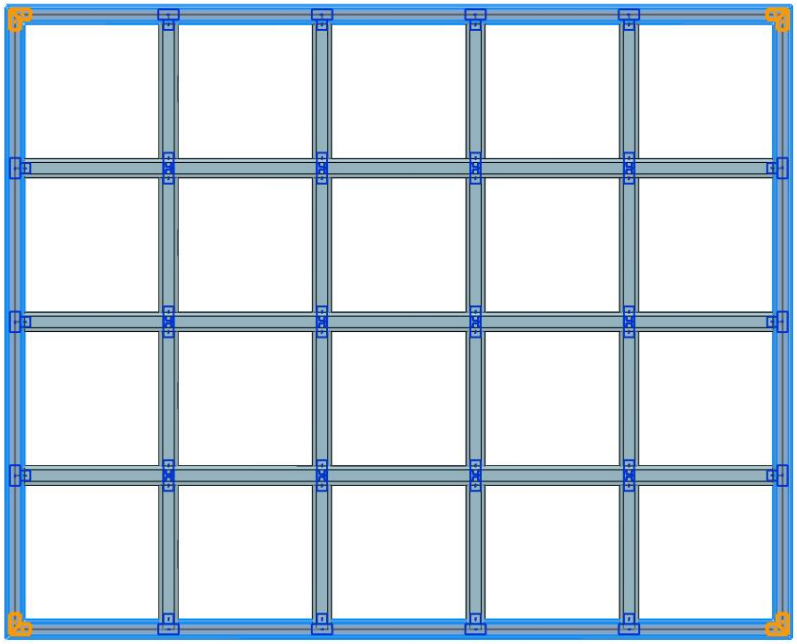
- If multiple joints of different types are selected, they can all be modified to another type.

- When selecting joints that include lap joints or face-trimmed joints, modifying their types is not supported.

**【Example】**

1) When designing window frames, batch modify the gap values of frame miters.



**【Where is it】**

Structure >> Joint Treatment >> **Edit Joint**

## 4.2.4 Structural BOM Optimizations

The usability of the structural components BOM has been improved, allowing users to more conveniently add custom attributes to structural components assemblies and customize the exclusion of components in the drawing sheet structural components BOM.

**【What users can do】**

✓ Users can exclude components that do not need to be displayed in the BOM while editing structural components BOM in the drawing sheet.

✓ In the assembly environment, users can export the current custom attributes to an Excel

spreadsheet or import custom attributes via the structural components BOM properties editing interface.

✓ In the drawing sheet, when a row in the structural components BOM is selected with the mouse, the corresponding structural component in the view will be highlighted.

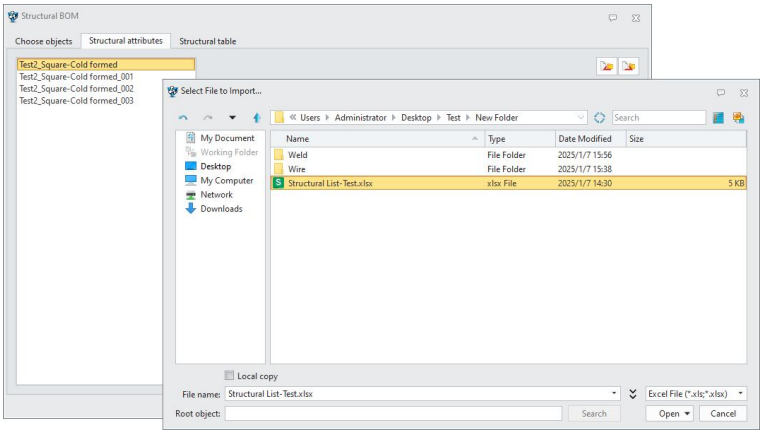
✓ In the assembly environment, users can edit the units and decimal places of numerical attributes of structural components BOM, such as length, weight, etc.

**【Notes】**

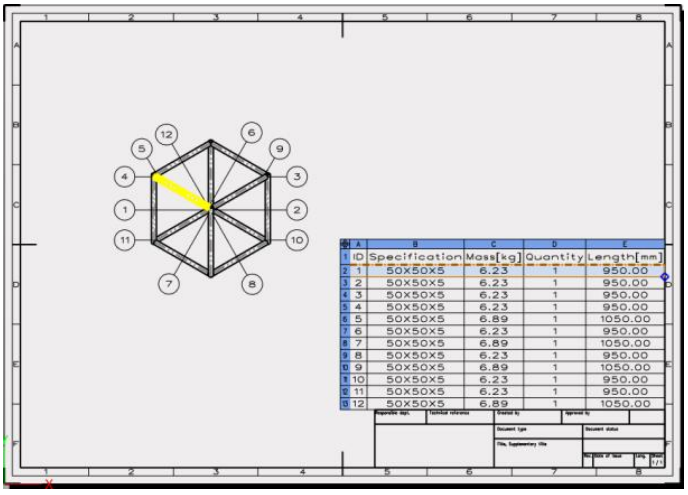
- If row merging is used in the structural components BOM in the drawing sheet, the highlighted row in the structural components BOM corresponds to the components associated with that row before merging.
- Units and decimal places can only be set in the assembly environment.

**【Example】**

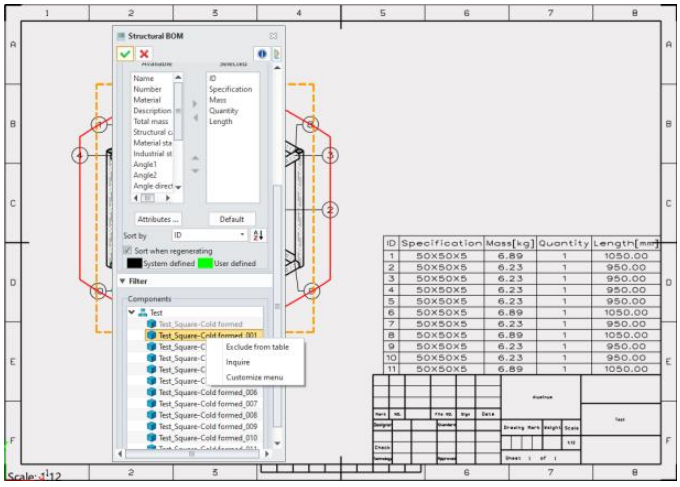
1) In the assembly environment, activate the structural components BOM, edit the structural component assembly attributes, and import or export custom user attributes using the import/export buttons on the right side.




2) In the drawing sheet, select a row in the structural components BOM, and the corresponding component will be highlighted.




3) In the drawing sheet, edit the structural components BOM to exclude components that do not need to be displayed.




**【Where is it】**

Assembly>>Structure >> **Structural BOM**

Drawing Sheet >>Table>> **Structural BOM**

## 4.2.5 ★New Reusing and Replacing Profiles

In structure design, it is very common to reuse the same profiles. The profile reuse/replace function allows users to quickly reuse the created profile components.

**【What users can do】**

✓ Support selecting a profile component and target curve, reusing this profile component at the target curve location.

✓ Support selecting a profile component and target curve, creating a new profile component with the same cross-section at the target curve location.

✓ Support selecting a profile component and target profile component, replacing the target profile component with this profile component.

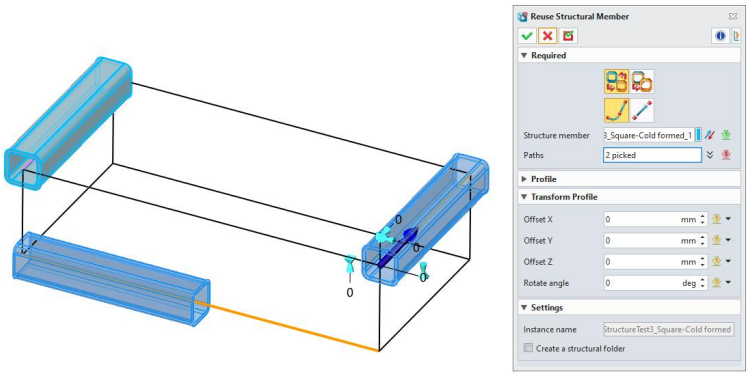
✓ Support selecting a profile component and target profile component, replacing the target profile component with a new profile component of the same cross-section.

**【Notes】**

- The target curve must be of the same type as the path referenced by the original profile component, for example, a straight line corresponds to a straight line, an arc corresponds to an arc, and the radii must be equal.
- The target profile component must be of the same type as the original profile component, for example, a straight profile corresponds to a straight profile, an arc profile corresponds to an arc profile, and the radii must be equal.
- The result after reuse/replace can be adjusted by reversing the direction.

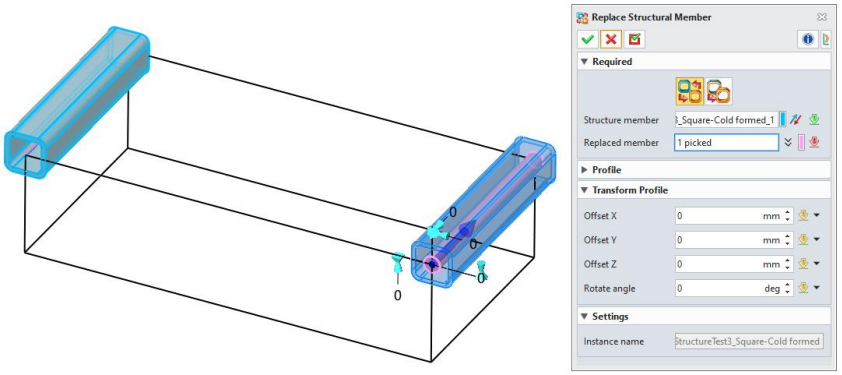
**【Example】**

1) Select a profile component and target curve, reusing this profile component at the target curve location.




2) Select a profile component and target profile component, replacing the target profile

component with this profile component.




**【Where is it】**

Assembly>>Structure>>**Reuse Structural Member**

Assembly>> Structure >>**Replace Structural Member**

## 4.3 Weldments

### 4.3.1 Groove Weld Optimizations

Single-sided V-groove and J-groove for groove welds support the generation of welds with the top surface fitting flush with the adjacent surface of the weld edge, allowing users greater freedom to control the shape of the weld entity.

**【What users can do】**

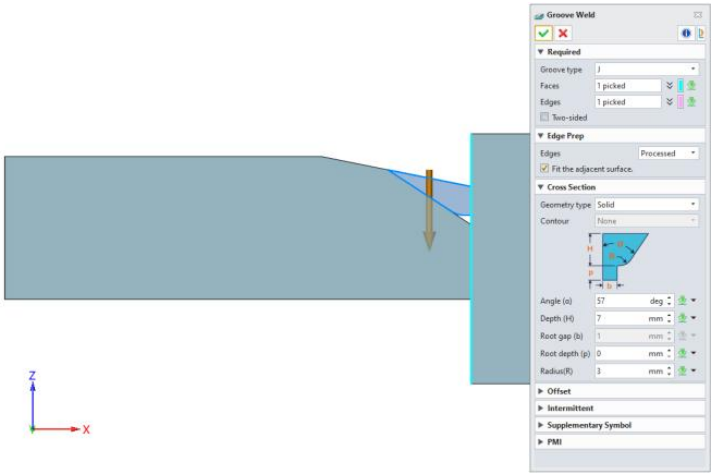
✓ Single-sided V-groove and J-groove for groove welds can be used to generate welds with the top surface fitting flush with the adjacent surface of the weld boundary.

**【Notes】**

- This option can only be used when the edge selection has been processed.

**【Example】**

By checking the "Fit the adjacent surface" option, ZW3D will automatically recognize the adjacent surface of the weld edge and cause the top surface of the weld to fit.




**【Where is it】**

Part/Assembly>>Weldments>>**Groove Weld**

## 4.4 Harness Design

### 4.4.1 Define Wiring Rules

When routing wires, there are special requirements for certain positions. For example, some positions require the wire harness to meet a certain temperature resistance level before they can be placed. In such cases, routing rules can be predefined and applied to the corresponding networks/paths, so that automatic routing can be carried out according to the rules, resulting in more accurate routing outcomes.

**【What users can do】**

- ✓ Attributes of wire harnesses can be directly defined, with rules set for individual attributes.
- ✓ Composite rules are supported, allowing multiple rules to be combined through operations.
- ✓ Wiring rules can be applied when creating paths and networks, and during automatic routing, decisions can be made based on these rules on whether to pass through regulated paths/networks.
- ✓ For rules applied to networks/paths, prompts are provided to facilitate rule modification.

**【Notes】**

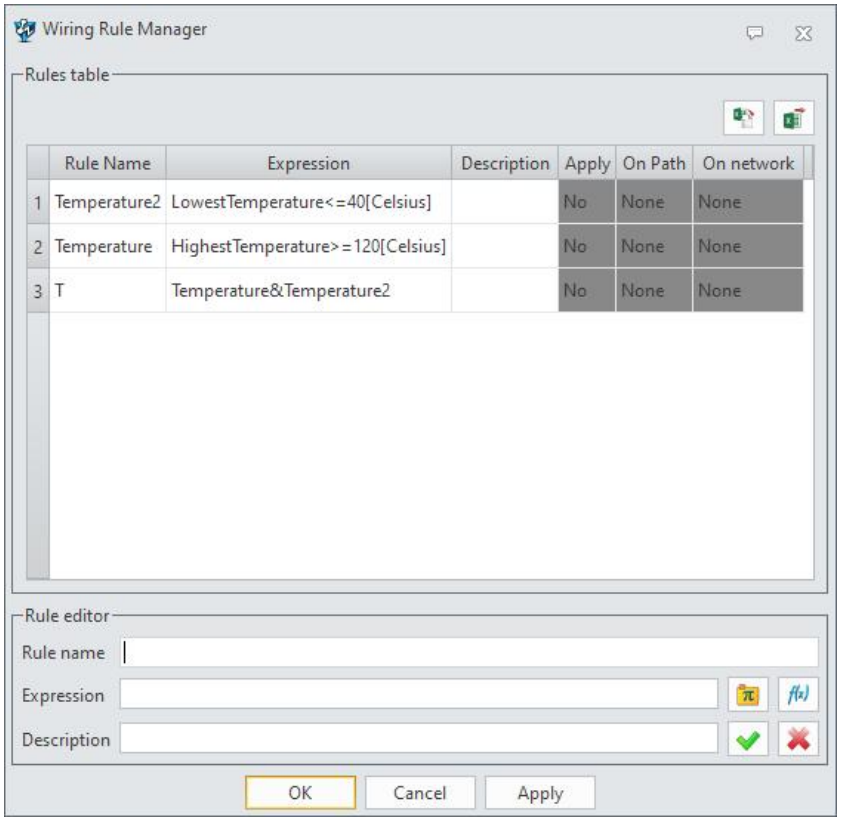
- When defining rules, it is required that the units corresponding to the attributes are correct,

and the units should be enclosed using;

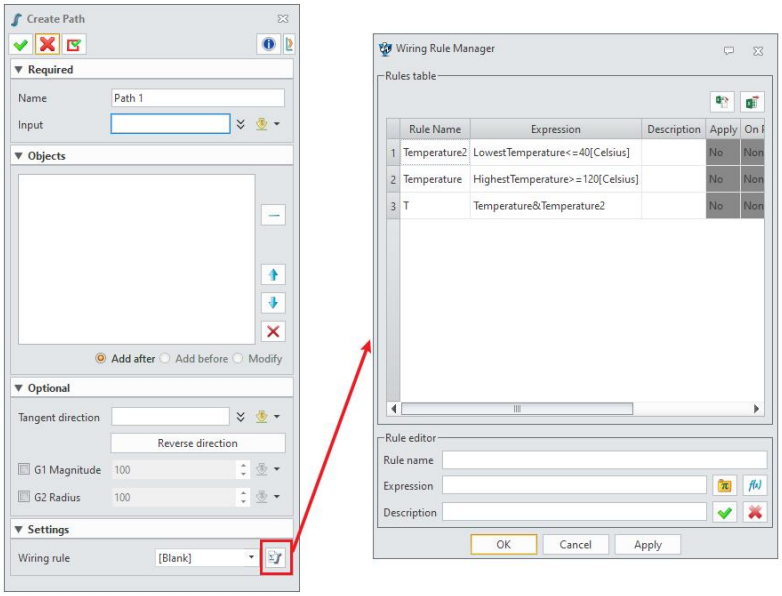
- The rules defined in the wiring harness can only be used for wiring harnesses and cannot be used for other modeling.

**【Example】**

1) Enter the wiring harness assembly environment and use the "Define Wiring Rules" function to define the rules by specifying their names, expressions, and descriptions.




2) Enter the wiring harness component environment, use the "Create Path/Create Network/Automatic Routing" function, and click the "Define Wiring Rules" button to pop up the routing rule definition panel for defining new rules。




**【Where is it】**

Wire harness assembly >> Attribute >> **Define wiring rules**

Wire harness components >> Paths and route >> **Create path/ Create network/auto route >> Define wiring rules**

## 4.4.2 ★Path and Route

### 4.4.2.1 New “Create path” Function

In order to achieve better results in automatic routing, it is necessary to draw some paths and index them during automatic routing. The wiring harness will automatically attach to the path, making the routing smoother.

**【What users can do】**

✓ In the online harness component environment, paths can be freely drawn and edited without any restrictions.

✓ The path can be automatically routed or manually routed directly, and the wire harness can achieve automatic attachment and section management on the path.

✓ Adjust the path direction, and the attached wiring harness automatically follows the path to

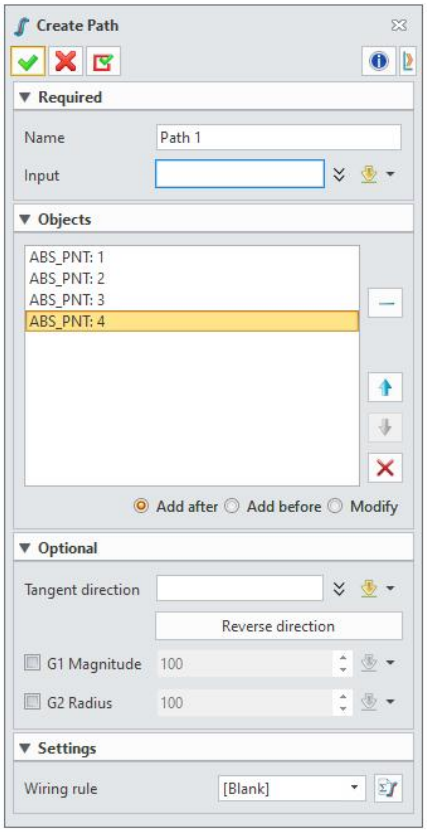
adjust the direction.

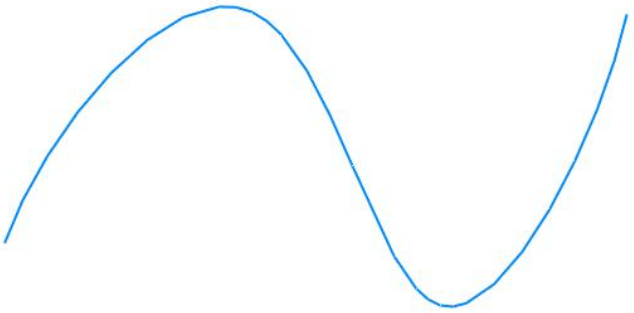
**【Notes】**

- When wiring, if the two paths used are continuous but not tangent, these two paths cannot be used for successful wiring, and only one path can be selected for wiring.

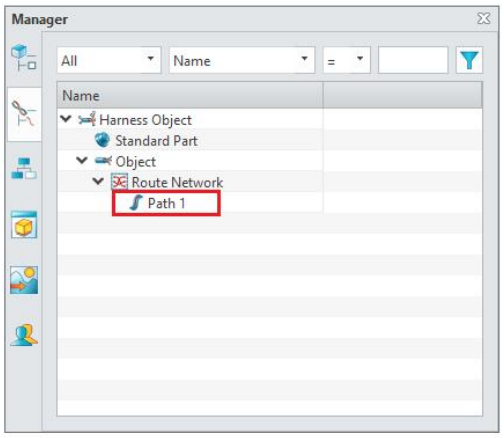
**【Example】**

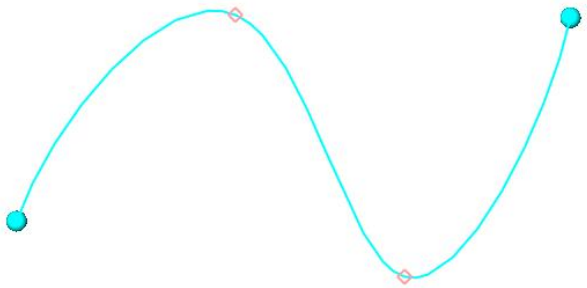
1) Enter the wiring harness component environment and use the "Create Path" function to select multiple points at any position in the drawing area and freely create paths.



2) Complete the created path, and the corresponding node information can be seen in the harness manager.



【Where is it】

Wire harness components >> Paths and route >> Create path

### 4.4.2.2 New “Extract path” Function

In order to directly use the common harness segments that have attached harnesses when wiring harnesses, reduce the use of "attach" commands, add the "extract path" function, and conveniently extract harness segments as paths, achieving quick use of routing paths and automatic attachment of harnesses.

**【What users can do】**

✓ In the wire harness component environment, any wire harness segment can be extracted as a path.

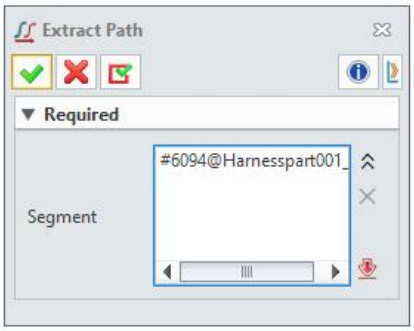
✓ The extracted path has the same function as the directly drawn path, so even if the referenced harness segment is deleted, the extracted path can still exist independently.

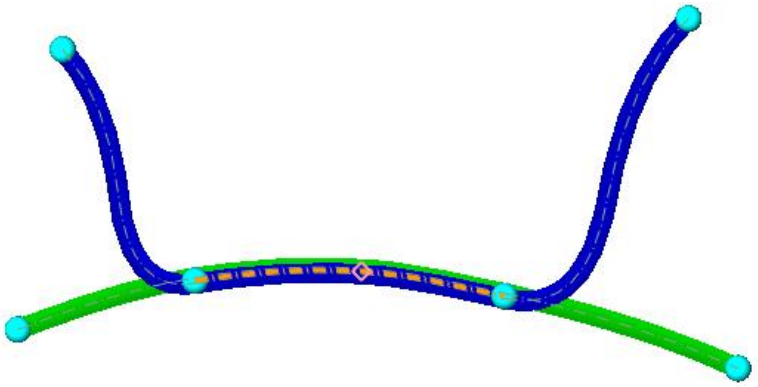
**【Notes】**

• After extracting the path, the wire harness automatically attaches to the path. When detaching, the attached wire harness segment needs to select the extracted path.

**【Example】**

Enter the wiring harness component environment and use the "Extract Path" function to select any wiring harness segment in the drawing area to extract it as a path.



【Where is it】

Wire harness components >> Paths and route >> **Extract path**

### 4.4.2.3 New “Create network” Function

When multiple paths are used for automatic routing, the traversal speed will be slower. In order to improve the traversal speed of automatic routing, it is necessary to group the paths, so that each group of paths can be quickly traversed to the optimal routing path. Therefore, the function of creating a network has been added.

**【What users can do】**

✓ Quickly form a network from existing paths.

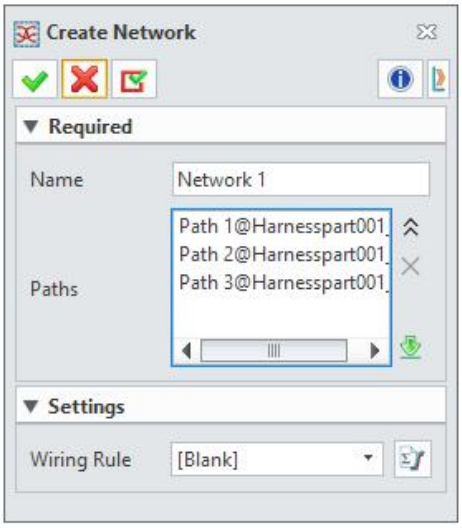
✓ You can create a pathless network and use other operations to add paths to an empty network in the future.

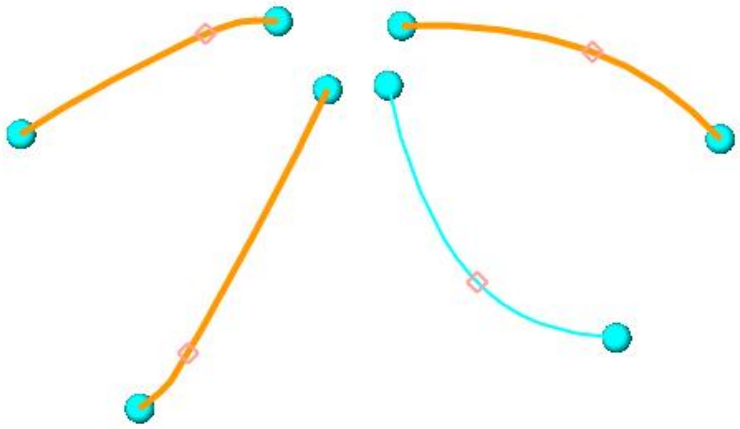
**【Notes】**

• If a network is not created or an empty network is created, paths that are not in the network will be classified as one network during automatic routing, and the speed of automatic routing will be reduced.

**【Example】**

1) Enter the wiring harness component environment and use the "Create Network" function to select any path in the drawing area, which can be collected into one network.



2) In the "Create Network" function, if you do not select a path and directly determine the generation network, you can create an empty network.

**【Where is it】**

Wire harness components >> Paths and route >> **Create network**

### 4.4.2.4 New “Add to Network” Function

To continue adding paths to an existing network, you can use the Add to Network feature to add paths to any existing network. In addition, for the convenience of operation, this effect can also be achieved directly by using the left mouse button to pull the path on the line harness manager.

**【What users can do】**

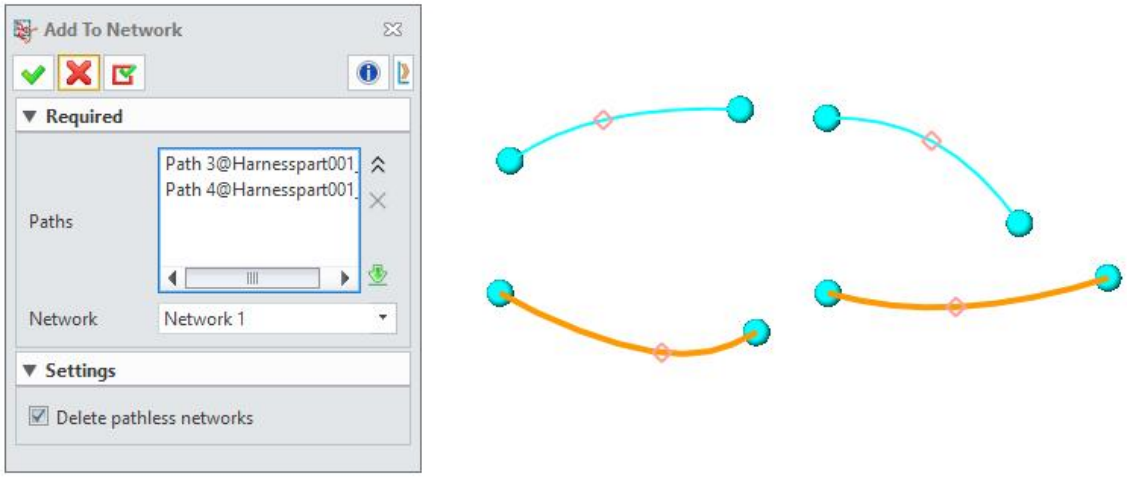
- ✓ Add paths that are not in the network to the existing network.
- ✓ Add paths from other networks to the selected network.
- ✓ For pathless networks, they can be directly deleted.

**【Notes】**

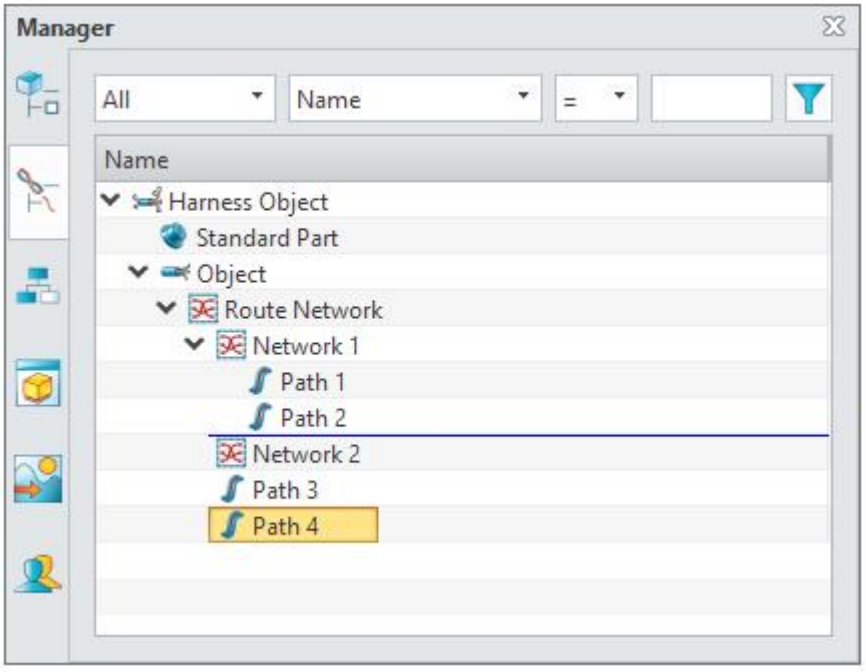
- If you do not want to delete an empty network, you need to uncheck the "Delete pathless network" option in the settings.

**【Example】**

1) Enter the wiring harness component environment and use the "Add to Network" function. You can select any path in the drawing area to add the selected path to the target network. If the "Delete Network without Path" option is checked, the empty network will be automatically deleted.




2) In addition to using this command, you can also use the left mouse button to select the target path in the bundle management, drag it to the target network, or pull the path out of the network to make it exist independently.




【Where is it】

Wire harness components >> Paths and route >> **Add to network**

### 4.4.2.5 New “Auto Route” Function

When there are a large number of wire harnesses, the efficiency of manual wiring is significantly lower. In order to improve wiring efficiency, the automatic wiring function can be used to arrange wire harnesses.

**【What users can do】**

✓ Automatically select the correct wire from the online material library for wiring based on the principle table.

✓ Automatically select the shortest path for wiring during the wiring process or directly route without using a path.

✓ Automatic verification of necessary wiring information can be performed before the start of automatic wiring.

✓ After the automatic wiring is completed, the wiring results can be displayed.

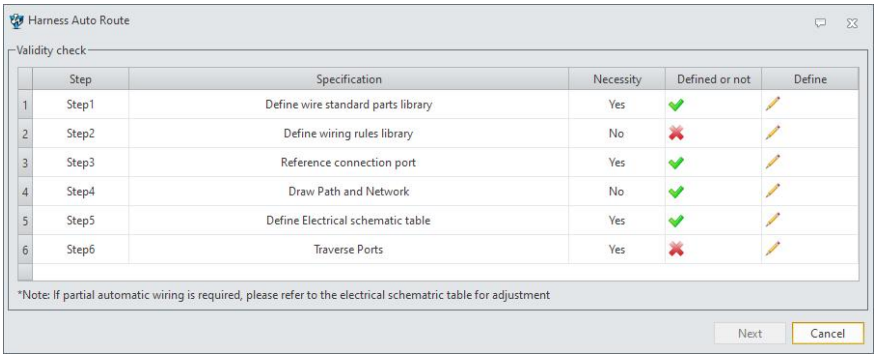
✓ Automatic wiring of designated harnesses can be done on the electrical schematic table.

**【Notes】**

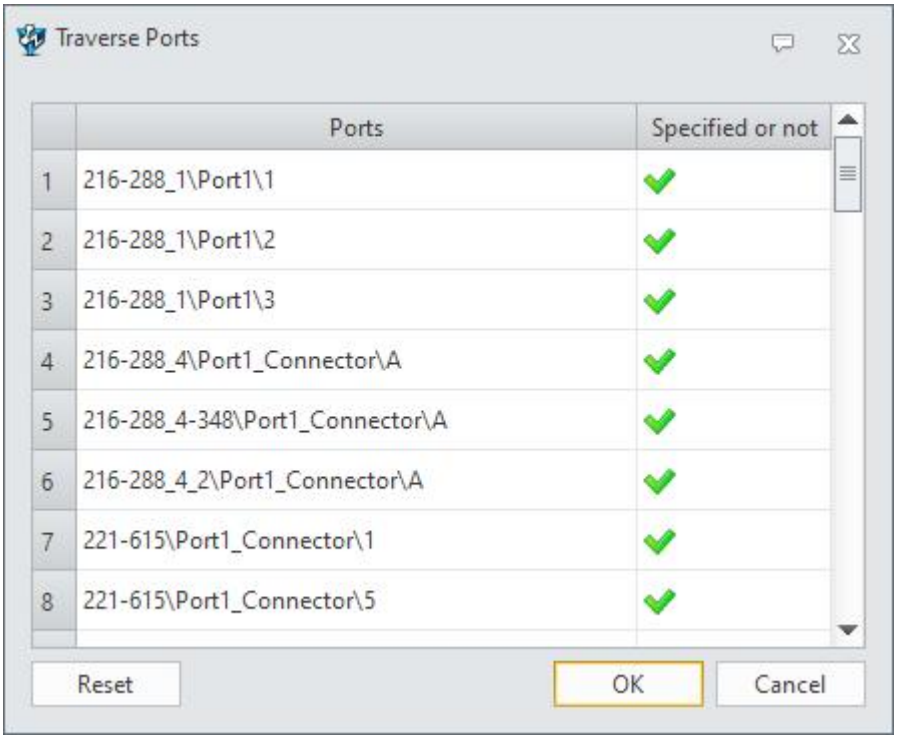
• Before automatic wiring, it is necessary to ensure the completion of connector port definition, connector port traversal, wire library definition, and electrical schematic table writing.

**【Example】**

1) Enter the wiring harness component environment and use the "automatic wiring" function to first perform a legality check on the automatic wiring.

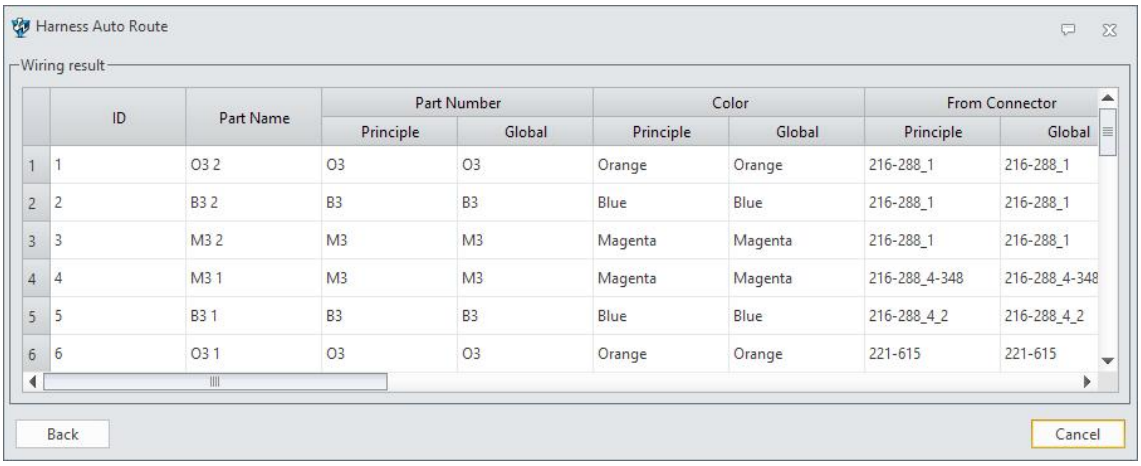



2) When the necessary items are not defined, automatic routing cannot be performed. Click the button in the definition column to enter the editing interface. For example, "Traverse Ports" can be clicked to automatically traverse ports, matching the port information in principle with the port information in this harness component one by one.

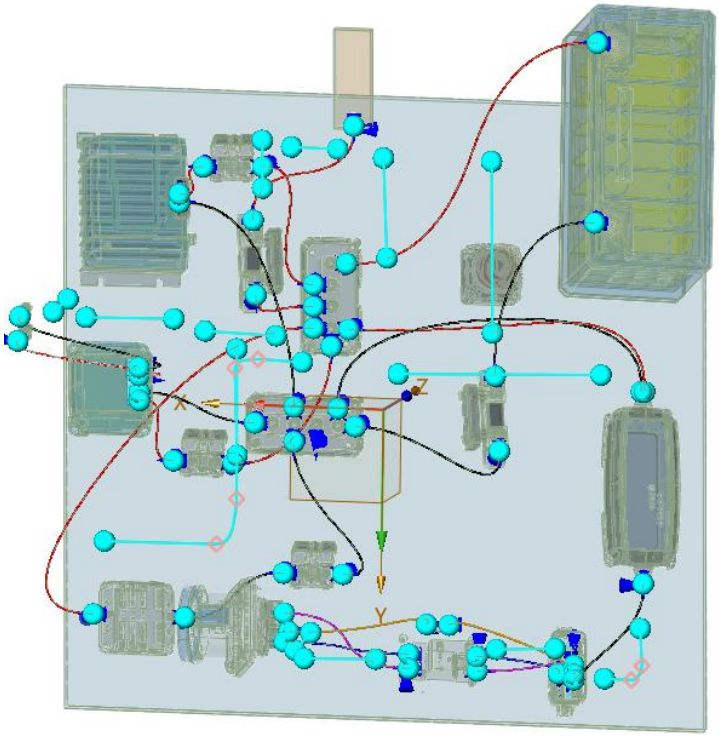



3) Click "Next" to select the wiring conditions, including whether to use paths, wiring rules, etc.

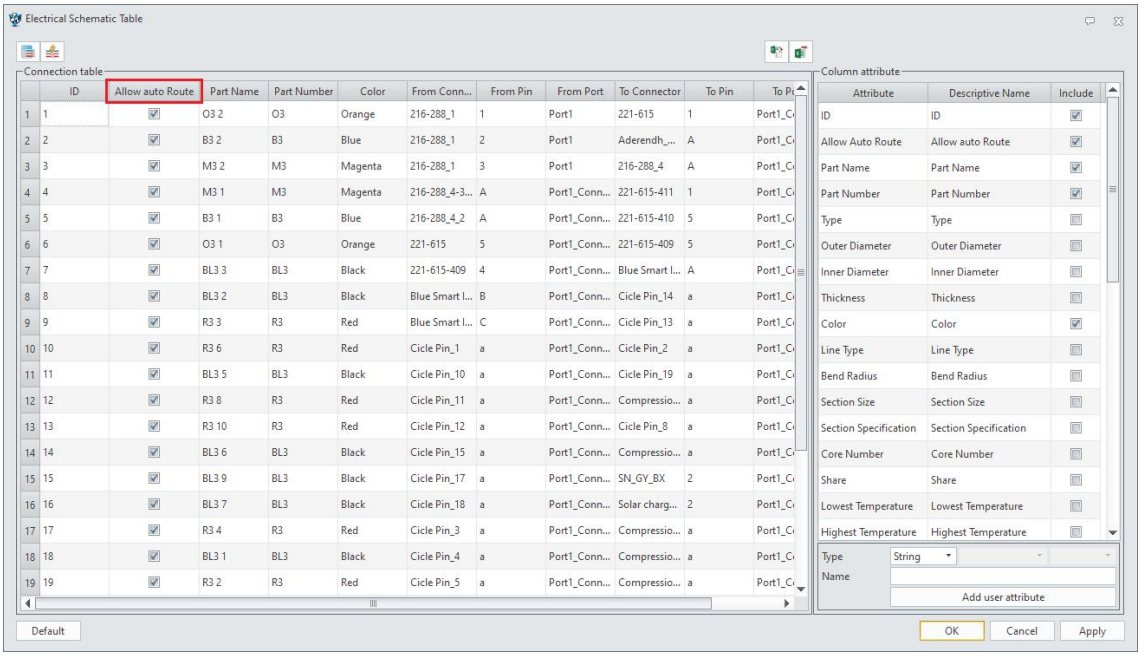
4) Click "Start Wiring" to start automatic wiring according to the wiring conditions. After wiring, enter the result interface, which displays whether all wiring harnesses have been arranged.




| | ID | Part Name | Part Number | | Color | | From Connector | |
|---|---|---|---|---|---|---|---|---|
| | | | Principle | Global | Principle | Global | Principle | Global |
| 1 | 1 | O3 2 | O3 | O3 | Orange | Orange | 216-288_1 | 216-288_1 |
| 2 | 2 | B3 2 | B3 | B3 | Blue | Blue | 216-288_1 | 216-288_1 |
| 3 | 3 | M3 2 | M3 | M3 | Magenta | Magenta | 216-288_1 | 216-288_1 |
| 4 | 4 | M3 1 | M3 | M3 | Magenta | Magenta | 216-288_4-348 | 216-288_4-348 |
| 5 | 5 | B3 1 | B3 | B3 | Blue | Blue | 216-288_4_2 | 216-288_4_2 |
| 6 | 6 | O3 1 | O3 | O3 | Orange | Orange | 221-615 | 221-615 |



5) Specific requirements for automatic wiring of wire harnesses can be specified in the electrical schematic, and port traversal and automatic wiring operations can also be performed in the electrical schematic.



【Where is it】

Wire harness components >> Paths and route >> **Auto route**

Wire harness components >> Table >> Electrical schematic table >> **Auto route**

Wire harness components >> Table >> Electrical schematic table >> **Traverse ports**

### 4.4.2.6 New "Copy Path/Network" Function

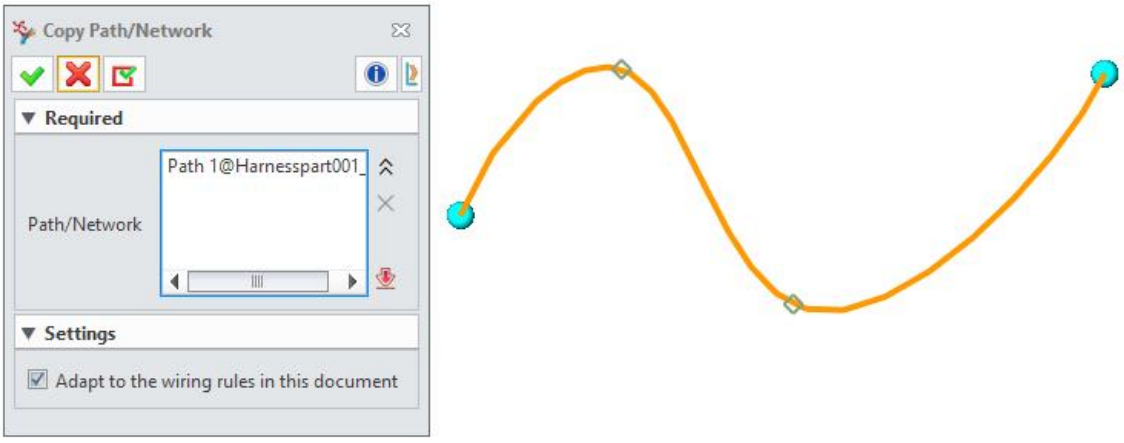
In order to facilitate the reuse of paths/networks, reduce the time required to repeatedly draw the same path, and improve modeling efficiency, the function of copying networks/paths has been added.

**【What users can do】**

- ✓ Allow copying paths/networks from other harness components in the same harness assembly.
- ✓ Allow copying paths/networks from harness components in other harness assemblies.
- ✓ Allow to choose whether to apply rules during the process of copying networks/paths.

**【Example】**

Enter the wiring harness component environment and use the "Copy Network/Path" function to select any path/network from other wiring harness component environments in the drawing area, and then copy the selected path/network to the current wiring harness component environment.




【Where is it】

Wire harness components >> Reference >> **Copy Path/Network**

## 4.4.3 Efficiencies of Harness Modeling

### 4.4.3.1 Definition of Bending Radius

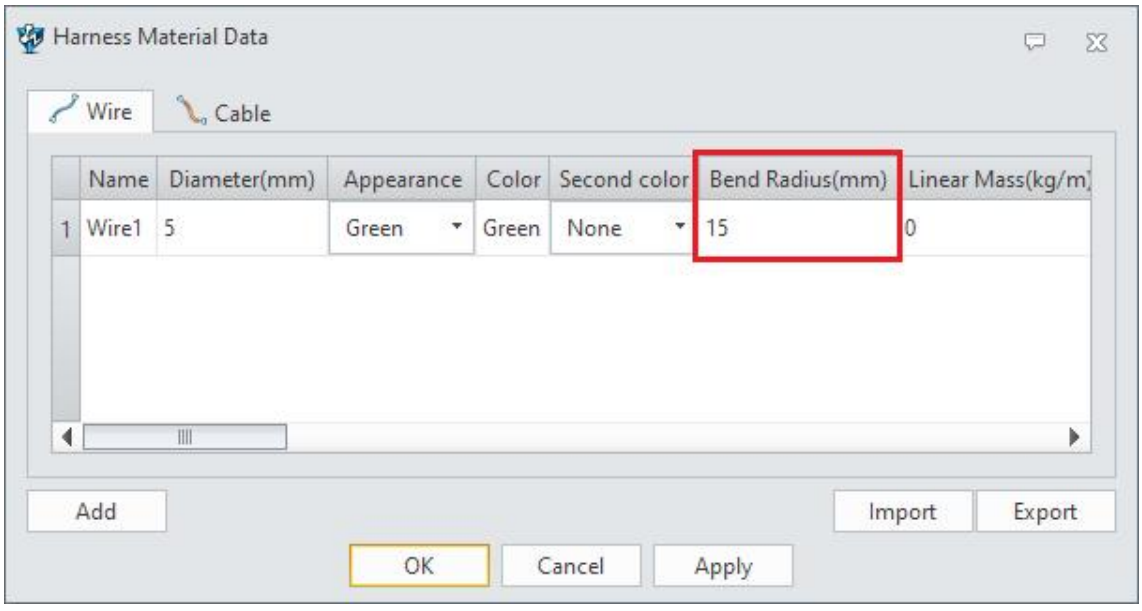
In order to facilitate users' understanding of the actual definition value of bending radius, the definition method of wire harness bending radius has been changed from the original "diameter * bending radius ratio" calculation to direct definition。

**【What users can do】**

✓ Directly display the minimum bending radius value that needs to be defined, without the need for calculation.

**【Example】**

Enter the wiring harness assembly environment, use the "Define Wires/Cables" and "Define Protective Layers" functions, and add any wire to discover that the definition of the bending radius of the wiring harness is based on the actual value, without the need for calculation.




**【Where is it】**

Wire harness assembly >> Attribute >> **Define Wire/Cable**

Wire harness assembly >> Attribute >> **Define protection**

### 4.4.3.2 Improve the Efficiency of Attach

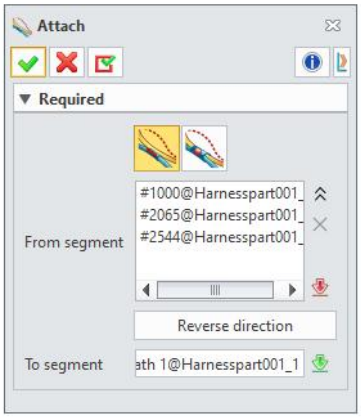
In order to improve the efficiency of attach function, the function of attaching multiple wires to a common path at once has been implemented, avoiding multiple clicks on the "attach" command.

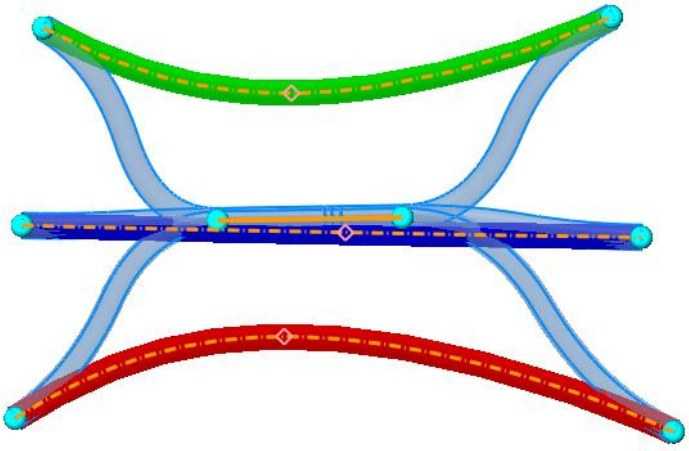
**【What users can do】**

- ✓ Select multiple wire harnesses at once to attach to a wire harness segment/path/port.
- ✓ You can adjust the attachment direction of each segment that needs to be attached.

**【Example】**

Enter the wiring harness component environment, use the "attach" function, select multiple "From segment", and then select the "to segment" that needs to be attached. After confirmation, multiple wiring harnesses can be attached at once.



**【Where is it】**

Wire harness components >> Edit >> **Attach**

### 4.4.3.3 Improve the Efficiency of Wiring and Editing Harnesses

In order to improve the efficiency of wiring and editing harnesses, automatic offset can be achieved when selecting points on the wiring/editing harness selection surface, without interfering with the selected surface and reducing adjustment operations; You can also directly edit the entire wiring harness

without the need for segmented editing.

When there are a large number of terminals or virtual terminals are used, in order for users to select the target terminal more quickly, the wiring and editing harness operation command box can conveniently select the target terminal, improve modeling efficiency, and ensure the accuracy of virtual terminal connection during use.

**【What users can do】**

✓ When selecting a point on the surface, the wire harness automatically offsets without interfering with the surface.

✓ The direction of the entire wire harness can be edited at once without the need for segmented editing.

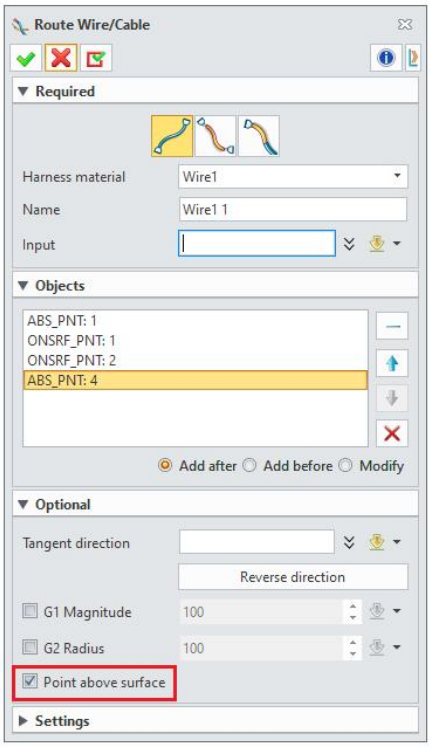
✓ Double clicks the target harness on the harness manager to enter the edit harness interface without clicking the "Modify Harness" command;

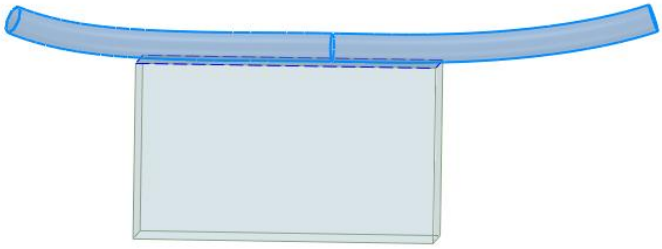
**【Notes】**

• After selecting the port, if the specific terminal name of the wiring harness connection is adjusted for the actual terminal, the wiring harness model will automatically switch to the terminal with the corresponding terminal name; For virtual terminals, the wiring harness is always connected at the port position.

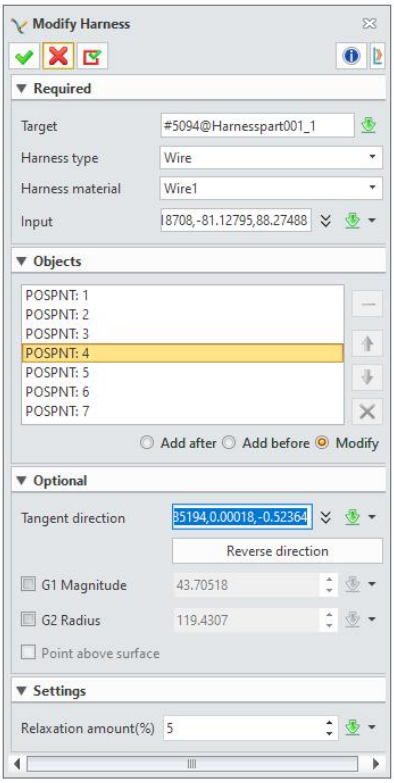
**【Example】**

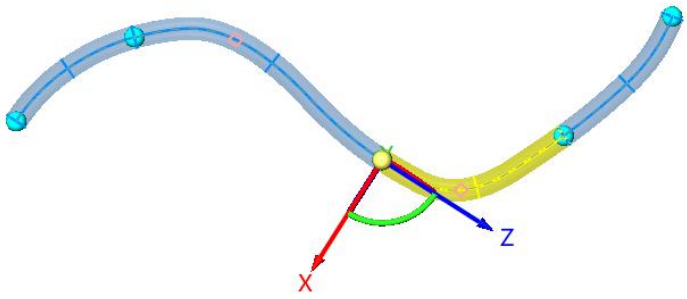
1) Enter the wiring harness component environment, use the "Route Wire/Cable" function, check the "Point above surface" option when routing, and then select a point on the reference surface to automatically offset the radius height of the wiring harness at the selected point on the surface, without interfering with the surface. This feature can be used in "Route Wire/Cable", "Create Protection", "Create Branch", and "Modify Harness".



2) Choose any harness with multiple segments and use the "Modify Harness" function to select the entire harness for editing. You can also double-click the target harness on the harness manager to directly enter the edit harness command interface for editing。




**【Where is it】**

Wire harness components >> Harness route >> **Route wire/cable**

Wire harness components >> Harness route >> **Create protection >> General**

Wire harness components >> Harness route >> **Create branch**

Wire harness components >> Edit >> **Modify harness**

## 4.4.4 Optimized Nailboard Diagrams

### 4.4.4.1 Optimizing the Layout of Nailboard Diagrams

In order to make the layout of the nail board diagram clearer and avoid layout overlaps, the automatic layout of the nail board diagram is optimized to improve the efficiency of layout adjustment.

**【What users can do】**

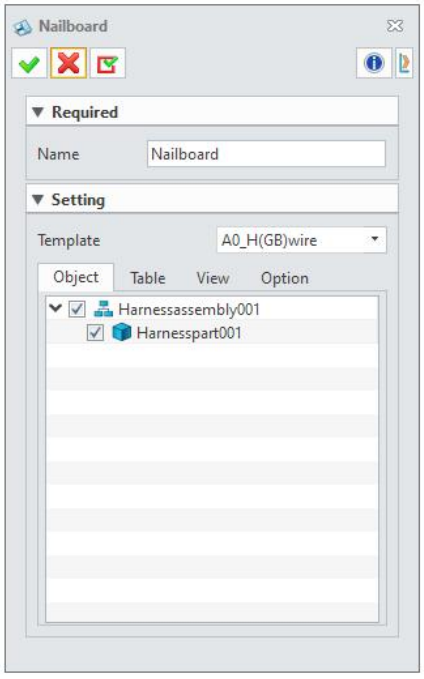
- ✓ After flattening the nail board diagram, there is no overlapping interference.

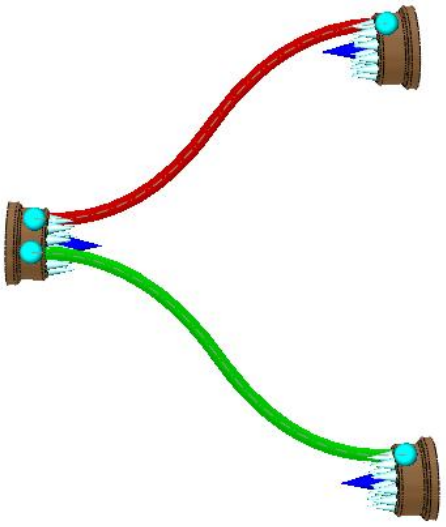
✓ After the nail board diagram is flattened, it is arranged uniformly from top to bottom, with the area occupied by each flattened diagram arranged from small to large.

✓ After flattening the nail board diagram, the area with the largest proportion of flattened area is placed on the drawing frame.

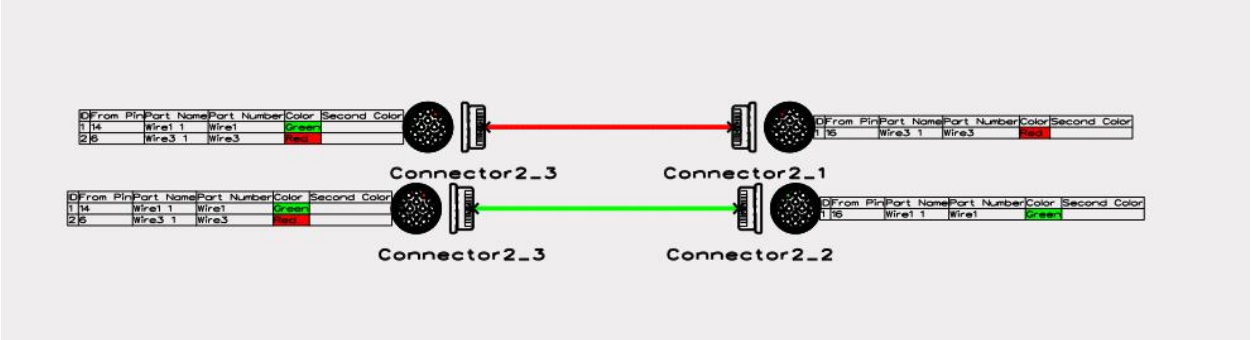
**【Example】**

1) Enter the wiring harness assembly environment and use the "nail board diagram" command to automatically flatten the completed drawn wiring harness parts into nail board diagrams.



2) Upon entering the nail board diagram environment, it can be observed that the wiring harness in the nail board diagram is flattened and arranged vertically, without any overlapping issues.

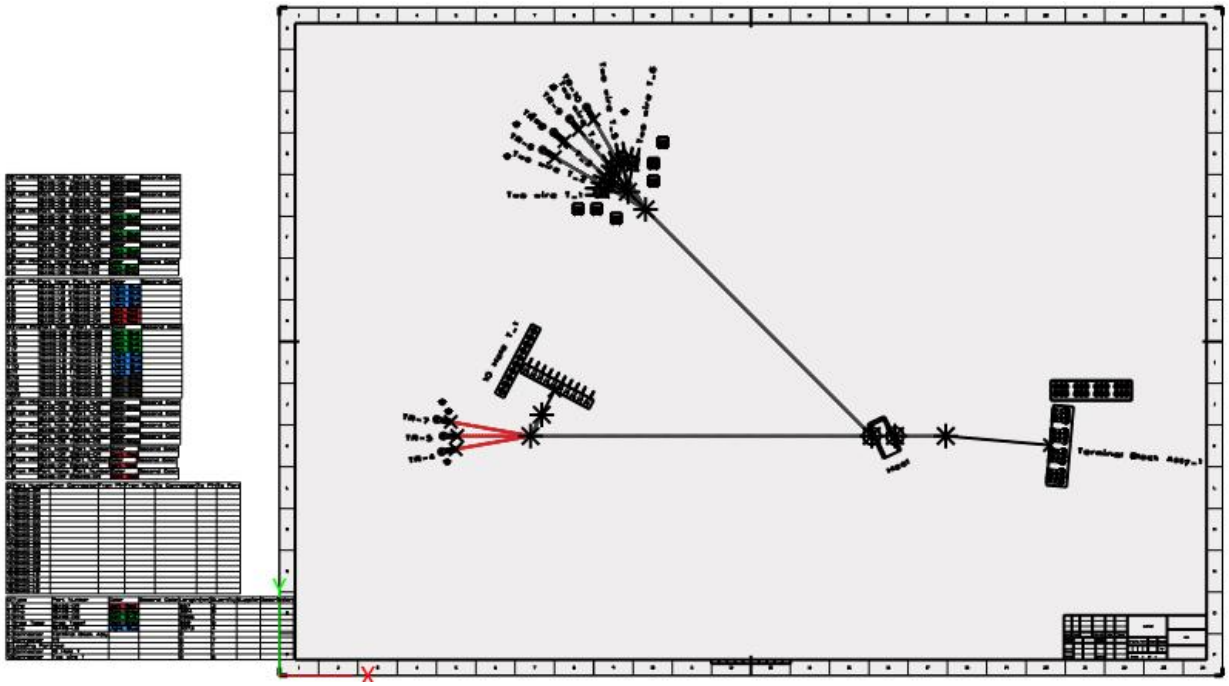


**【Where is it】**

Wire harness assembly >> Drawings >> **Nailboard**

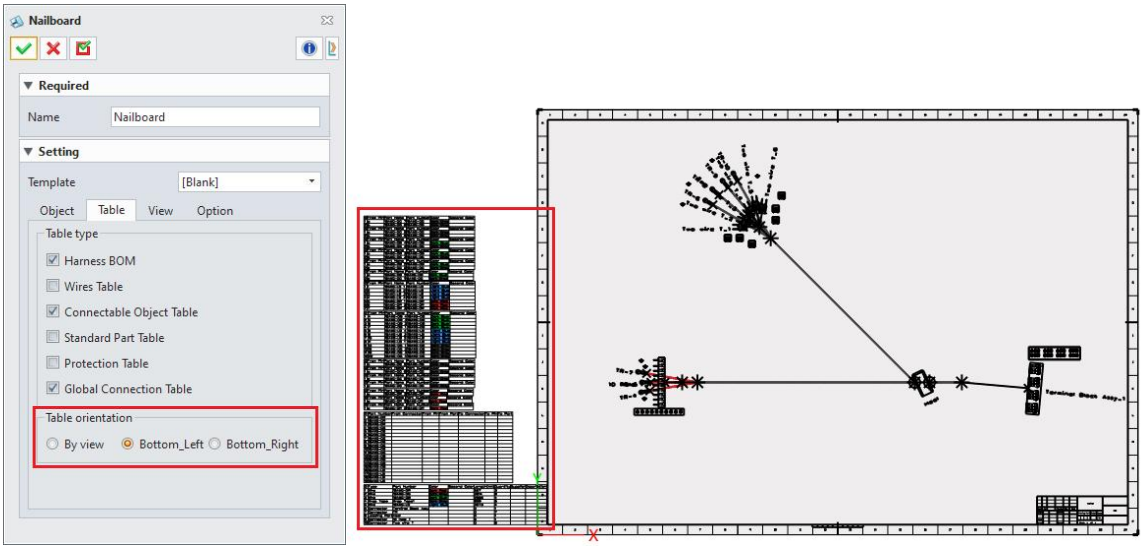
## 4.4.4.2 ★Nailboard table and label position update（ZW3D 2026 SP1）

Adding "Table orientation" and "Label position" to the nailboard in wiring harness design can make the nailboard clearer after automatic flattening, avoiding interference and overlap between tables, labels, and flattened wiring harnesses.

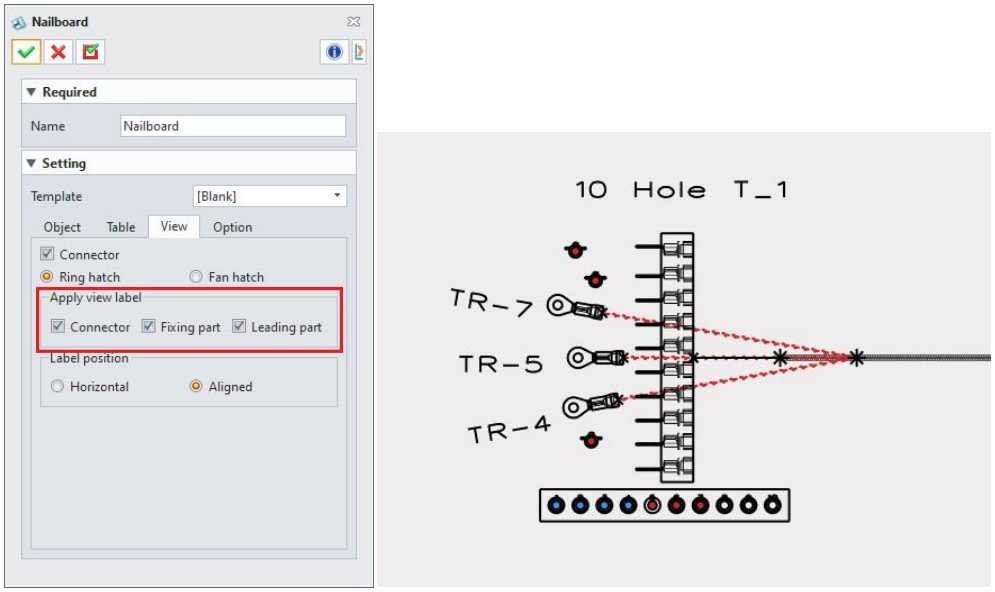


**【What users can do】**

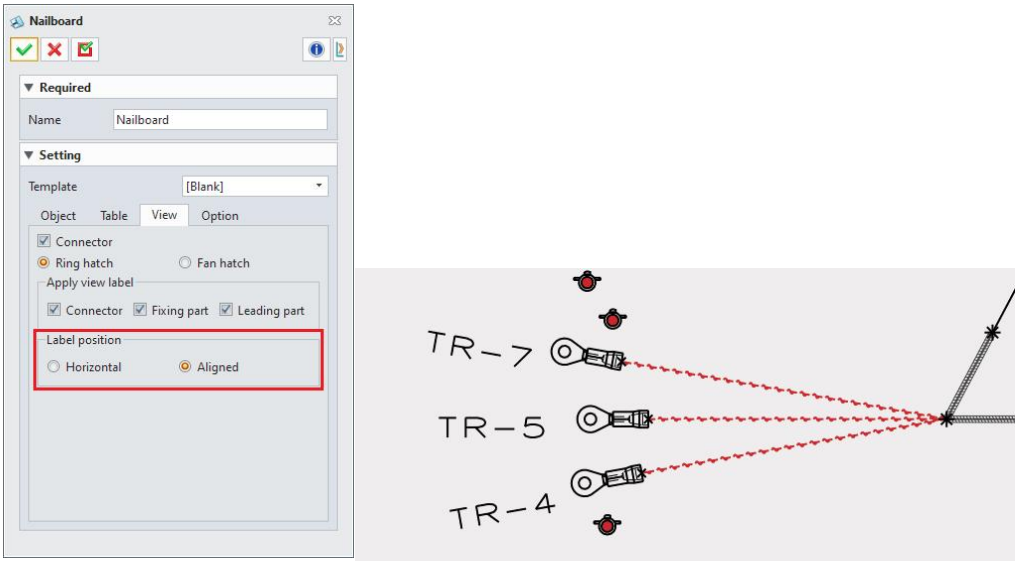
✓ You can control the automatic placement of the table, allowing three ways: By view (placing it next to the connector view), Bottom_Left (placing it in the lower left corner of the entire flattened view), and Bottom_Right (placing it in the lower right corner of the entire flattened view).




✓ Can control whether different standard parts output labels.




✓ The placement of labels can be controlled, allowing for horizontal placement or placement in a direction where the bundle is flattened.




**【Notes】**

- In order to avoid interference between the table and the flattened map, the table will appear beyond the map, and the placement position can be adjusted by dragging as needed.

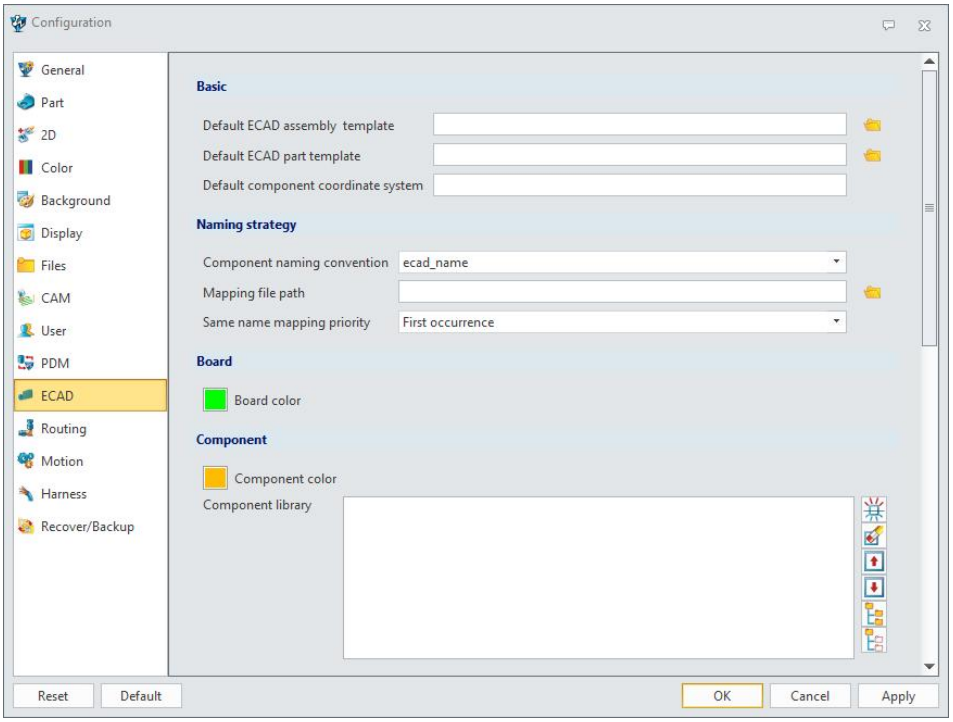
## 【Where is it】

Wire harness assembly >> Drawing >> Nailboard >> Setting >> Table/View

## 4.5 ECAD

### 4.5.1 ECAD Configuration

Through initiatives like refining the mapping strategy and expanding the electronic component library, we can enhance the interaction between ECAD and MCAD. This enables us to meet the workflow requirements for design in relevant scenarios within sectors such as communication equipment, automotive electronics, and consumer electronics, facilitating designers to complete product design and validation more expeditiously.




Adding "Default ECAD assembly template" and "Default ECAD part template" in Configuration - ECAD -Basic, it can help you predefine templates for ECAD file creation and better standardized management.

Adding a "Naming strategy " in Configuration - ECAD can help you manage mapping and define MCAD component names according to enterprise standards during the import/export IDF files.

In the configuration - ECAD - electronic component library, content such as support for multi-path can help you set up multiple different electronic component libraries, which helps enterprises better

manage their component libraries.

### 4.5.1.1 Default Template

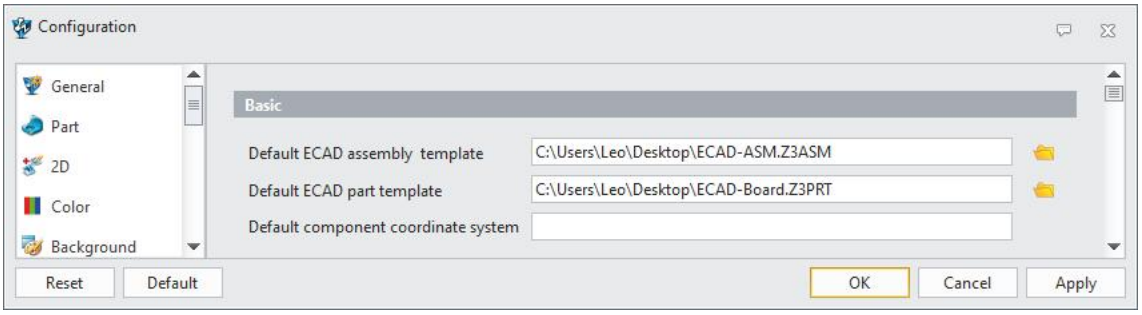
In the early stages of ECAD design, by setting the "Default ECAD Assembly Template" and "Default ECAD Part Template", users can quickly select the corresponding templates when creating files, thereby improving design efficiency.

**【What users can do】**

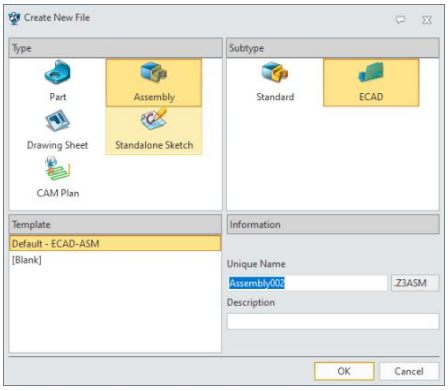
✓ Assembly Template: Used to set the template files required for creating ECAD assemblies. First, the template is defined and stored in the corresponding path. When users create new ECAD assemblies, they can directly call the template to create the corresponding assembly.

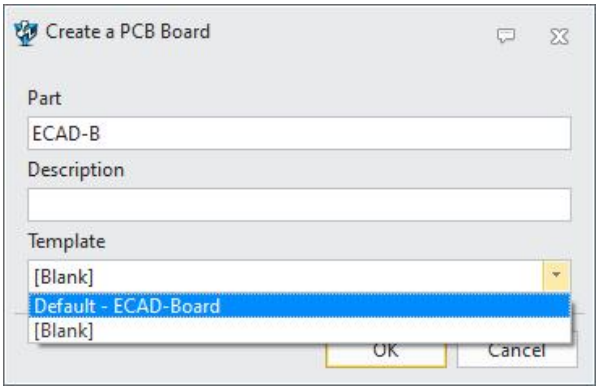
✓ Part Template: Used to set the template files required for creating ECAD boards. First, define the template and store it in the corresponding path. When users create a new ECAD board, they can directly call the template to create the corresponding ECAD board.

Set template file.




✓ When creating, you can choose the corresponding template.



**【Notes】**

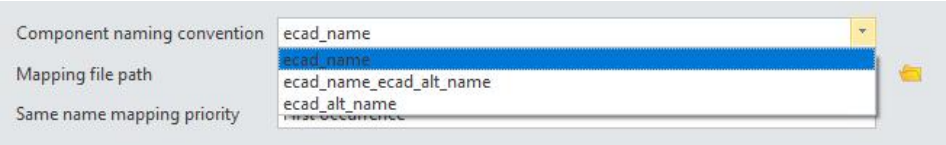
- The assembly template must be selected for ECAD assembly to be effective and cannot be selected for regular assembly.
- The part template must be selected as an ECAD board file to be valid, and ordinary parts cannot be selected.

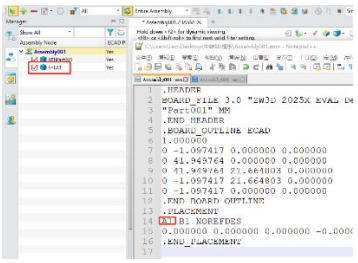
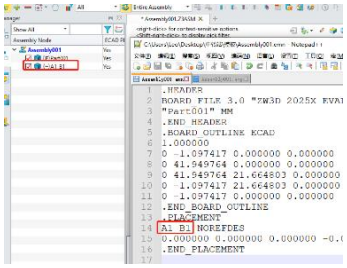
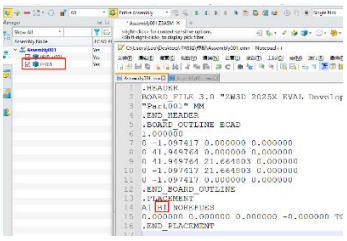
### 4.5.1.2 Naming Strategy

When importing or exporting IDF files, users can control the names of related fields after import or export by setting options such as "Component Naming Convention", "Mapping File Path", and "Same Name Mapping Priority", such as ECAD_NAME, ECAD_ALT-NAME, etc.

**【What users can do】**

✓ Component Naming Convention. You can control which name to use as the component name after importing the IDF file, with options including ECAD_NAME ECAD_NAME_ ECAD_ALT_NAME 、ECAD_ALT_NAME.



| ECAD_NAME | ECAD_NAME_<br>ECAD_ALT_NAME | ECAD_ALT_NAME |
|---|---|---|
|  |  |  |

✓ Mapping file path: Set the default mapping file path so that there is a mapping file in that location. When importing/exporting IDF, the corresponding file can be selected by default as the mapping file for parameter mapping.

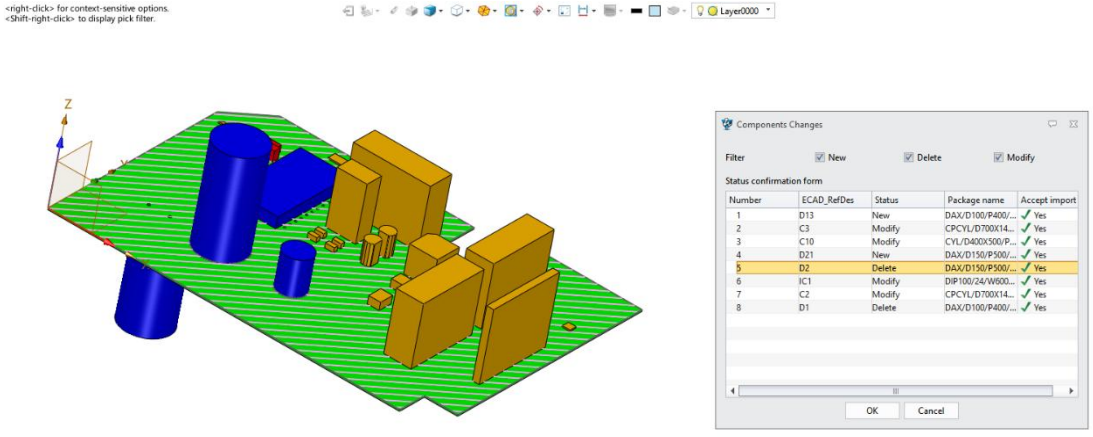
✓ Same name mapping priority: When encountering the same name mapping in the mapping file, it can be set to map in order (First occurrence) or in reverse order (Last occurrence).

### 4.5.2 ★Compare IDF

The new compare IDF function can compare component changes (including New, Delete, and Modify) and control whether to accept changes。

**【What users can do】**

✓ Using ECAD_RefDes as the unique identifier, when a component change is identified, such as adding, deleting, or modifying a component, a change list will pop up for you to choose whether to accept these changes.



**【Notes】**

- The ECAD_RefDes is the unique identification code used for comparison, and components must have an ECAD_RefDes to be compared. Otherwise, they will be considered as new items and imported directly

**【Where is it】**

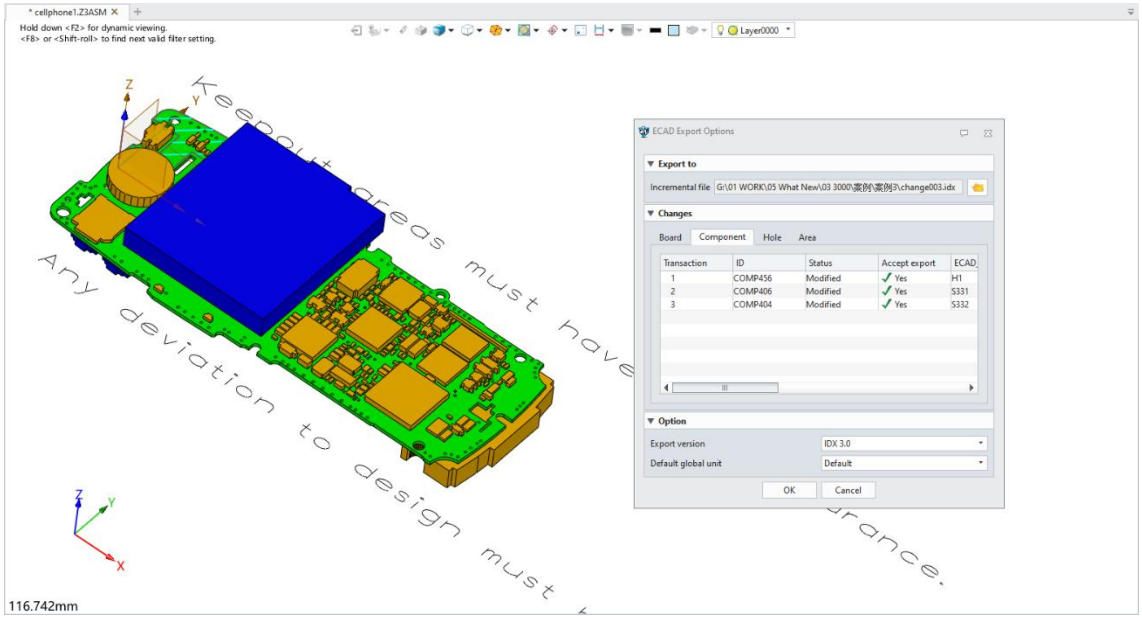
ECAD Assembly >> Data Exchange >> PCB Exchange>>**Compare IDF**

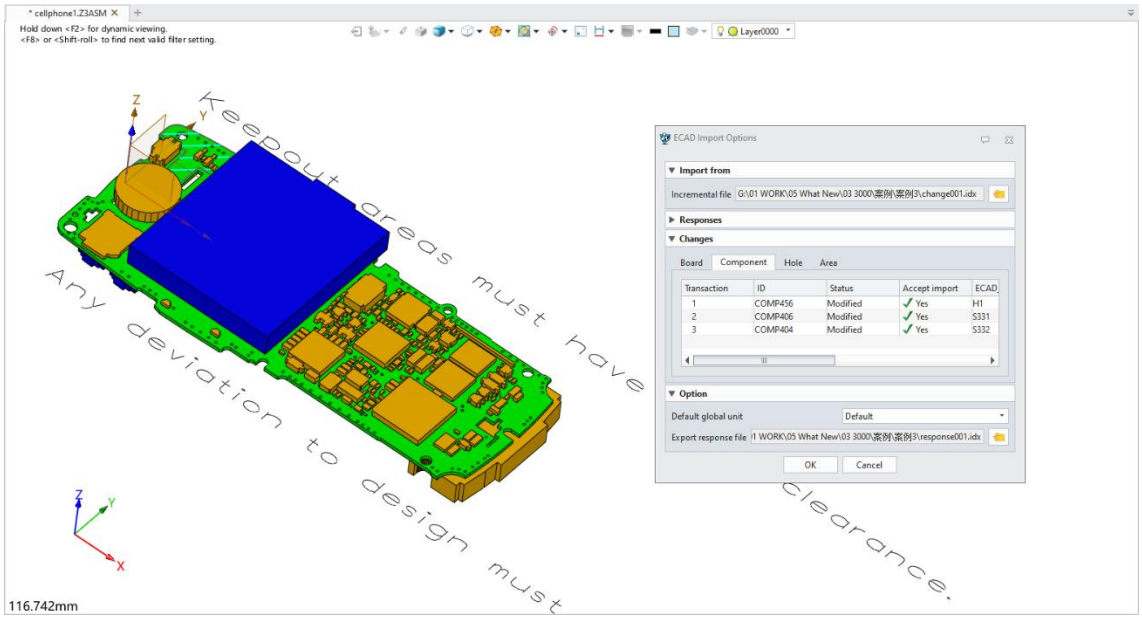
## 4.5.3 ★IDX Import and Export

The new IDX import and export function can achieve incremental import and export, greatly improving design efficiency while reducing error rates

**【What users can do】**

- ✓ Mark Baseline
- ✓ Export Increment
- ✓ Import Increment






**【Where is it】**

ECAD Assembly >> Data Exchange >> PCB Exchange>>**Mark Baseline/Export Increment/ Import Increment**

## 4.5.4 .emn file import optimization（ZW3D 2026 SP1）

While importing an .emn file, you can now input the fraction value of Instead board thickness, Default board thickness, Default component height and Default hole diameter locating at ECAD Import Options manager with fraction.

**【What users can do】**

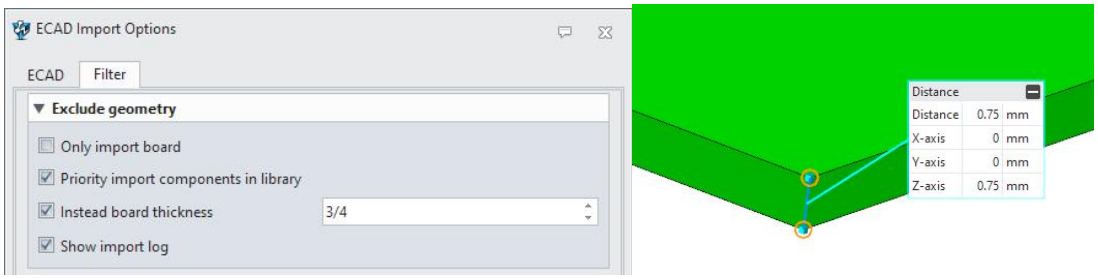
✓ Input fraction such as 3/4 in Instead board thickness, Default board thickness, Default component height and Default hole diameter (Plus+, minus-, times* are also available).

**【Notes】**

- Step widget will be inactive while +, -, *, / is in input box.

**【Example】**

Input 3/4 in the box of Instead board thickness, and you can check the board thickness is 0.75 mm.




## 【Where is it】

File >> Import >> Import··· >> Select File to Import··· >> Pick an .emn file >> **Import**

Or: Open an .emn file directly.

# 5 CAM

## 5.1 2-Axis Module （ZW3D 2026 SP1）

### 5.1.1 “Contour” Operation Optimization

When the " Contour" operation uses the "Zigzag" cutting direction to machine open regions, if there is a boundary, the toolpath at the boundary maintains climb milling to achieve better side wall machining results.

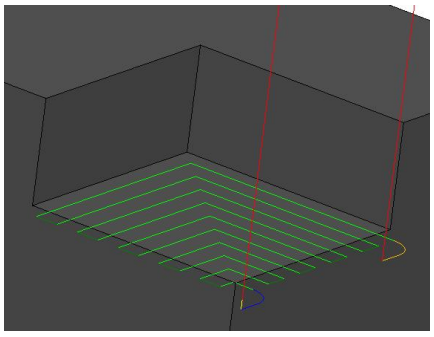
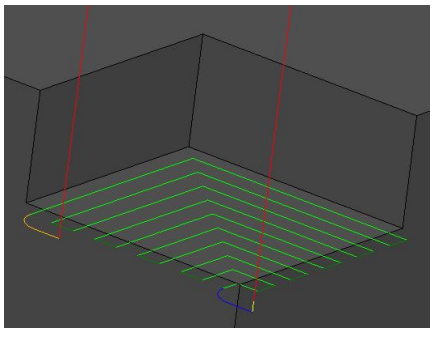
**【What Users can do】**

When the cutting direction is "Zigzag," the tool path at the boundary of the open region maintains climb milling.

**【Notes】**

- Effective only for the boundaries of open regions.

**【Example】**

| ZW3D 2025 | ZW3D 2026 SP1 |
| --- | --- |
|  |  |

**【Where is it】**

2x Mill >> 2D Pocket >> Contour >> Cutting Control >> **Cut Direction**

## 5.2 QM Module

### 5.2.1 New “Z Constant” Operation

The operation of "Z Constant" uses a new surface selection process. By selecting the part, cut region, avoidance and containment, it can provide accurate contour machining, suitable for machining parts with complex profiles, especially in the precision cutting of steep regions, especially in mold manufacturing and complex parts processing. It enables high quality surface finish and precise dimensional control.

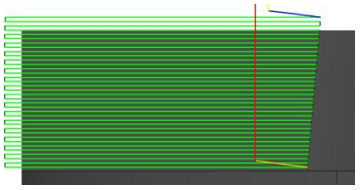
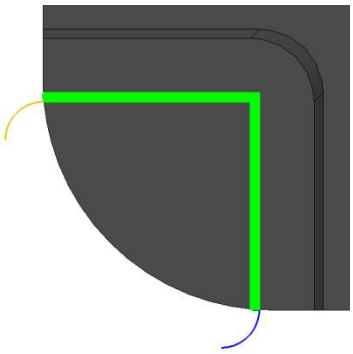
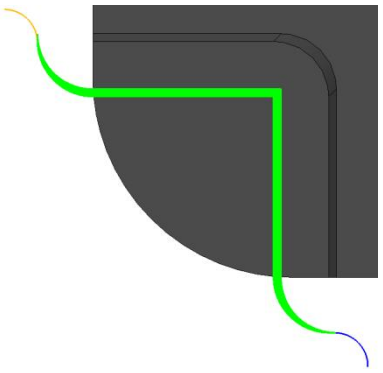
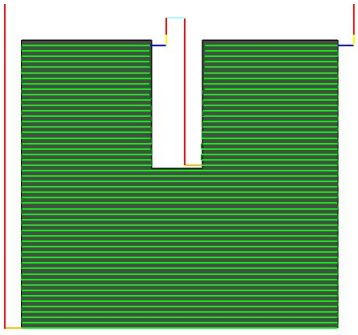
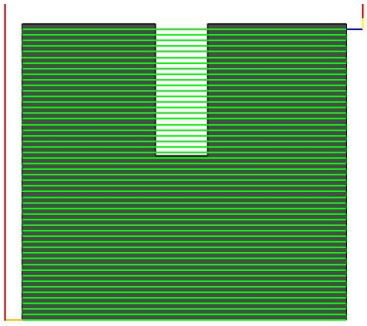
**【What users can do】**

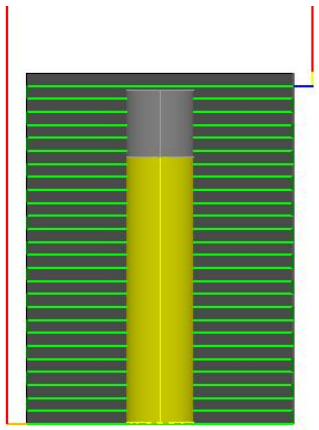
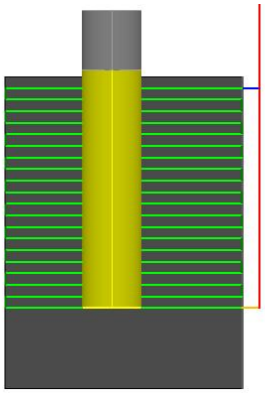
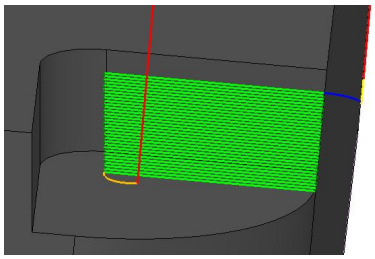
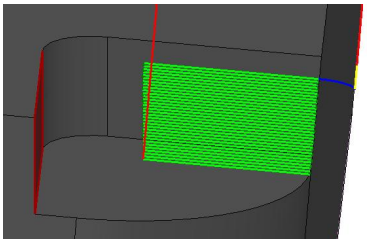
- ✓ You can customize common processing parameters.
- ✓ Accurately define part, cut region, avoidance and containment.
- ✓ In the operation interface, you can select available tools for the operation, view the selected tool, create a new tool and edit the selected tool.
- ✓ Custom operation tolerance, custom part thickness and avoidance thickness.
- ✓ New cut layers function for segmented finishing, tool joint and local density adjustment
- ✓ Supports the division of cut regions according to steep angle.
- ✓ Support for merging toolpaths within a specified distance of the same layer to reduce link and lead.
- ✓ Supports tangential extension of the toolpaths.
- ✓ You can select whether to generate rolling toolpaths.
- ✓ The tool neck, shank, and holder are checked for cutting and non-cutting toolpaths respectively.
- ✓ Supports adding notes to the operation.

**【Notes】**

- The toolpaths can only extend up to 25% of the tool diameter.
- When selecting the holder for collision detection, the tool shank inspection distance can be set at the same time.

**【Example】**

| Functions | Unused | After use |
|---|---|---|
| Tool Path Extension | |  |
| Tool Rolling |  |  |
| Merge Distance |  |  |

| | | |
|---|---|---|
| Check Neck |  |  |
| Avoidance Thickness |  |  |

**【Where is it】**

3x Quick >> **Z Constant**

## 5.2.2 New “Region Mill” Operation

The operation of "Region Mill" uses a new surface selection process. By selecting the part, cut region, avoidance and containment, it is usually used in complex surface machining. “Region Mill” can be realized respectively on the steep region and shallow region for processing. Because of its flexibility and adaptability, it plays an important role in numerical control programming. It can meet the requirements of different complexity and precision machining tasks. Through reasonable parameter setting and process planning, the machining efficiency and quality can be effectively improved.

**【What users can do】**

- ✓ You can customize common processing parameters.
- ✓ Accurately define part, cut region, avoidance and containment.
- ✓ In the operation interface, you can select available tools for the operation, view the selected

tool, create a new tool and edit the selected tool.

✓ Custom operation tolerance, custom part thickness and avoidance thickness.

✓ Supports the division of cut regions according to steep angle.

✓ Supports machining of shallow region and steep region separately and also supports machining of shallow region and steep region simultaneously.

✓ The strategy of “Z Constant” is supported in steep regions and the strategy of “Lace” is supported in shallow regions.

✓ Support for merging toolpaths within a specified distance of the same layer to reduce link and lead.

✓ Supports tangential extension of the toolpaths.

✓ You can select whether to generate rolling toolpaths.

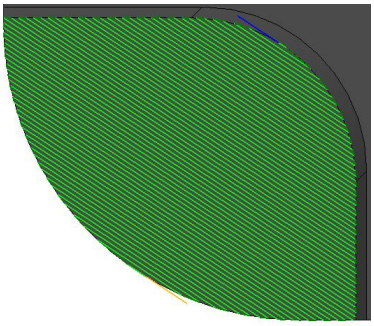
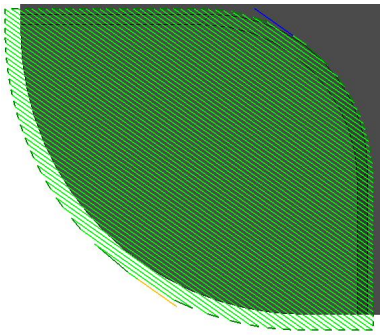
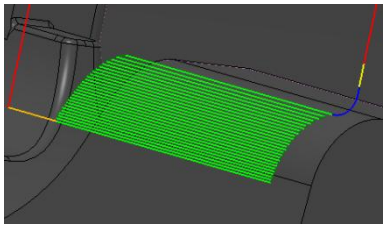
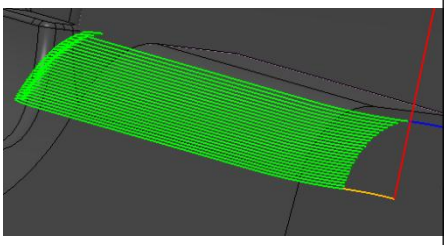
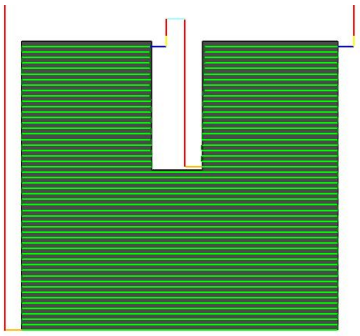
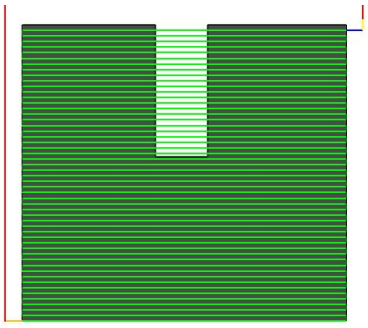
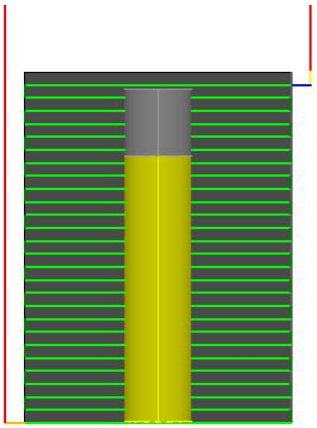
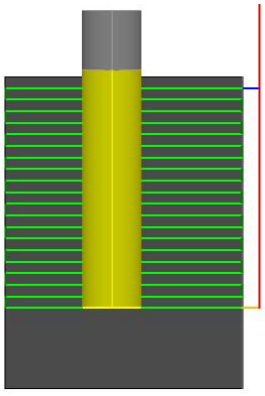
✓ The tool neck, shank, and holder are checked for cutting and non-cutting toolpaths respectively.

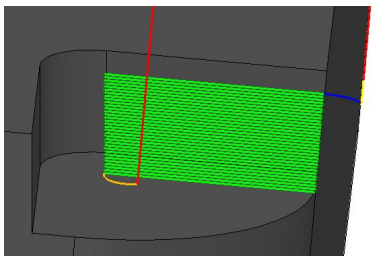
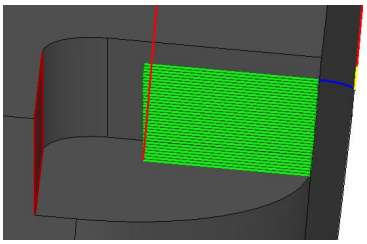
✓ Supports adding notes to the operation.

**【Notes】**

- The toolpaths can only extend up to 25% of the tool diameter.
- When selecting the holder for collision detection, the tool shank inspection distance can be set at the same time.
- Only normal thickness is supported.

【Example】

| Functions | Unused | After use |
|---|---|---|
| Tool Path Extension |  |  |
| Tool Rolling |  |  |
| Merge Distance |  |  |
| Check Neck |  |  |

| | | |
|---|---|---|
| Avoidance Thickness |  |  |

**【Where is it】**

3x Quick >> **Region Mill**

## 5.2.3 New “Chamfer Debur” Operation

Chamfer Debur Operation is commonly used to process the sharp edges of parts, which can reduce the amount of subsequent manual grinding and polishing work, saving time and costs.

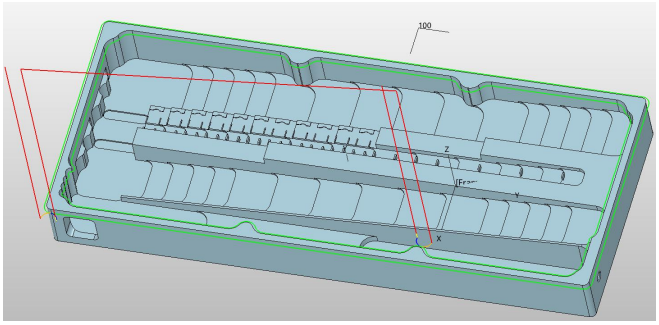
**【What users can do】**

- ✓ Edge Deburring
- ✓ Ignore Holes

**【Notes】**

- You need to select a plane perpendicular to the Z-axis of the MCS as the machining object.
- It does not support the selection of the model's sharp edge lines for processing.
- Areas that you do not wish to machine can be excluded using the exclude edge function

**【Example】**

| | |
|---|---|
| Edge Deburring |  |

**【Where is it】**

3x Quick >> Finish >> **Chamfer Debur**

## 5.2.4 New “Chamfer Plunge”Operation

Chamfer Plunge Operation is commonly used for processing the sharp edges of convex corners on parts, reducing the sharpness of the part edges to prevent cuts or wear. This operation is highly efficient in terms of processing, saving a significant amount of time.

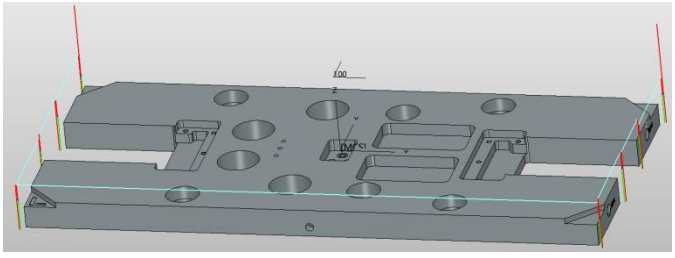
**【What users can do】**

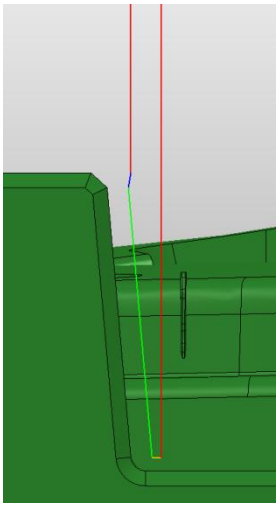
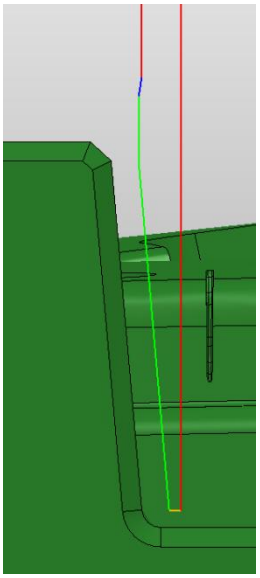
- ✓ Vertical Convex Sharp Edge.
- ✓ Convex Sharp Edge with Draft Angle.
- ✓ Z-Axis Extension of Tool Path.
- ✓ Tangential Extension of Tool Path.
- ✓ Chamfering Curved Edges.

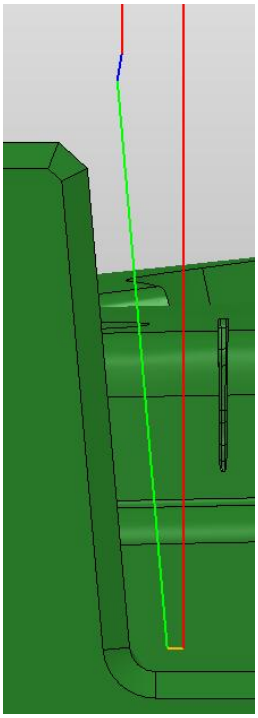
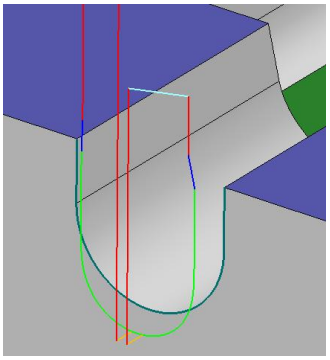
**【Notes】**

- The selection filter for machining draft angle chamfers needs to adjust the maximum convex angle value to avoid being filtered out and not selectable.

**【Example】**

| | |
|---|---|
| Vertical Convex Sharp Edge |  |

| | |
|---|---|
| Convex Sharp Edge with Draft Angle |  |
| Z-Axis Extension of Tool Path |  |

| | |
|---|---|
| Tangential Extension of Tool Path |  |
| Chamfering Curved Edges |  |

**【Where is it】**

3x Quick >> Finish >> **Chamfer Plunge**

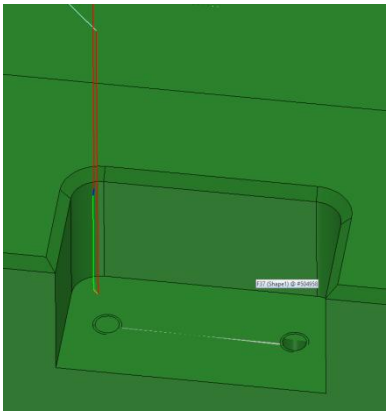
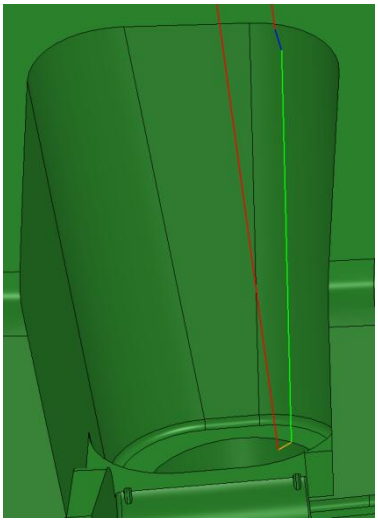
## 5.2.5 New “Corner Plunge” Operation

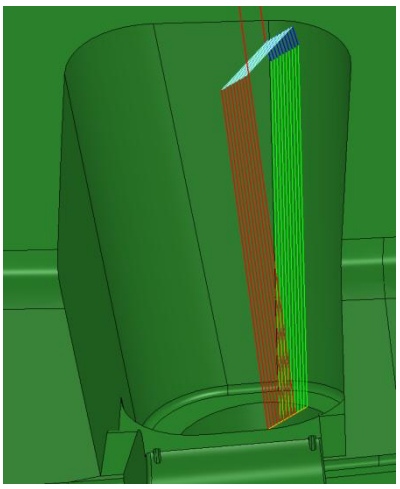
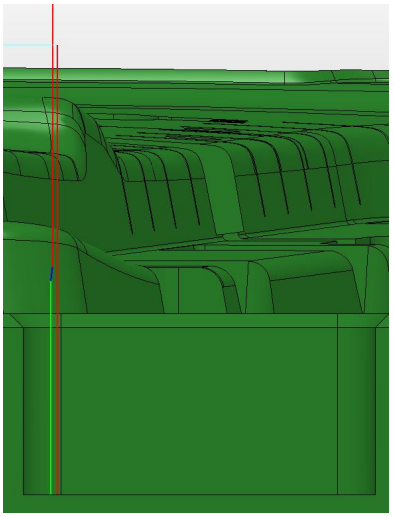
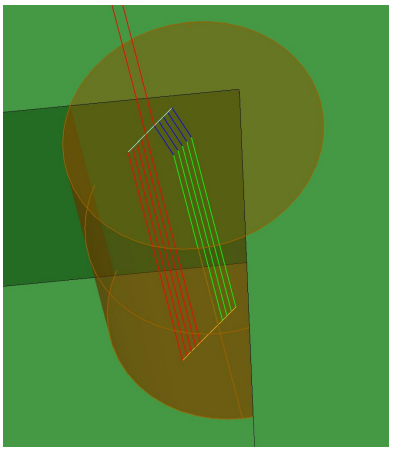
Corner Plunge Operation is commonly used for processing the concave corner fillet transitions on parts, which helps to reduce stress concentration and improve the appearance quality of the parts. This operation is highly efficient in terms of processing, saving a significant amount of time.

**【What users can do】**

- ✓ Vertical Arc Concave Corner.
- ✓ Arc Concave Corner with Draft Angle.
- ✓ Multi-Toolpath Offset.
- ✓ Toolpath Extension.
- ✓ Avoidance Angle Chamfering.

**【Example】**

| | |
|---|---|
| Vertical Arc Concave Corner |  |
| Arc Concave Corner with Draft Angle |  |

| | |
|---|---|
| Multi-Toolpath Offset |  |
| Toolpath Extension |  |
| Avoidance Angle Chamfering |  |

**【Where is it】**

3x Quick >> Finish >> **Corner Plunge**

## 5.3 5-Axis Module

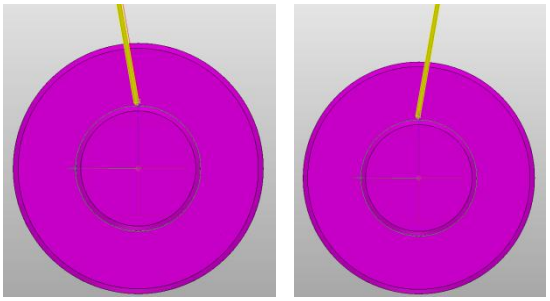
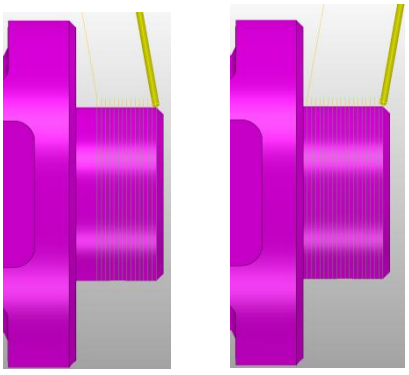
### 5.3.1 New Lead Angle and Side Tilt Angle Functions in the "Guiding Surface Iso" Operation

Guiding Surface Iso Operation has added tool axis control functionality, allowing users to perform tool axis lead inclination and side inclination machining, which can cover more five-axis machining scenarios.

**【What users can do】**

- ✓ Tool Axis Front and Rear Lead.
- ✓ Tool Axis Left and Right Roll

**【Example】**

| | |
|---|---|
| Tool Axis Front/Rear Lead |  |
| Tool Axis Left/Right Roll |  |

**【Where is it】**

5x Mill >> Cut >> Guide Surface Iso >> **Axis Control**

## 5.4 Platform Matching Functionalities

### 5.4.1 Tool Improvement

#### 5.4.1.1 New Milling Tools

Milling added End Mill-Straight and Ball Mill, suitable for more application scenarios, to further meet the actual processing needs.

**【What users can do】**

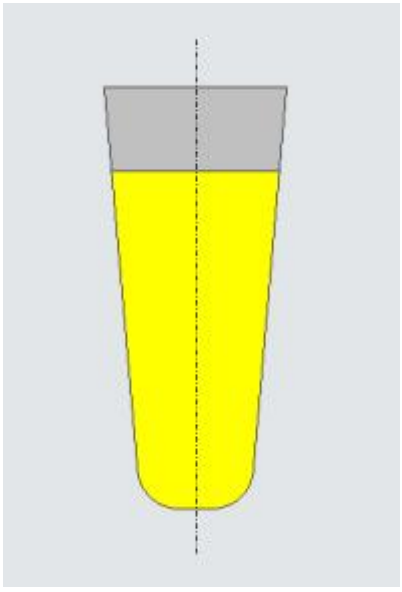
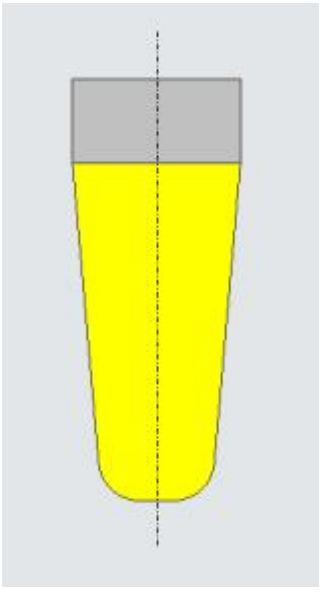
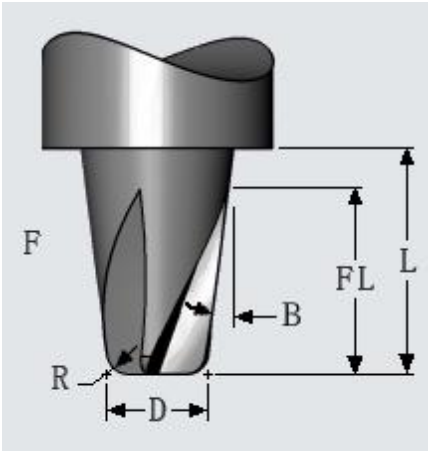
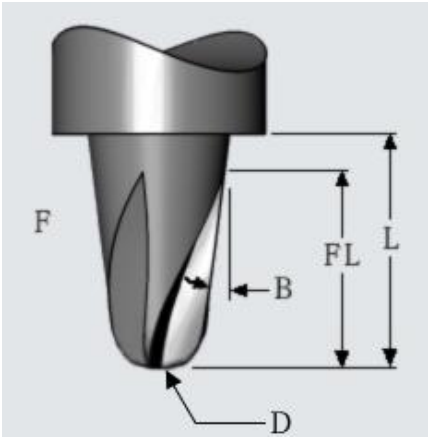
✓ Supports the creation of End Mill-Straight, and the angle of the neck part does not change with the "Angle" of the tool parameter.

✓ Supports the direct creation of the Ball Mill, no longer need to create the Ball Mill according to the End Mill;

**【Notes】**

- One of Angle, Flute Len and Cutter Dia of End Mill-Straight is calculated automatically.
- There is one and only one automatic calculation parameter for End Mill-Straight.
- In End Mill-Straight, when the selected calculation parameter is manually modified, the next parameter is automatically used as the calculation parameter.

【Example】

| End Mill | End Mill-Straight |
| --- | --- |
|  |  |
| End Mill | Ball Mill |
|   |   |

【Where is it】

Setup >> Tool Manager >> **Mill**

### 5.4.1.2 The Parameter of Chamfer Adjustment

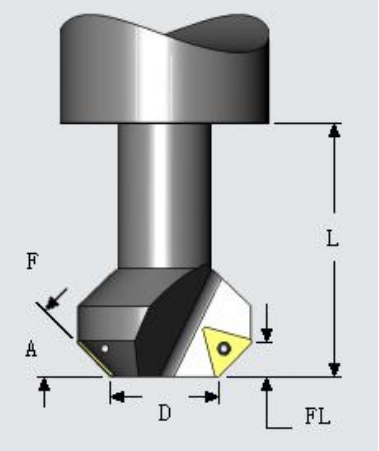
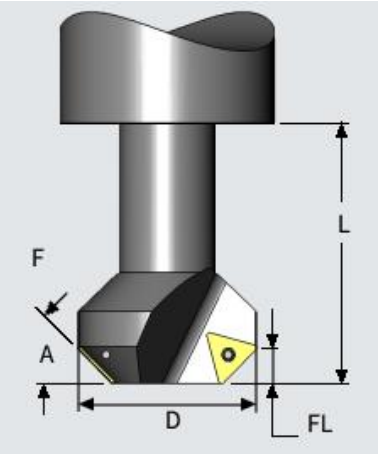
Adjust the chamfer cutter diameter from the inside diameter to the outside diameter, which is more

in line with the actual tool parameters.

**【What users can do】**

- ✓ Define Chamfer by the outside diameter;

**【Example】**

| Chamfer（ZW3D2025） | Chamfer（ZW3D2026） |
| --- | --- |
|  |  |



**【Where is it】**

Setup >> Tool Manager >> **Mill**

### 5.4.1.3 The (OS) Offset of Holder Supporting Simple Calculations

The (OS) offset of holder supports simple addition, subtraction, multiplication and division operations, which is convenient for users to set clamping parameters, without the user's own calculation.

**【What users can do】**

- ✓ Support simple addition, subtraction, multiplication and division operations.

**【Where is it】**

Setup >> Tool Manager >> Mill >> **Holder**

### 5.4.1.4 Turning Default Cutting Parameters Adjustment

Adjust the default feed speed of the turning tool, which is close to the actual turning parameters

**【What users can do】**

- ✓ The default turning tool feed speed is 0.3 in mm/RPM;

**【Where is it】**

Setup >> Tool Manager >> Lathe >> **Speed/Feed**

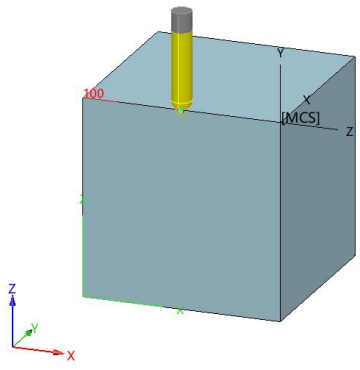
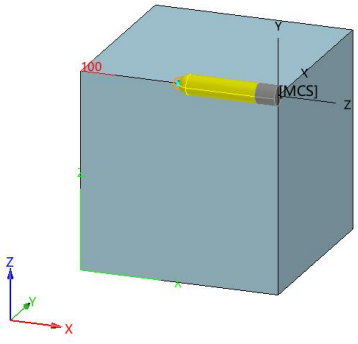
### 5.4.1.5 Tool follows the active coordinate system display（ZW3D 2026 SP1）

The creation of a tool following the active coordinate system display offers greater convenience when handling complex products and is better suited for machining programming solutions involving intricate components.

**【What Users can do】**

- ✓ Supports the creation of a tool following the active coordinate system display.

**【Example】**

| ZW3D 2025 | ZW3D 2026 SP1 |
| --- | --- |
|  |  |

**【Where is it】**

Setup >> **Tool Manager**

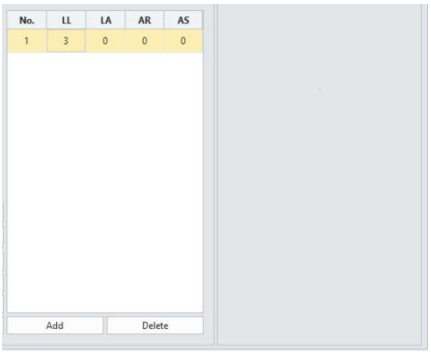
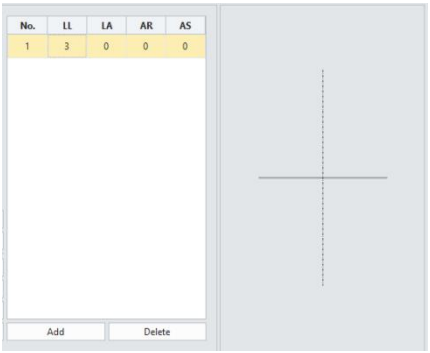
### 5.4.1.6 Preview Supported When Creating Custom Tools（ZW3D 2026 SP1）

When creating a custom tool with LA=0, a preview will be displayed to prevent users from assuming the functionality is invalid.

**【What Users can do】**

- ✓ Preview is displayed even when LA=0 during custom tool creation.

**【Example】**

| ZW3D 2025 | ZW3D 2026 SP1 |
| --- | --- |
|  |  |



**【Where is it】**

Setup >> Tool Manager >> Mill >> **User Define Tool**

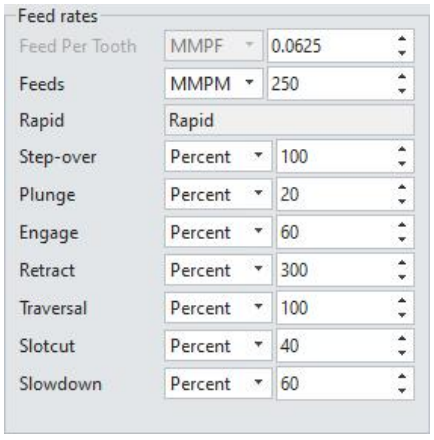
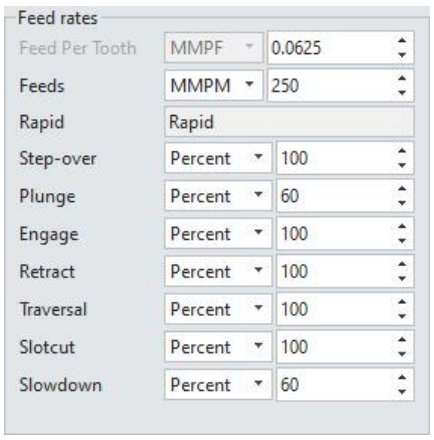
### 5.4.1.7 Milling Default Cutting Parameters Adjustment（ZW3D 2026 SP1）

Adjust the default feed speed of the milling tool, which is close to the actual milling parameters.

**【What Users can do】**

- ✓ Milling tool plung feed rate defaults to 60%.
- ✓ Milling tool Engage feed rate defaults to 60%.
- ✓ Milling tool Retract feed rate defaults to 60%.
- ✓ Milling tool Slotcut feed rate defaults to 60%.

**【Example】**

| ZW3D 2025 | ZW3D 2026 SP1 |
| --- | --- |
|  |  |





**【Where is it】**

Setup >> Tool Manager >> Mill >> **Speed/Feed**

## 5.4.1.8 Added Groove Turning Tool Tracking Points（ZW3D 2026 SP1）

Added groove turning tool tracking point category, with the tracking points in orthogonal position.

**【What Users can do】**

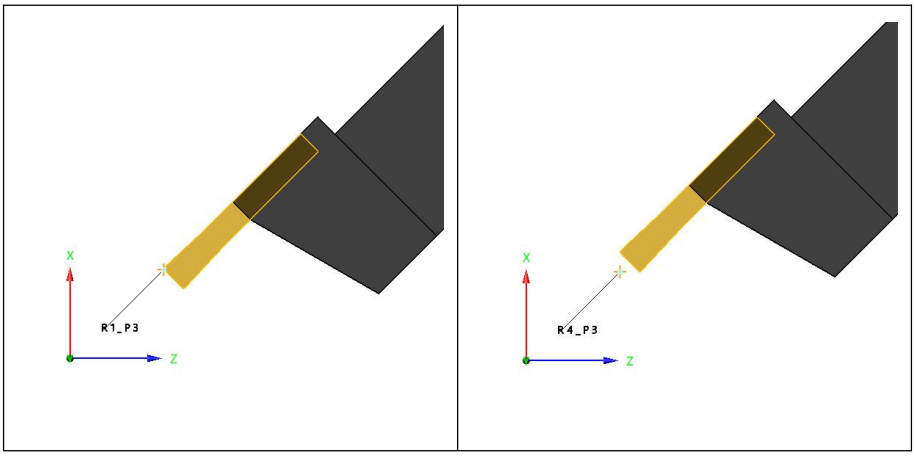
✓ Supports orthogonal position as tracking point.

**【Notes】**

- Only supported for groove tools.

**【Example】**

| ZW3D 2025 | ZW3D 2026 SP1 |
| --- | --- |




**【Where is it】**

Setup >> Tool Manager >> Lathe >> **Tracking Point**

## 5.4.2 Enhancement of Machining Plan Usability

The new features of "One-click to Display " and "One-click to Interrupt Toolpath Calculation" has been added.

### 5.4.2.1 One-click to Display

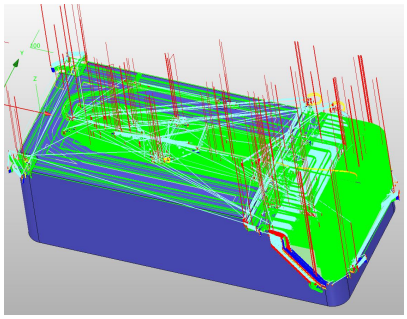
The new "One-click Display" function allows you to display only the selected process toolpaths.

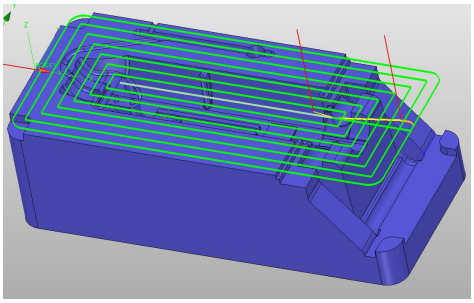
**【What users can do】**

✓ The “~” key is used to display only the selected process toolpath and hide all other process toolpaths.

**【Example】**

| Before use | After use |
|---|---|





**【Where is it】**

Keystroke "~"

### 5.4.2.2 One-click to Interrupt Toolpath Calculation

The new "One-click Interrupt Toolpath Calculation" feature allows you to interrupt the toolpath calculation process.

**【What users can do】**

- ✓ Interrupt Toolpath Calculation.

**【Example】**

1) Without one-click interrupt toolpath calculation: Software crashes and freezes.

2) With one-click interrupt toolpath calculation: Timely interruption of toolpath calculation to prevent software crashes and freezes.

**【Where is it】**

The toolpath calculation will automatically prompt whether to interrupt the toolpath calculation function after 5 seconds.

## 5.4.3 CAM Plugin

Four new plugins have been added to enhance usability and enable rapid programming:

- ✓ Batch Modify Operation
- ✓ Batch Rename Operation
- ✓ Batch Move Operation

- ✓ Batch Create/Edit Folder

### 5.4.3.1 Batch Modify Operation

The new "Batch Modify Operation" function allows for the bulk editing and modification of process parameters for selected operations.

**【What users can do】**

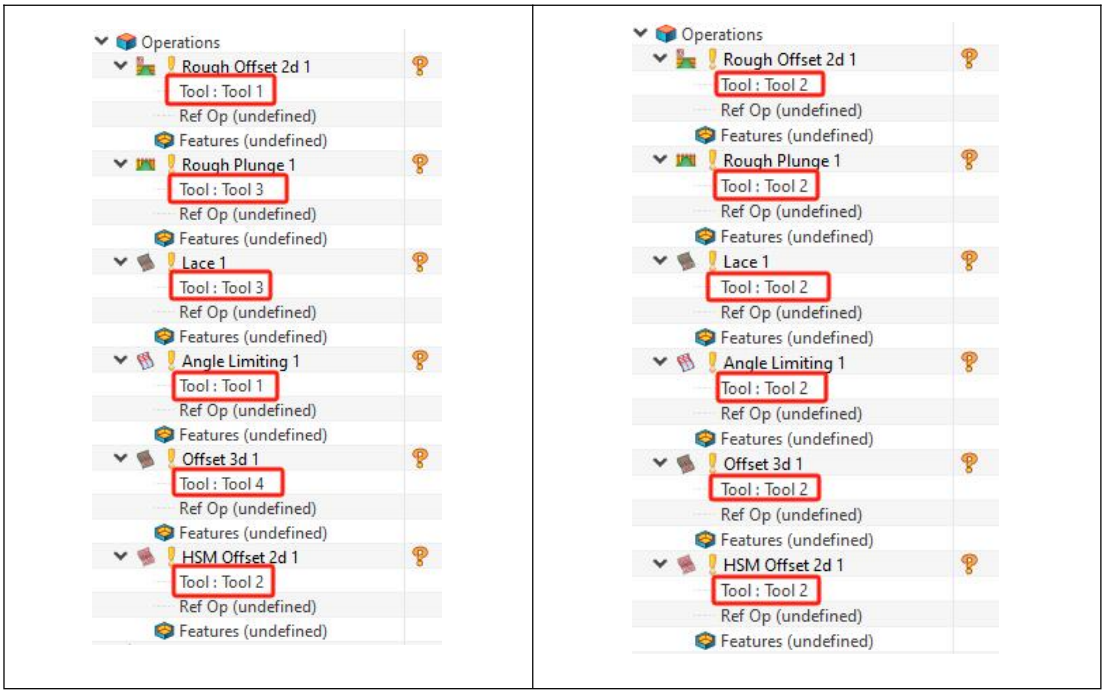
- ✓ Modify Stepover.
- ✓ Modify Thick.
- ✓ Modify Cutting Parameters.
- ✓ Modify Tool

**【Notes】**

- The new "Batch Modify Operation" function only takes effect for supported operations, and unsupported operations will not be modified.

**【Example】**

| Before use | After use |
| --- | --- |




**【Where is it】**

CAM Plugin >> Programme Tools >> **Batch Modify Operation**

### 5.4.3.2 Batch Rename Operation

The new "Batch Rename Operation" function allows you to edit and modify the names of selected processes in bulk.

**【What users can do】**

✓ The Operation can be renamed according to the rule of "Operation Name + Tool Name + Number";

**【Example】**

| Before use | After use |
|---|---|

**【Where is it】**

CAM Plugin >> Programme Tools >> **Batch Rename Operation**

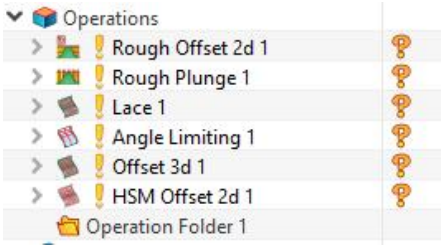
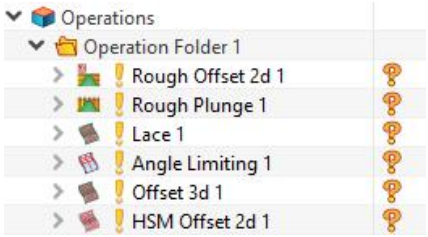
### 5.4.3.3 Batch Move Operation

The new "Batch Move Operation" function allows you to move the selected processes to different locations in bulk.

**【What users can do】**

✓ Move the selected operations into a specified folder.

**【Example】**

| Before use | After use |
|---|---|
|  |  |



**【Where is it】**

CAM Plugin >> Programme Tools >> **Batch Move Operation**

### 5.4.3.4 Batch Create/Edit Folder

The newly added "Batch Create/Edit Folder " function allows users to create a specified number of folders according to different rules.

**【What users can do】**

✓ Create a specified number of folders according to various rules.

**【Example】**

| | |
|---|---|
| Based on folder names | |

**【Where is it】**

CAM Plugin >> Programme Tools >> **Batch Create/Edit Folder**

## 5.4.4 Toolpath Verify Improvement

### 5.4.4.1 Toolpath Verify Directly Associated with the MCS

The default coordinate system of toolpath verify is adjusted from the WCS to the Tool Path which is MCS, without manual adjustment by the user.

**【What users can do】**

✓ The default coordinate system for toolpath verify is the Tool Path.

**【Where is it】**

Output >> **Toolpath Verify**

### 5.4.4.2 Turning Operation Supports Diameter Display

The diameter value is displayed during the toolpath verifying of the turning operations, so that users can see the commonly used data more intuitively, without the need for users to convert the radius value to the diameter.

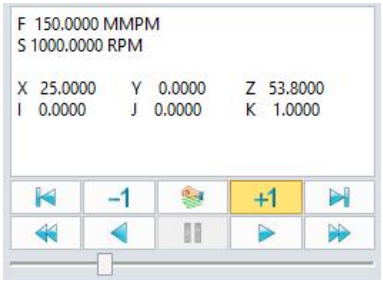
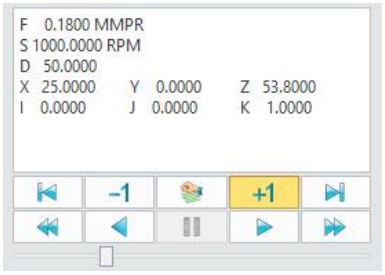
**【What users can do】**

- ✓ The tool path diameter is directly displayed in toolpath verification.

**【Notes】**

- Only the turning operation is supported

**【Example】**

| ZW3D 2025 | ZW3D 2026 |
| --- | --- |
|  |  |



**【Where is it】**

Output >> **Toolpath Verify**

## 5.4.5 Solid Verify Improvement

### 5.4.5.1 The Function of Process Stock Supporting Viewing the Final Result

After the solid verify, users can directly click to view the result of any operation, including the last operation, without selecting the last process and clicking Start verifying to get the final result.

**【What users can do】**

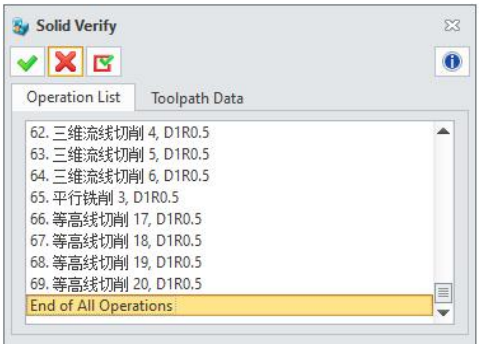
- ✓ Support users to directly click to view the results of any operation.

**【Notes】**

- You can only quickly click to see the operation that has been simulated.

**【Example】**

Click "End of All Operations" to view the final simulation results.




**【Where is it】**

Output >> **Solid Verify**

### 5.4.5.2 Visual Analysis Supporting Parameter Customization

The visual analysis function supports custom color and tolerance zone default parameters, which is convenient for users to set personal common color and tolerance zone.

**【What users can do】**

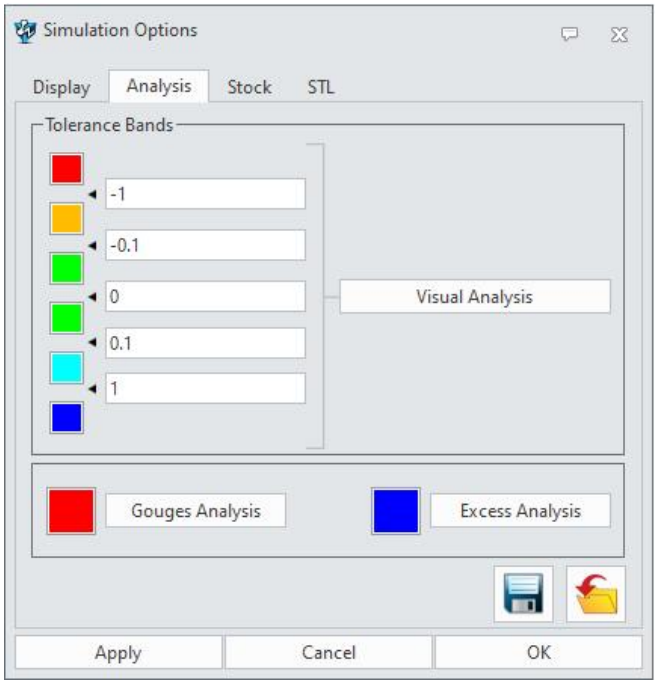
✓ Support custom color, tolerance zone default parameters.

**【Notes】**

- After the customization, you need to save the configuration file to the default folder path to define the default parameter.

**【Example】**

Customize visual analysis color and tolerance zone parameters by saving a profile or selecting a custom profile.




**【Where is it】**

Output >> Solid Verify >> Options >> **Analysis**

### 5.4.5.3 Stl Output Supporting Associative Model Paths

Supports automatic saving of Stl output to the folder where the model file is located after simulation, and users can also customize the saving path.

**【What users can do】**

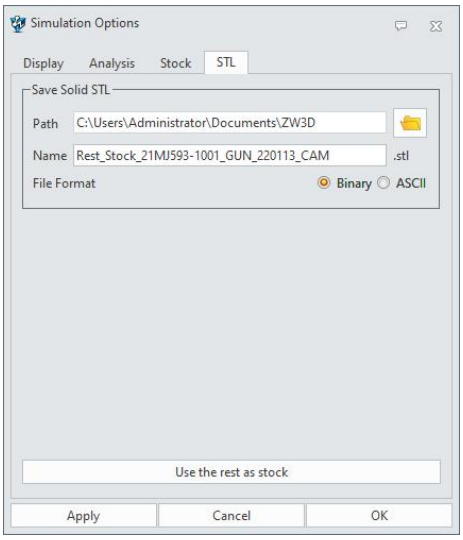
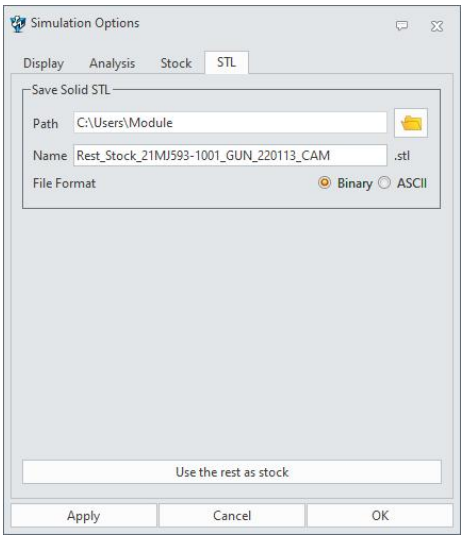
- ✓ The STL can be directly associated with the model file path.
- ✓ To save the Z3PRT format, you can directly associate it with the model file path.
- ✓ If the new file has not been saved, follow the set CAM output folder path.

**【Notes】**

- If the new file has not been saved, follow the set CAM output folder path.

**【Example】**

Before improvement, the CAM output folder is set by default, and after improvement, the folder where the model file is located is selected by default.

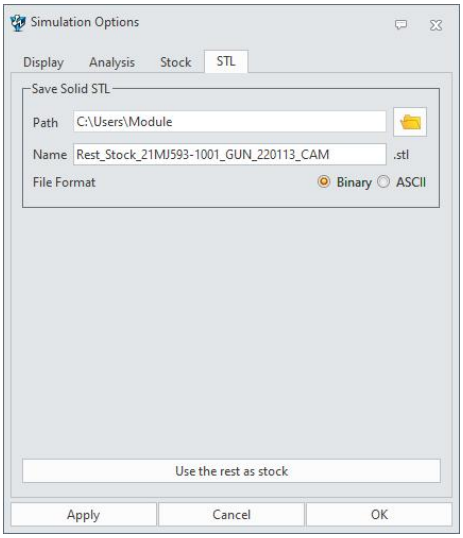
| Before | After |
| --- | --- |
|  |  |





**【Where is it】**

Output >> Solid Verify >> Options >> **STL**





## 5.4.6 New Post Debug Tool

The Post Debug Tool can help users configure ZNC files and can locate the corresponding ZNC statement and CL statement from the NC program for accurate modification.

**【What users can do】**

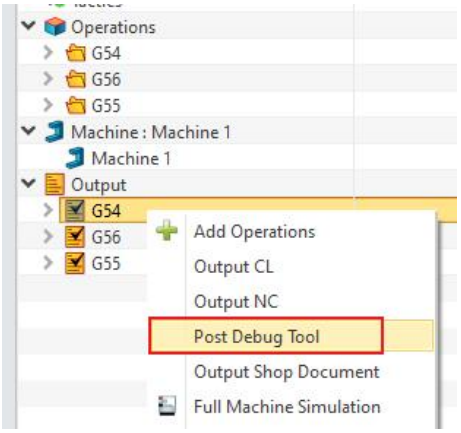
- ✓ The corresponding ZNC statement and CL statement can be located through the NC program.
- ✓ NC file, CL file, ZNC file, support search function.
- ✓ ZNC Function tree can display all ZNC functions, and support quick jump.
- ✓ Diff ZNC supports the ZNC text difference comparison function.
- ✓ Update the NC function.

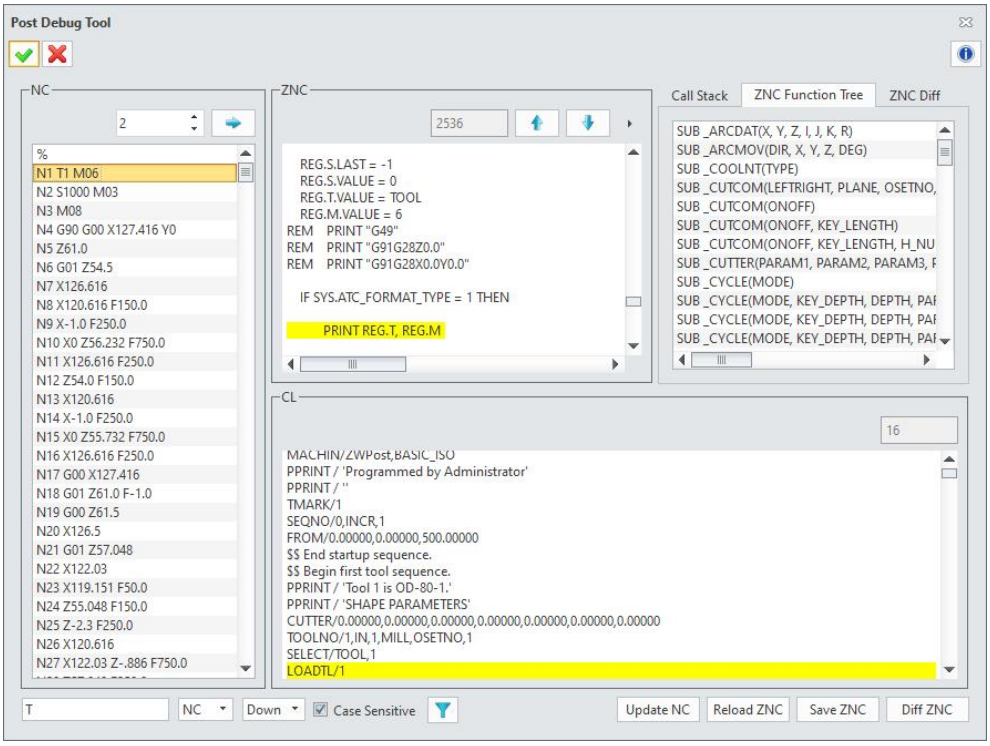
**【Notes】**

- Saving the ZNC directly overwrites the original ZNC file. Save and back up the ZNC file.

**【Example】**

In the Output NC folder, right-click and select the Post Debug Tool. After entering the tool interface, use the tool to find the NC statement that needs to be confirmed or modified. Click the NC statement, and the corresponding ZNC statement and CL statement will be highlighted. You can modify the ZNC file in the ZNC window, and then you can use the functions of "Update NC, Reload ZNC, Save ZNC, Diff ZNC".






## 【Where is it】

Output NC folder>> Header Right-click>> **Post Debug Tool**

# 6 API

## 6.1 Systemic Upgrade of API and Interface Supplement

### 6.1.1 Supplemented PMI Module Interfaces

The command-related interfaces of the PMI module have been improved, providing methods for creating baseline features, hole annotations, envelopes, stacked bubbles, etc., along with related interfaces. Additionally, for advanced requirements in PMI scenarios, interfaces related to discrete data and display data have been provided, supporting the processes such as 3D automated annotation, lightweight display, and object editing in secondary development scenarios. For details, please refer to the interface change list of ZW3D 2026 version.

**【What users can do】**

- ✓ Acquisition of various discrete data of PMI.
- ✓ Implementation of creating and editing PMI commands using interfaces.
- ✓ Knowledge of version changes based on the interface change list.

**【Where is it】**

Please check API Release Note and ApiHelpDoc.chm.

### 6.1.2 Supplemented Engineering Drawing Annotation Interfaces

The related interfaces for engineering drawing annotations have been improved, including not only the creation interfaces for objects such as elevation, text, bubble text, and stacked bubbles, but also the use interfaces for tools like switching references, magnetic lines, format brushes, and organizing annotations, enriching the capabilities for engineering drawing annotation and editing in secondary development scenarios. Furthermore, for discrete projection scenarios, the mutual query capabilities between 2D and 3D have been improved, and automatic matching of projections and drawing lines in discrete projections have been adapted. For details, please refer to the interface change list of ZW3D 2026 version.

**【What users can do】**

- ✓ More complete capabilities for engineering drawing annotation and editing.
- ✓ New interfaces adapted for discrete projection scenarios.
- ✓ Knowledge of version changes based on the interface change list.

**【Where is it】**

Please check API Release Note and ApiHelpDoc.chm.

### 6.1.3 Enhanced 3D Modeling Interface Capabilities

In addition to PMI and engineering drawing annotations, based on customer feedback, we have specifically improved interfaces related to 3D modeling, such as datums, history trees, assembly trees, and features, enhancing the capabilities for modeling and historical operations in secondary development scenarios. New editing capabilities for undefined datums have been added, along with interfaces for flexible editing of history trees and assembly trees, improving the capabilities for secondary development modeling.

**【What users can do】**

- ✓ Movement and editing of history tree objects (datums, features);
- ✓ Movement and editing of assembly tree objects.
- ✓ Knowledge of version changes based on the interface change list.

**【Where is it】**

Please check API Release Note and ApiHelpDoc.chm.

## 6.2 Chinese Interface Annotations

At present, the main customer base for ZW3D's secondary development is domestic developers. To optimize the user experience, lower the learning threshold, and improve the quality of annotations, we have optimized and translated the interface annotations of the English version and provided Chinese versions of interface annotations and help documents.

**【What users can do】**

- ✓ Chinese version of interface annotations and help documents.

- ✓ More accurate and clear interface annotations.

**【Where is it】**

Chinese version of interface annotations (ApiHelpDoc.chm Chinese) and help documents.

## 6.3 Scene Code Examples

New code examples for PMI and engineering drawing scenarios have been added, using specific functions as cases to facilitate user learning of ZW3D's secondary development mechanism. Through scenarios such as fillet replacement, automatic annotation, PMI formatting, etc., understand the creation and editing of features, the recognition of entities and the creation of annotations, and the acquisition and setting of PMI object properties.

**【What users can do】**

- ✓ Understanding of ZW3D's development approach in specific scenarios through examples.
- ✓ Rapid formation of plugins through code reuse.

**【Where is it】**

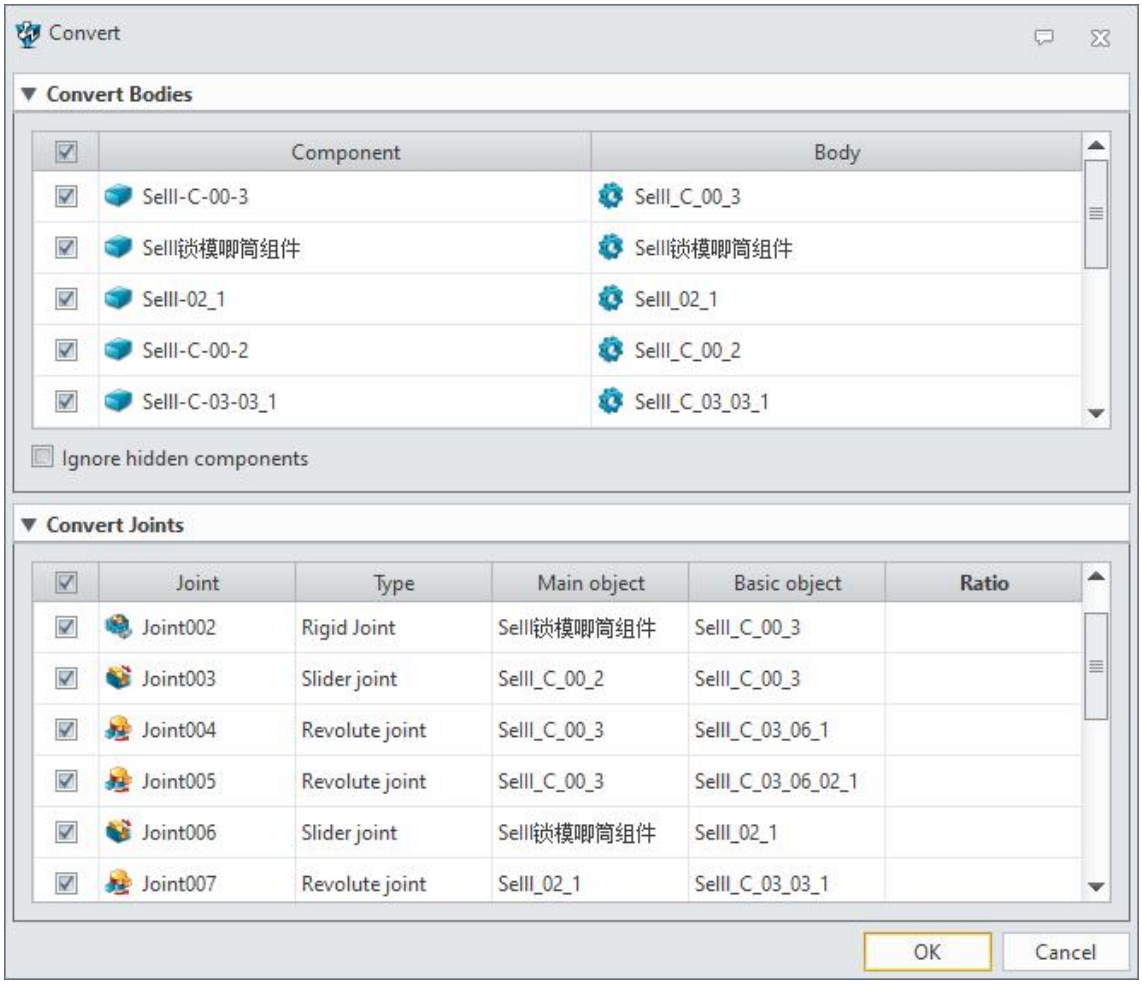
Code example resources.

# 7 Simulation

## 7.1 Motion

### 7.1.1 ★New Convert

The motion simulation has added a "Convert" feature, which supports the direct conversion of assembly constraints and kinematic pairs. The system will automatically recognize the predefined assembly constraints and provide a conversion interface, allowing users to select the corresponding kinematic pair for direct conversion, thereby eliminating the process of building the kinematic pair.



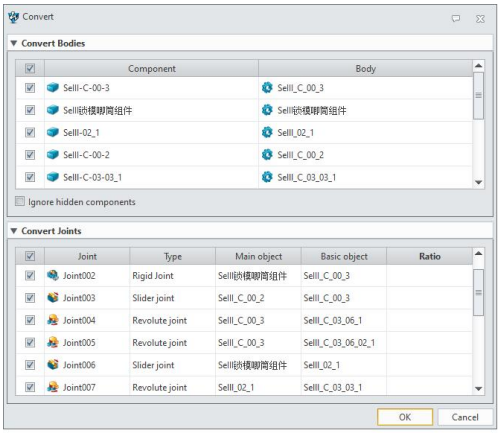

Converted interface

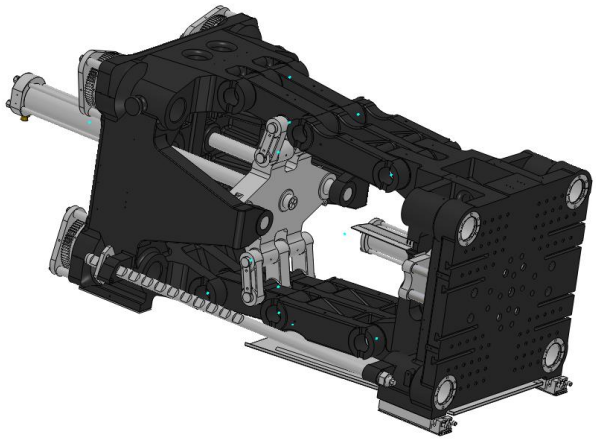
**【What users can do】**

✓ The assembly constraints defined in the assembly are directly used for simulation analysis.

**【Example】**

In the motion simulation analysis of the die casting system of the injection molding machine, if the assembly constraints have been defined in the assembly, you can directly use the conversion command to complete the creation of the moving body and joint.



**【Where is it】**

Motion >> Mechanism Setting >> **Convert**

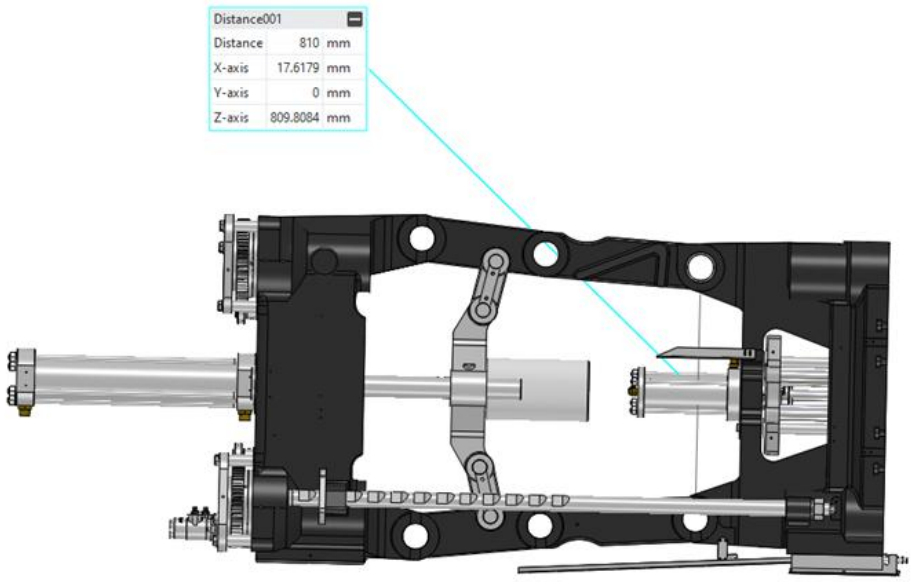
## 7.1.2 Improved Motion Analysis Ability

The motion simulation module supports faster and more efficient analysis and verification of the rationality of structural design.

**【What users can do】**

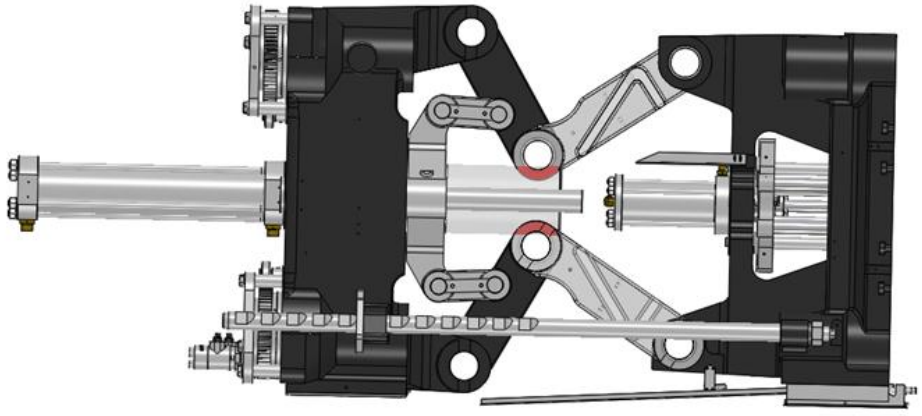
✓ Improved ability to measure motion distance

✓ When conducting motion simulation analysis, the distance between two moving bodies can be measured directly. The average improvement efficiency of inter-entity distance measurement reaches 80%.




✓ Dynamic interference capability is improved

✓ When motion interference occurs, the interference region can be displayed dynamically, and the efficiency of interference analysis can be increased to more than 70%.

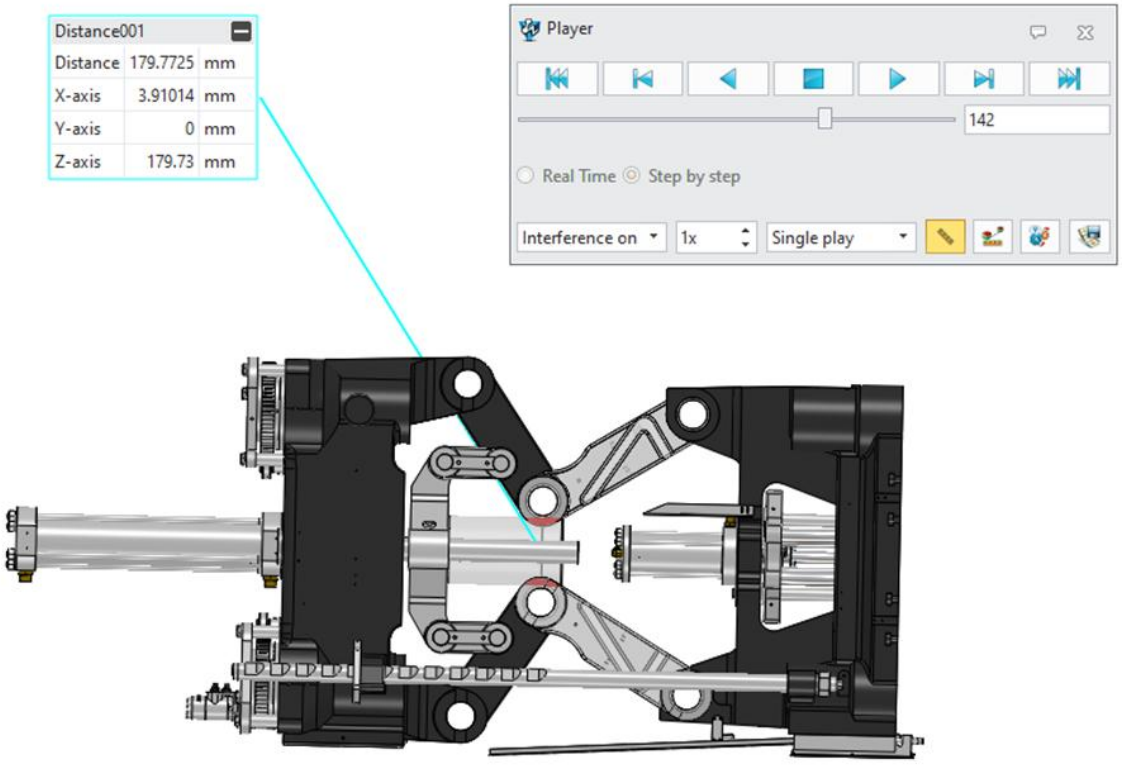


✓ Improved Band analysis efficiency

✓ The analysis efficiency of the band analysis function has been improved by more than 80%, and the analysis efficiency has been greatly improved.

**【Example】**

In the product design of injection molding machine equipment, the minimum distance and interference between the moving parts are detected and analyzed.




**【Where is it】**

Motion >> Inquire >> Measure >> **Distance**

Motion >> Animation >> **Interference Check**

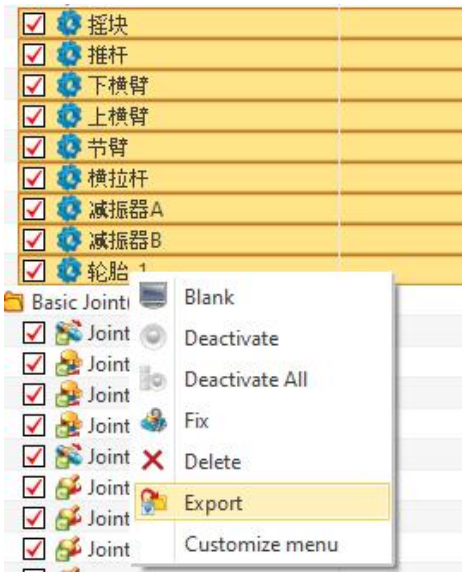
Motion >> Animation >> **Band Analysis**

## 7.1.3 Motion Data Export

After the motion simulation is completed, the obtained simulation data can be used for related design work. Motion supports exporting motion data of bodies' centroid. It is convenient for the designer to continue the follow-up work.

**【What users can do】**

✓ Export the motion data of bodies' centroid.

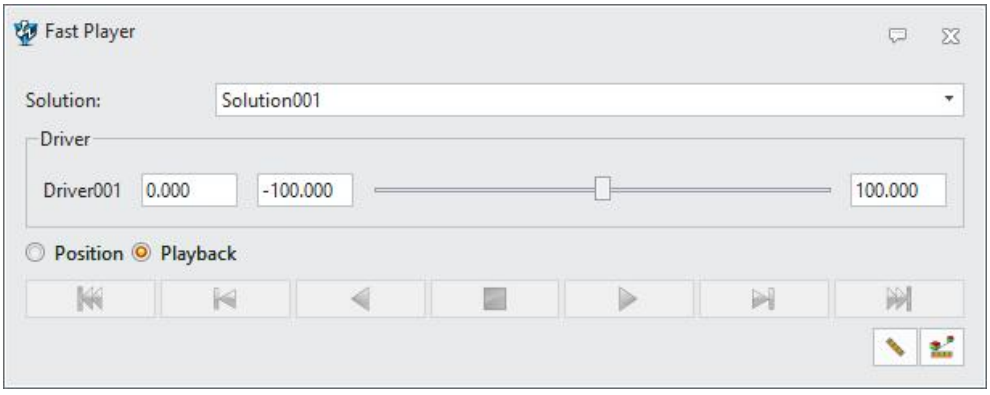



**【Where is it】**

Motion >> Motion node >> Body >> Right-click menu >> **Export**

## 7.1.4 Fast Player

A new "Fast Player" function has been added, supporting browsing and analyzing the positional relationship of mechanisms under a certain positional state.




**【What users can do】**

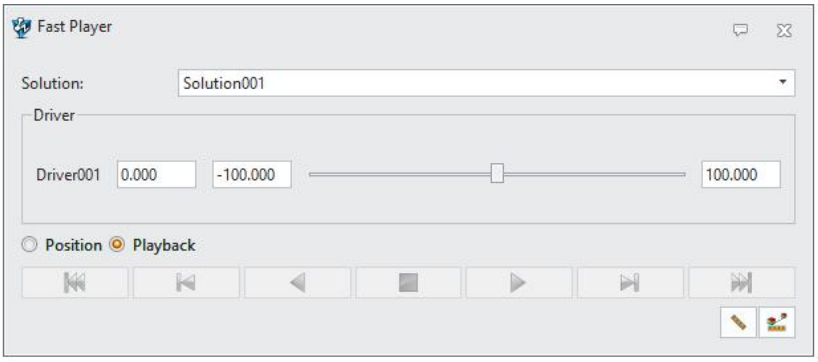
✓ The newly added fast player can support quickly adjusting the drive to a certain value without

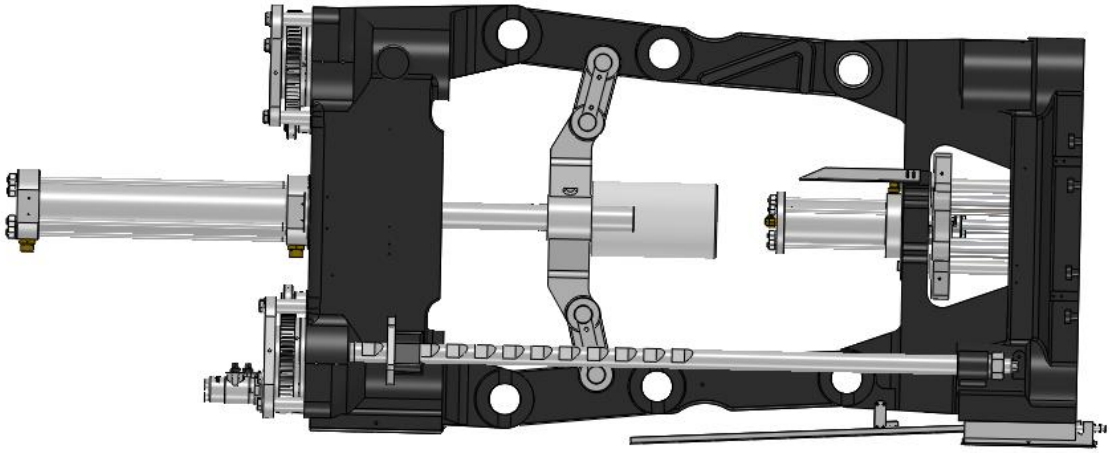
defining a drive function, so as to achieve rapid playback simulation of the mechanism.

✓ It supports directly rendering the model to a specified position, or using linear interpolation to play the motion animation of the mechanism between its starting and ending positions.

**【Example】**

In the motion simulation analysis of the injection molding machine equipment, the drive can be directly adjusted to the limit position state, and the interference between the moving parts of the position state can be viewed.



**【Where is it】**

Motion >> Animation >> **Fast Player**

## 7.2 Structural Simulation

### 7.2.1 Pre-processing

#### 7.2.1.1 Added Set Reference Function

The new version adds the ability to create connections and contacts by referencing predefined sets. Users can define the set which contains nodes, element faces, element edges, or entire elements, and then the constraints, loads, or contact conditions could be conveniently applied by referencing these sets. This eliminates the need for repetitive selection of the same boundary regions or contact surfaces, significantly streamlining the pre-processing workflow for complex models. This feature is particularly beneficial in scenarios where the same boundary or contact conditions need to be applied multiple times.

**【What users can do】**

✓ Improved Efficiency: In large assembly analyses, defining a set only once and referencing it multiple times reduces repetitive selection operations.

**【Notes】**

- Ensure that the set includes all relevant elements or nodes to avoid missing critical regions.

**【Where is it】**

Simulation >> Mesh Selection Tools >> **Node/Element Face/Element Edge/Element Set Pick**

#### 7.2.1.2 Added Material Manager

The new version introduces a Material Manager, enabling users to batch view and edit material parameters (e.g., Young's modulus, Poisson's ratio, density) and assign colors to each material. This enhances the efficiency and intuitiveness of material management in complex models.

**【What users can do】**

✓ Time Savings: Reduces material parameter adjustment time by over 30% in multi-part assembly analyses.

✓ Visual Validation: Color mapping helps quickly identify material misassignment issues.

**【Where is it】**

Simulation >> Properties and Expressions >> **Material Manager**

### 7.2.1.3 Added Mesh Adjacent Display Function

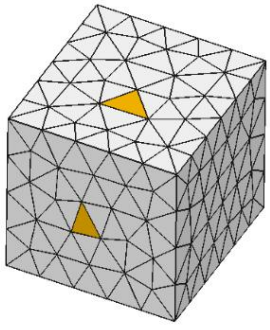
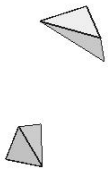
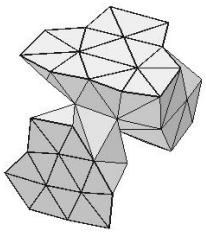
The tool of mesh adjacent display enhances mesh inspection efficiency in structural simulations. It identifies the neighboring regions of selected elements and automatically expands the display to show directly connected element layers, helping users quickly locate local mesh topology.

**【What users can do】**

✓ Focused Inspection: Breaks the limitations of traditional "full model display" by focusing on key areas and supporting layer-by-layer expansion of adjacent elements.

**【Notes】**

• This tool provides visual assistance only. Quantitative optimization should be performed using mesh quality metrics (e.g., Jacobian ratio, warpage).

| Select elements | Show only part of the mesh | Show adjacent elements |
|---|---|---|
|  |  |  |

**【Where is it】**

Simulation >> Toolbar >> **Show Adjacent Elements**

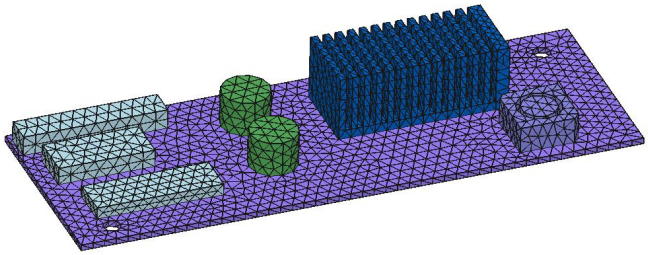
### 7.2.1.4 Support Classification to Define Mesh Colors

The new version supports color-coded mesh differentiation based on part type or material

properties. Users can directly modify material-associated colors in the Material Manager or adjust color configurations by part in the Mesh Collector. This feature enhances mesh visualization management, especially for large, complex assemblies (e.g., battery packs, ship structures), allowing users to quickly distinguish between material regions (e.g., metal/rubber), part types (e.g., bolts/shells), or element dimensions.

**【What users can do】**

✓ Quick Identification: Use colors to locate specific materials or parts, such as distinguishing battery cells (red), housings (gray), and bolt connections (yellow) in a 2-million-element battery pack model.



Display mesh colors by material

**【Where is it】**

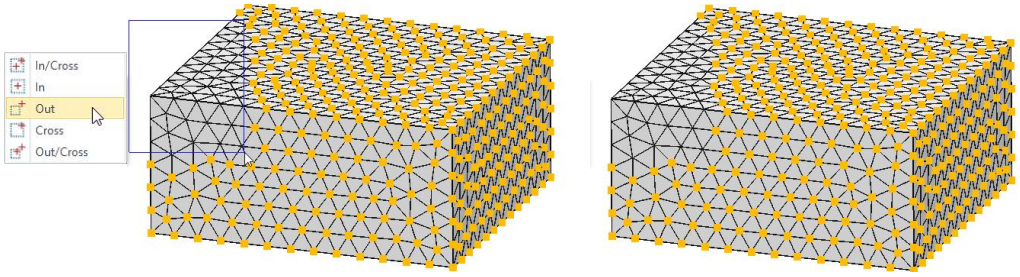
Simulation >> Toolbar >> **Collector Colors/Material Colors**

## 7.2.1.5 Optimized Mesh Picking Function

The new version enhances existing mesh tools with support for picking elements by plane, element type, or connected domain. Additionally, the box selection feature is optimized to allow picking surface-only meshes, as well as selecting interior, exterior, or intersecting parts of the selection box.

**【Notes】**

- For large models, batch operations are recommended to avoid real-time rendering lag.




**【Where is it】**

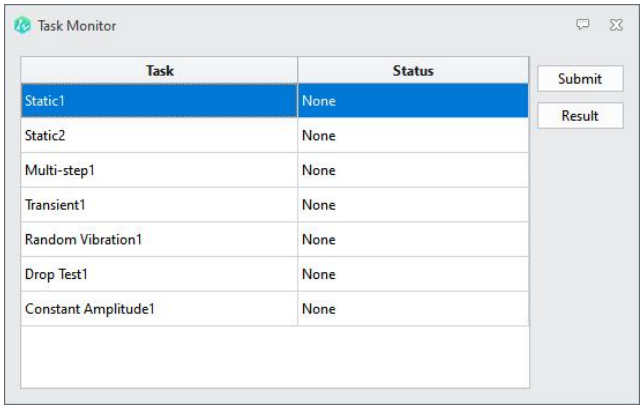
Simulation >> Mesh Selection Tools >> Select by Plane/Element Type/Connected Domain

### 7.2.1.6 Added Task Monitor

The new version adds a Task Monitor, enabling users to submit tasks, monitor progress in real-time, and jump to post-processing upon completion. This helps users efficiently manage and schedule simulation tasks.

**【What users can do】**

- ✓ Batch Submission: Provides an interface for batch job submissions.
- ✓ Real-Time Progress Visualization: Displays task status (e.g., running/completed/aborted).
- ✓ Post-Processing Integration: Automatically loads result files and jumps to the post-processing module upon task completion.




**【Where is it】**

Simulation >> Analyze >> Task Monitor

## 7.2.1.7 Added the Function of Automatically Creating Beams

Centerline extraction simplifies 3D solid structures into 1D lines representing their geometric centers, reducing computational effort and complexity in finite element analysis. This is typically used for structures with high aspect ratios.

**【What users can do】**

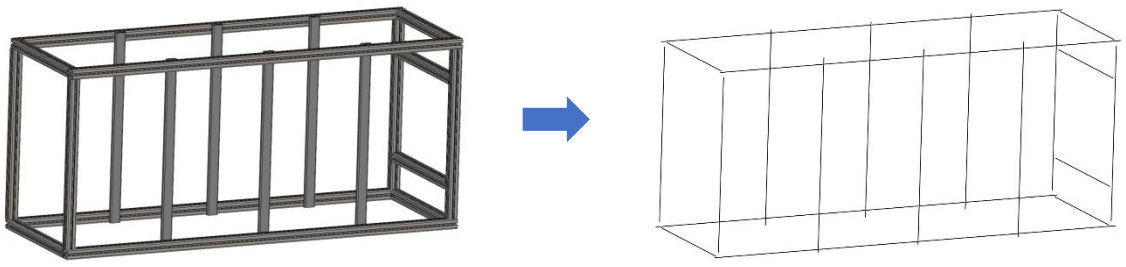
✓ Automatic Extraction: Extracts centerlines (e.g., beam/column axes, pipe centerlines) using geometric topology algorithms, supporting complex geometries like variable cross-sections and curved paths.

✓ Section Property Association: Automatically links beam section properties (area, moment of inertia, torsional constant) via geometric feature matching (e.g., circular, I-beam, box sections) or user-defined parameters.

✓ Model Simplification: Reduces 3D solids to 1D beam models, cutting element counts by up to 90% and significantly improving computational efficiency.

**【Notes】**

- For complex cross-sections, refine 2D meshes on beam ends before extraction.



Automatically creating beams

**【Where is it】**

Simulation >> Preprocessing >> **Automatically Create Beam**

## 7.2.1.8 Added the Function of Automatically Calculating Beam Section Parameters

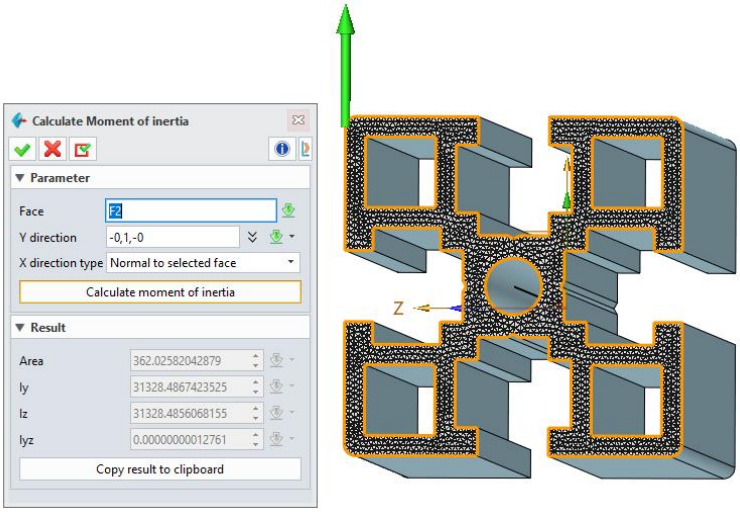
The new version integrates an intelligent beam section property calculation module, enabling automatic computation of section parameters (e.g., area, moment of inertia, torsional constant, shear center). Based on finite element discretization principles, this feature uses 2D meshing to analyze complex cross-sections, eliminating the inefficiency of traditional manual formula derivation or lookup tables. It is particularly suitable for designing irregular cross-sections in bridges, steel structures, and similar applications.

**【What users can do】**

✓ One-click output of key parameters, including section area, moment of inertia, shear area, and torsional constant.

**【Notes】**

• For complex cross-sections, refine 2D meshes on beam ends before extraction to get an accurate result.




Automatically calculating beam section parameters

**【Where is it】**

Simulation >> Geometry & Meshing >> **Calculate moment of inertia**

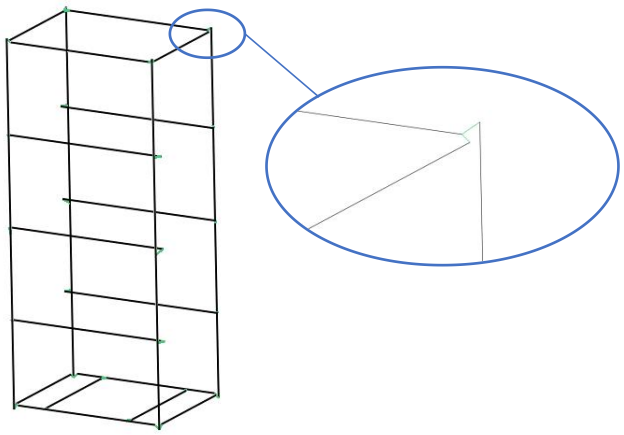
## 7.2.1.9 Added the Function of Automatically Creating Beam Connections

The automatic beam connection feature identifies adjacent beam connections in wireframe models and generates connection constraints based on defined rules. This reduces manual definition errors and improves modeling efficiency and accuracy for complex frame structures.

**【What users can do】**

✓ Batch processing hundreds to thousands of connections, reducing processing time from hours to minutes.

✓ Avoid omissions or incorrect degree-of-freedom settings in manual definitions, ensuring constraints align with actual working conditions.



Automatically connect the beam nodes

**【Where is it】**

Simulation >> Geometry and Meshing >> **Auto Joint Connection**

## 7.2.1.10 Added Mid-Surface Creation by Surface Offset

The new version introduces an automatic mid-surface generation feature based on surface offset. Using intelligent algorithms, it automatically identifies geometric pairs in thin-walled structures without manual surface selection. The system generates mid-surface geometry based on defined angle tolerance

and thickness range parameters. The resulting mid-surfaces can be directly used for shell element analysis and inherit the thickness properties of the original geometry.

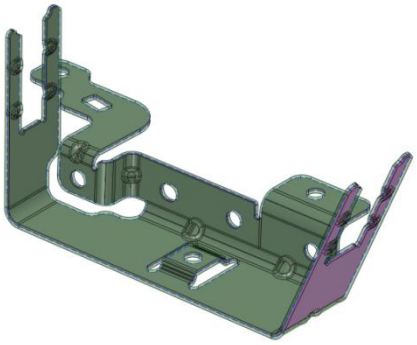
**【What users can do】**

✓ Reduces pre-processing time by over 80% for thin-walled structures like automotive sheet metal and plastic housings, avoiding the high computational cost of solid meshing.

✓ Mid-surfaces can be directly linked to shell element properties and passed to the simulation module for meshing and solving.

**【Notes】**

- Ensure the original model has no missing faces or gaps to avoid mid-surface generation failure or partial loss.
- Angle tolerance settings that are too small may miss non-strictly parallel faces, while settings that are too large may misclassify non-thin-walled regions.
- Define a reasonable thickness range based on the actual model to avoid ignoring overly thin regions or misprocessing overly thick regions.



Automatically extract the mid surface for the sheet metal parts

**【Where is it】**

Simulation >> Geometry and Meshing >> **Automatical Mid-Surface**

## 7.2.1.11 Added Virtual Topology Functions for Merging Faces and Suppressing Edges

The new version supports virtual topology to merge faces and suppress edges without modifying the actual geometry. This reduces the impact of geometric features on element quality, resulting in high-quality mesh elements.

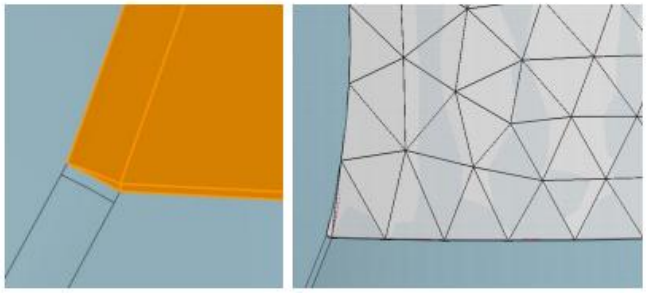
**【What users can do】**

✓ Merges small faces/edges to avoid mesh distortion or local densification caused by geometric complexity.

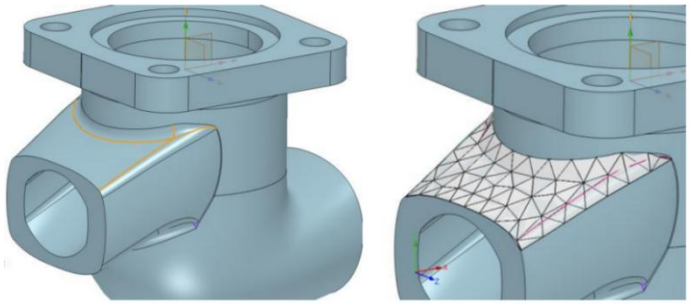
✓ Suppresses geometric details like fillets and chamfers that have minimal impact on analysis results, reducing computation time.

**【Notes】**

• After operation, preview the mesh to confirm the impact of virtual elements on node distribution, avoiding over-simplification that may lose critical features.



2D mesh results after virtual topology merge faces



2D mesh results after virtual topology suppress edge

**【Where is it】**

Simulation >> Geometry and Meshing >> Control >> **Merge Faces/Suppress Edges**

## 7.2.1.12 Added Mesh Renumbering Function

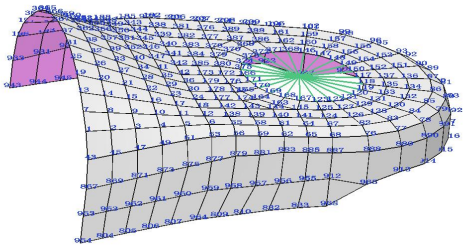
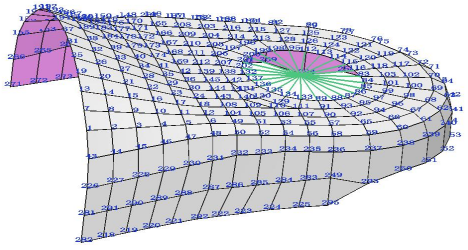
The new version supports renumbering discontinuous nodes or elements.

**【What users can do】**

✓ Resolves memory waste issues caused by ID jumps, improving computational efficiency.

**【Notes】**

• After renumbering, the boundaries of dependent nodes such as loads and constraints need to be updated synchronously.

| Before Node Renumbering | After Node Renumbering |
| --- | --- |
|  |  |

**【Where is it】**

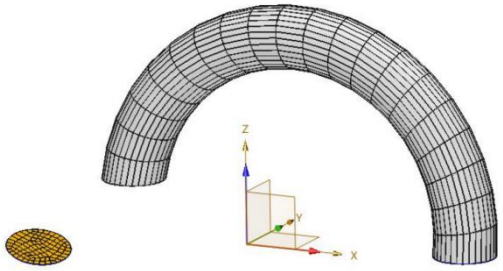
Simulation >> Geometry and Meshing >> Tool >> **Renumber**

## 7.2.1.13 Added Mesh Extrude Function

The new version introduces a mesh extrusion tool, which generates regular 3D meshes by extruding existing 2D meshes along a specified direction.

**【What users can do】**

✓ Supports linear extrusion (along normal/specified vector) and rotational extrusion (defined by axis and angle), meeting the expansion needs of complex geometries.



Mesh extrusion

【Where is it】

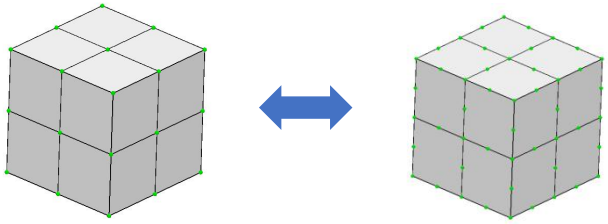
Simulation >> Geometry and Meshing >> Generate >> **Mesh Extrude**

## 7.2.1.14 Added the Function of Changing the Element Order

The new version adds the ability to change element order, such as switching from first-order to second-order elements. Users can flexibly adjust element order based on different analysis needs.

**【Notes】**

- After changing the order, the boundaries of dependent nodes such as loads and constraints need to be updated.



Changing the element order

【Where is it】

Simulation >> Geometry and Meshing >> Tool >> **Change Element Order**

## 7.2.1.15 Added the Function of Importing and Exporting INP files

The new version supports importing and exporting Abaqus solver input files (.INP), including most

common keywords like material definitions, boundary conditions, load types, contact algorithms.

**【What users can do】**

✓ Quickly migrate existing Abaqus models to ZW3D for simulation, reducing learning costs and repetitive modeling efforts.

✓ Supports smooth import of large-scale meshes.

**【Notes】**

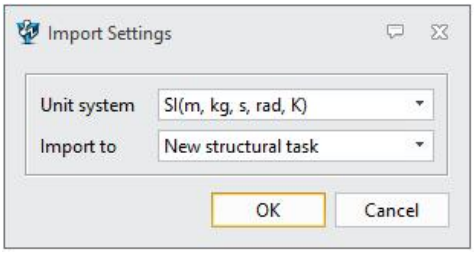
• Some advanced Abaqus features (such as user-defined subroutines, special element types) may require manual adjustments or are not yet supported.

**【Where is it】**

Simulation >> Solver >> **Import/Export**

## 7.2.1.16 Optimized Unit Selection for Simulation File Import

When importing solver files, the new version supports a combination of millimeters for length and tons for mass, improving data import flexibility.




New millimeter-ton system

**【Where is it】**

Simulation >> Solver >> Import >> **Unit System**

## 7.2.1.17 Added the Unit Selection Function for Material Library

The new version introduces unit switching in the material library, allowing users to directly switch between SI units, imperial units, and commonly used hybrid unit systems in the interface. This eliminates the need to convert material properties to fixed units before input.

**【What users can do】**

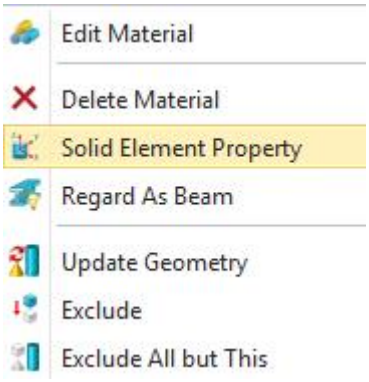
- ✓ Avoids calculation errors during manual unit conversion.
- ✓ Meets the unit switching needs of different industries or multinational teams.

**【Where is it】**

Simulation >> Material Library

## 7.2.1.18 Added the Function of regard as Beams

The new version adds "regard as beams" function, which enables users to simplify components with complex cross-sections into beam elements in geometric parts, facilitating subsequent simulation calculations.



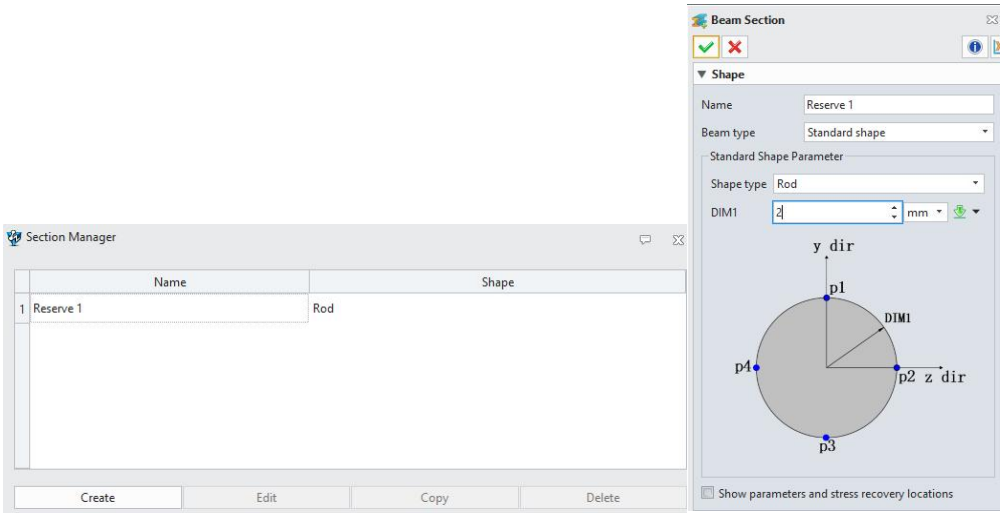

regard as beams

**【Where is it】**

Simulation >> Geometric component >> A certain component

## 7.2.1.19 Added the Function of Beam Section Manager

The new version adds a beam section manager function, which enables users to create beam elements in the geometric components and then assign corresponding geometric attributes to the sections of the beam elements. It can provide users with diverse section attribute settings.



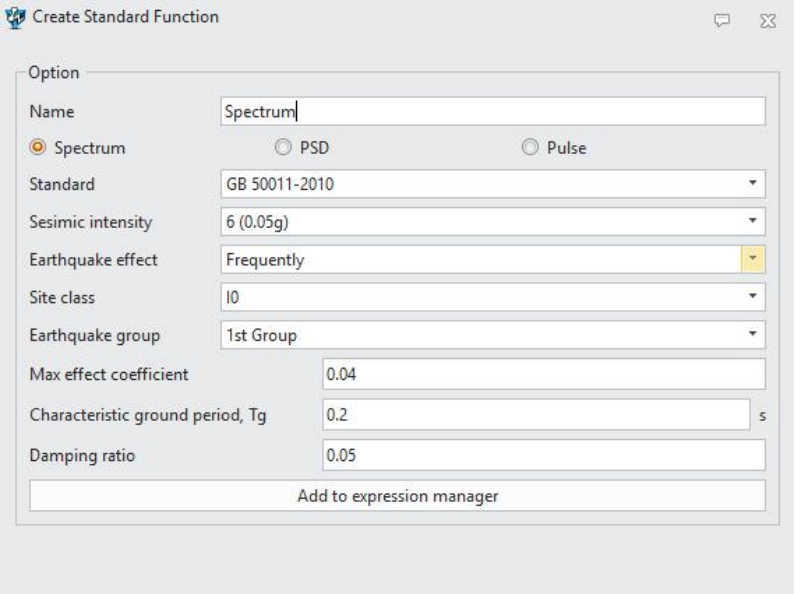

beam section manager

【Where is it】

Simulation >> Property Manager >> Section Manager

## 7.2.1.20 Added the Standard Function Calculator Function

The new version adds the function of creating standard functions, supporting the settings of spectrum, power spectral density and pulse function. In response to some professional scenario requirements of users, such as the calculation of seismic spectra, a method for calculating seismic spectra based on the current national standard (GB50011-2010) is provided, which significantly improves the usage efficiency of professional users and avoids complex calculations.




standard function calculator

**【Where is it】**

Simulation >> Tools >> Standard Functions

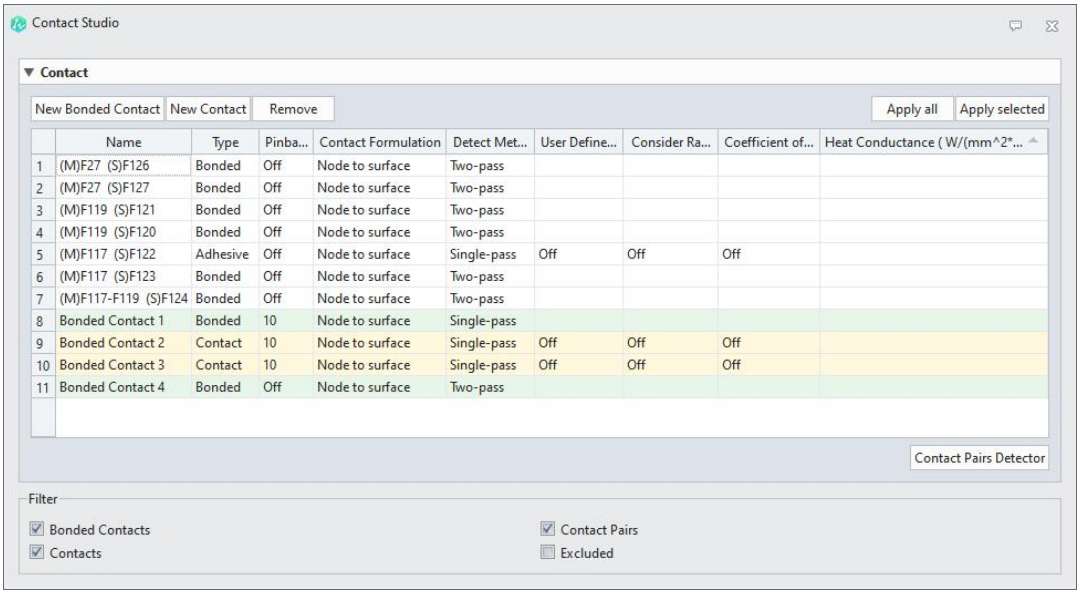
## 7.2.1.21 Added Contact Studio (ZW3D 2026 SP1)

The new version launches a new Contact Studio, which optimizes the contact detection algorithm and allows users to select an object, and the software will automatically find and list all surfaces that are in contact with it. It also provides a clearer batch management function of contact pairs, and stores all the contact pair structures generated by search in the Contact Studio. It supports users to batch apply contact types (such as binding, friction, etc.) and batch modify contact attributes (such as friction coefficient, penalty stiffness, etc.), and can more clearly view and check the detailed parameter attributes of each contact pair.

**【What can be done】**

✓ In the Contact Studio, users can batch select multiple or even all contact pairs, apply the same contact type to them at one time (such as setting all contact pairs as "bound" contacts) or modify the contact parameters uniformly (such as setting the friction coefficient in batches);

✓ The Studio clearly displays all contact pairs in the form of a list, and supports expanding each contact pair individually to visually view its associated geometric surfaces, currently applied contact types, friction coefficients and other key parameters.




| | Name | Type | Pinba... | Contact Formulation | Detect Met... | User Define... | Consider Ra... | Coefficient of... | Heat Conductance ( W/(mm^2*... |
|---|---|---|---|---|---|---|---|---|---|
| 1 | (M)F27 (S)F126 | Bonded | Off | Node to surface | Two-pass | | | | |
| 2 | (M)F27 (S)F127 | Bonded | Off | Node to surface | Two-pass | | | | |
| 3 | (M)F119 (S)F121 | Bonded | Off | Node to surface | Two-pass | | | | |
| 4 | (M)F119 (S)F120 | Bonded | Off | Node to surface | Two-pass | | | | |
| 5 | (M)F117 (S)F122 | Adhesive | Off | Node to surface | Single-pass | Off | Off | Off | |
| 6 | (M)F117 (S)F123 | Bonded | Off | Node to surface | Two-pass | | | | |
| 7 | (M)F117-F119 (S)F124 | Bonded | Off | Node to surface | Two-pass | | | | |
| 8 | Bonded Contact 1 | Bonded | 10 | Node to surface | Single-pass | | | | |
| 9 | Bonded Contact 2 | Contact | 10 | Node to surface | Single-pass | Off | Off | Off | |
| 10 | Bonded Contact 3 | Contact | 10 | Node to surface | Single-pass | Off | Off | Off | |
| 11 | Bonded Contact 4 | Bonded | Off | Node to surface | Two-pass | | | | |

**【Where is it】**

Simulation >> Contact >> Contact Studio

Simulation >> Contact >> Contact Pairs Detector

## 7.2.1.22 Added Various Mesh Editing Functions (ZW3D 2026 SP1)

The new version comprehensively enhances the mesh editing function and introduces a variety of practical tools including mesh translation, rotation, scaling, mirroring, projection, positioning, node placement, moving nodes, and remapping nodes. These functions significantly improve the user's flexible control over the mesh model in the pre-processing stage, allowing engineers to directly make necessary modifications to the existing mesh without returning to the geometric modeling stage to better meet analysis needs or improve mesh quality.

**【What can be done】**

✓ For structures with symmetrical or periodic characteristics, such as rotating machinery,

symmetrical supports, and arrayed holes, users can prioritize performing fine mesh division on one of the basic units (e.g., a sector or a period). Subsequently, by utilizing functions such as mirroring, rotation, and translation, they can swiftly replicate and assemble these high-quality meshes to generate the mesh for the complete model.

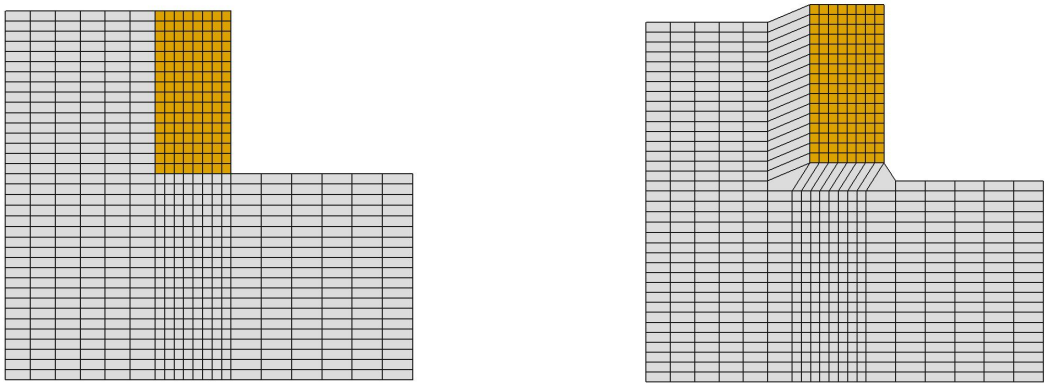
✓ By using the mesh translation, rotation, and scaling tools, users can easily adjust the position, orientation, and size of local meshes.

✓ The function of placing nodes and moving nodes allows users to directly "manually" fine-tune mesh nodes.These can be used to repair locally distorted elements, make the mesh better fit complex geometric contours, or implement more precise mesh control in critical areas (such as stress concentrations);

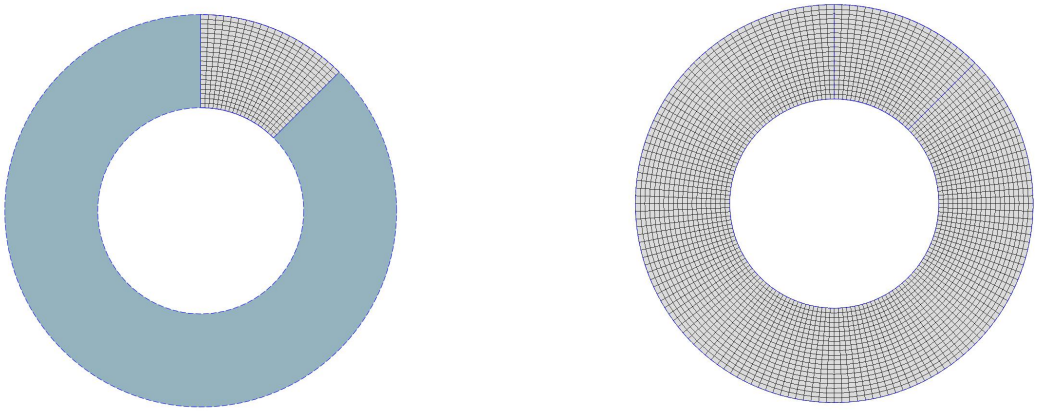
✓ The projection function enables rapid projection of mesh nodes located on flat or simple curved surfaces onto complex target surfaces (such as free-form surfaces), ensuring the mesh precisely conforms to the geometry's shape.

**【Notes】**

- Current mesh editing operations primarily affect the mesh itself, potentially weakening or severing the connection between the mesh and the original geometry;
- When using functions such as mirroring, rotation, and copying, ensure that the mesh nodes in the copy interface are continuous. If shared nodes are required, additional node merging operations may be necessary to guarantee the correct transfer of mechanical responses.



Translate



Rotate

【Where is it】

Simulation >> Pre-Processing >> Mesh

## 7.2.1.23 Added Batch Geometry Repair Function (ZW3D 2026 SP1)

The new version supports batch geometry repair functionality, enabling simultaneous correction of multiple geometry quality issues including elongated faces, self-intersecting faces, small faces, small edges, and open edges. Its core value lies in transforming repetitive manual operations — previously requiring individual handling—into an efficient automated workflow.

**【What can be done】**

✓ By batch-repairing elongated and small faces, cells with poor geometry (such as excessive aspect ratios) can be avoided, significantly improving mesh quality.

✓ It can quickly handle open edge issues throughout the entire model. This ensures the geometric integrity of the model at connection points.

**【Notes】**

• It is recommended to proceed by first repairing minor features (such as small edges and faces), followed by addressing complex topological issues (such as self-intersecting faces and open edges). Additionally, the software typically provides tolerance parameters for user adjustment; a reasonable repair tolerance should be set based on the model's overall dimensions. Excessively low tolerance

may result in incomplete repairs, while excessively high tolerance may alter critical geometric features.

- The batch geometry repair function will clear the model's geometry history tree. It is recommended to back up the model before performing geometry processing to avoid losing important historical features.

**【Where is it】**

Simulation >> Pre-Processing >> Geometric processing

### 7.2.1.24 Added Batch Geometry Simplification Function (ZW3D 2026 SP1)

The new version supports batch geometric simplification, designed to automatically identify and process numerous minor details on models — such as holes, fillets, chamfers, and logos — that have minimal impact on structural mechanical properties. Through intelligent recognition algorithms, it batch-selects similar features and employs geometric repair algorithms to automatically simplify them. This preserves the primary mechanical characteristics of the structure while yielding a cleaner geometric model better suited for high-quality mesh generation.

**【What can be done】**

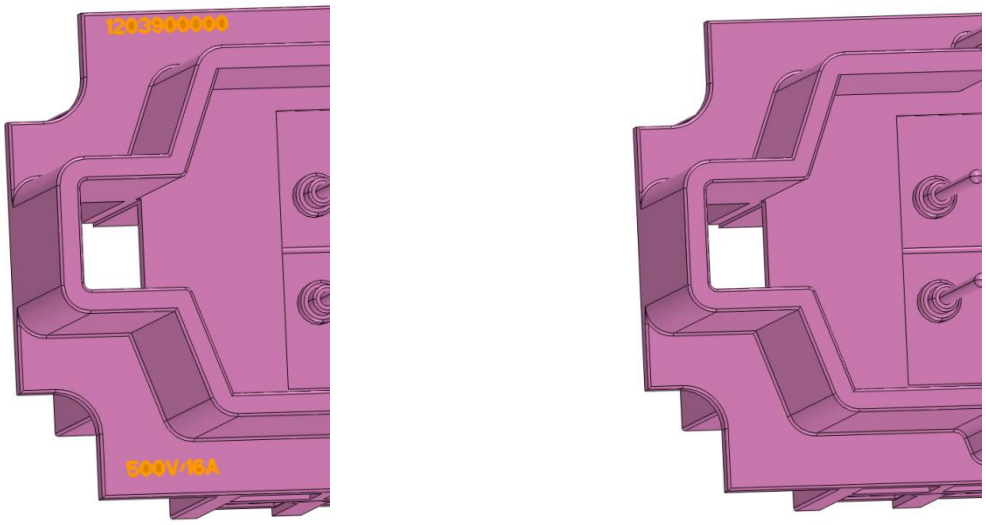
✓ By batch-removing dense features such as small holes and rounded corners from the model, the number of mesh elements can be significantly reduced, preventing excessive fine elements from forming in these local details.

✓ For complex assemblies containing numerous standard components (such as screws and bolts), their models typically feature a multitude of threaded holes, mounting holes, and similar details. The batch simplification function enables rapid cleanup of these repetitive detail features.

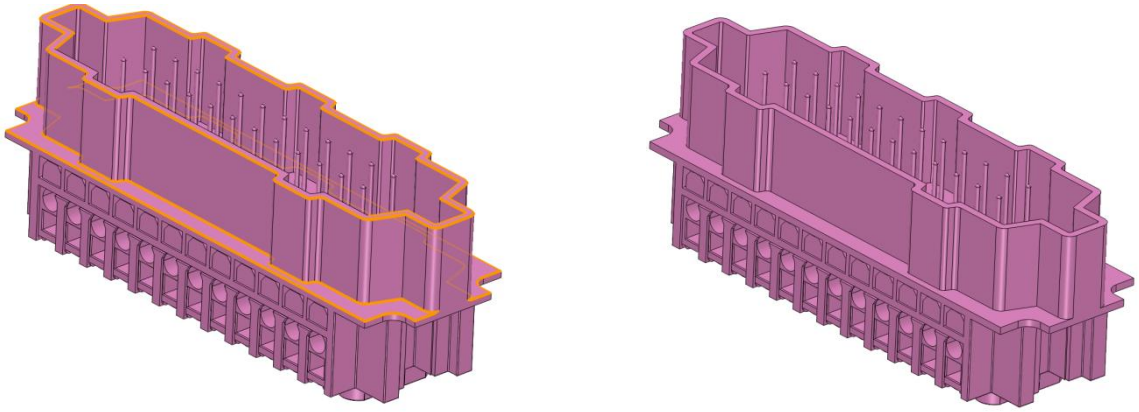
**【Notes】**

- The batch geometry repair function will clear the model's geometry history tree. It is recommended to save or back up the original model before proceeding, to enable rollback or comparative analysis if needed.
- Setting tolerance parameters appropriately; conduct small-scale trials before implementing

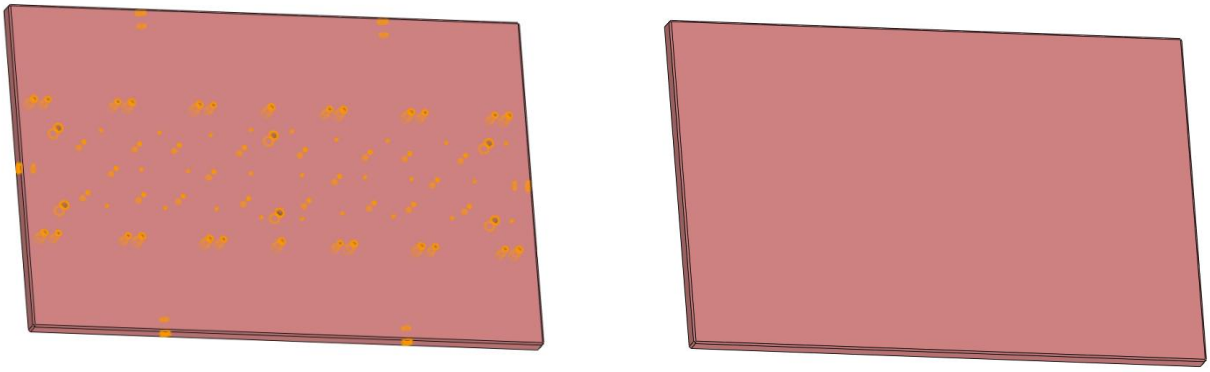
them on a large scale.




Batch logo removal



Batch fillet removal



Batch hole removal

**【Where is it】**

Simulation >> Pre-Processing >> Import/Export

## 7.2.1.25 Added Batch Bolt Recognition and Creation Functionality (ZW3D 2026 SP1)

The new version supports batch recognition and creation of bolts. The software automatically identifies these bolts based on user-entered names and enables one-click batch creation of bolted connections within the simulation environment. These connections are modeled using beam elements and dynamic rigid elements.

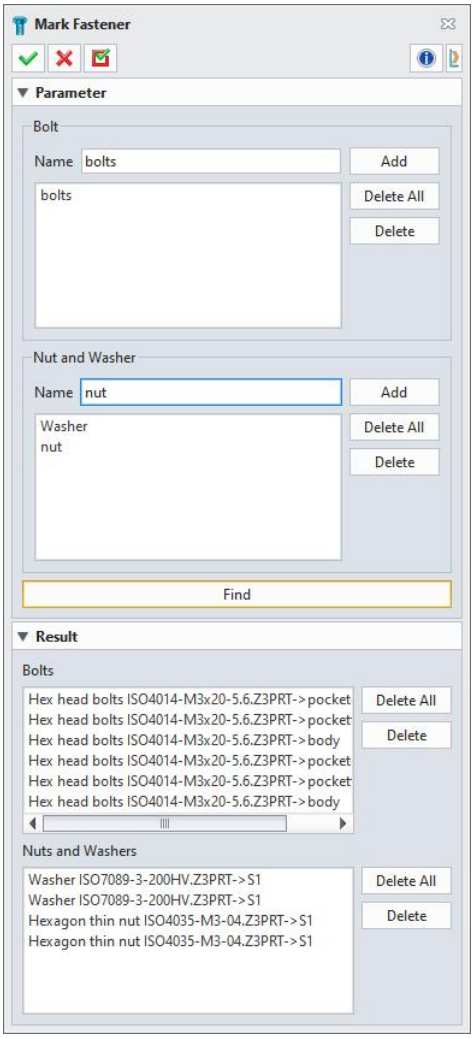
**【What can be done】**

✓ For assemblies containing numerous bolted connections, this feature reduces creation time from hours to minutes. Users no longer need to set connection relationships individually; simply ensure component names are standardized, and the software will automatically generate all bolted connections in bulk；

✓ Automatically generated bolt connections and related parameters are integrated into the software's Bolt Manager, enabling users to conveniently view, modify, and manage them in a unified manner.

**【Notes】**

- Ensure that the naming of components such as bolts, nuts, and washers in the model complies with the software's recognition specifications.
- For non-standard holes or special cases where the bolt axis is not perpendicular to the contact surface, the automatically generated results may not be applicable.




Mark Fastener

## 【Where is it】

Simulation >> Pre-Processing >> Marking Fasteners

Simulation >> Simulation Tree >> Connection >> Auto Create Bolt

## 7.2.1.26 Added Automatical Face Extension Function (ZW3D 2026 SP1)

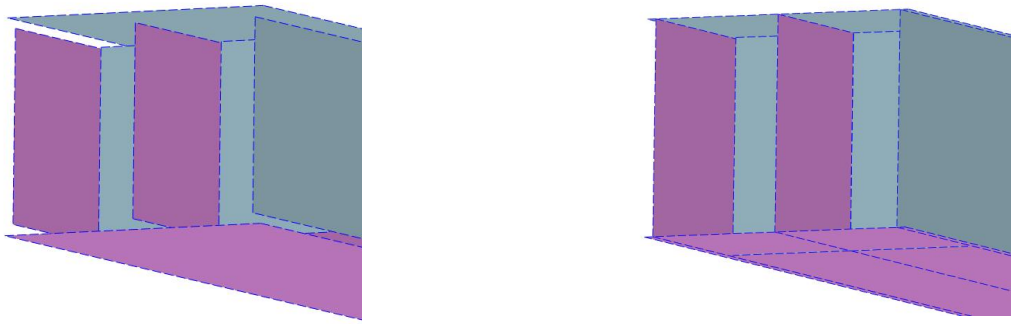
The new version supports automatic face extension functionality, allowing users to designate one or more faces as reference surfaces and set a distance tolerance. The software automatically searches the model for other geometric faces adjacent to the reference surface boundary and within the tolerance range. Upon locating target geometry, the system automatically extends the reference surface to intersect with the detected geometry, achieving automatic face repair and extension.

**【What can be done】**

✓ After the model automatically extracts intermediate surfaces, the generated surfaces may contain small gaps or broken faces. This feature allows you to quickly select adjacent surfaces with gaps as base surfaces, then bridge the gaps through automatic extension.

**【Notes】**

• After completing the surface extension operation, verify that the direction and outcome of the newly generated geometric extension align with expectations.



Automatical Face Extension

**【Where is it】**

Simulation >> Pre-Processing >> Automatical Face Extension

#### 7.2.1.27 Optimized INP Import Functionality (ZW3D 2026 SP1)

The new version of the software features significant enhancements to its INP file import functionality. It now parses INP file syntax and structure with greater precision, enabling complete model information—including meshes, material properties, loads, and boundary conditions—to be imported into the simulation environment with higher fidelity.

**【Notes】**

- After importing the model, it is still necessary to carefully verify whether the key parameters have been correctly applied;
- When importing a model, you must select the corresponding unit system.

**【Where is it】**

Simulation >> Import

### 7.2.2 Solver

#### 7.2.2.1 Added Multiple Load Case Analysis of Linear Statics and Harmonic Response

A new module for multiple load case analysis in linear statics and harmonic response is added. It allows users to create multiple independent load cases under a unified analysis framework, defining different loading conditions within the same linear static analysis or harmonic response analysis step. Intermediate results from prior analysis steps (e.g., modal calculations) are automatically inherited, avoiding redundant calculations.

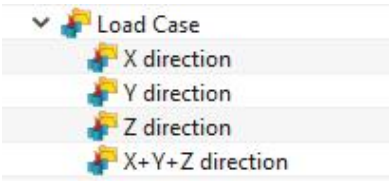
**【What users can do】**

- ✓ By reusing modal calculation results, harmonic response analysis time is reduced by 30%-50% (depending on model complexity).
- ✓ Simultaneously simulate structural responses under combined conditions such as self-weight, wind load, and thermal load in linear statics.
- ✓ Harmonic response analysis supports the study of vibration characteristics under

multi-frequency excitation force combinations.

**【Notes】**

- Multiple load cases must satisfy the principle of linear superposition. Nonlinear cases require a separate analysis module.




Define different load cases in the same analysis

**【Where is it】**

Simulation >> New Structural Simulation >> Multi-step Analysis >> **Load Cases**

### 7.2.2.2 Added Bolt Pre-load Function

The new version introduces the bolt pre-load function in multi-step analysis. Users can set the preload force or preload length of bolts in analysis steps to achieve pre-tightening effects. In subsequent steps, the bolt length can be fixed to simulate assembled bolt connections. This function is widely used for assembly structures, power transmission devices, and critical connection components.

**【What users can do】**

✓ Accurately simulate the mechanical behavior of bolted connections, and reproduce the entire process from bolt pre-tightening to assembly fixation to working load superposition through step-by-step loading, avoiding the errors of traditional single-step analysis.

✓ Evaluate the impact of preload force on connection performance, such as increasing stiffness, reducing vibration-induced loosening risk, and optimizing sealing performance (e.g., airtightness verification for flange connections).

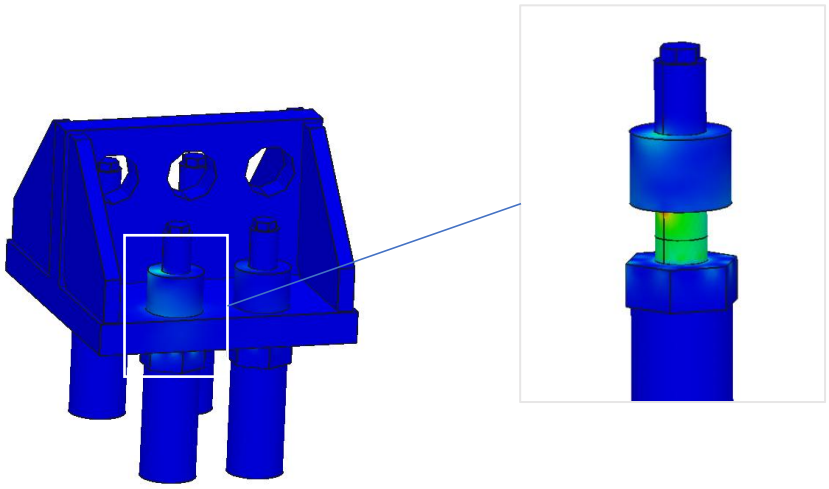
✓ Compare models with and without preloading to provide a basis for bolt selection.

**【Notes】**

- The bolt shaft must be meshed with at least two layers of elements along its axis to ensure

stress distribution accuracy.

- The bolt shaft mesh must be compatible with surrounding elements.
- Preload force must be applied in the initial loading step, with subsequent steps only allowing "fix as current length" or applying additional external loads.



Bolt Pre-load Function

**【Where is it】**

Simulation >> Multi-step Analysis >> Load >> **Bolt Pre-load**

### 7.2.2.3 Added Hyperelastic and Hyperfoam Materials

The new version introduces a hyperelastic material model library, covering six classic constitutive models: Mooney-Rivlin, Ogden, Neo-Hookean, Polynomial Form, Arruda-Boyce, Yeoh. Additionally, the HyperFoam model was introduced to support large deformation analysis of compressible foams (e.g., Ogden Foam model).

A built-in experimental data fitting tool allows automatic fitting of hyperelastic and foam material parameters using test data from uniaxial tension/compression, biaxial tension, shear, and volumetric deformation experiments.

**【What users can do】**

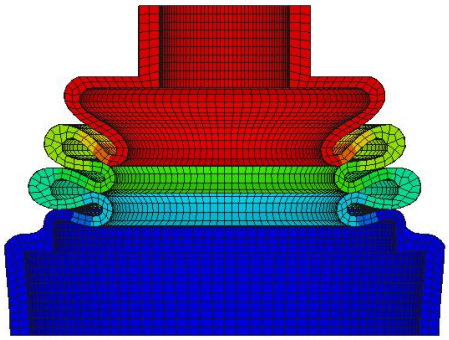
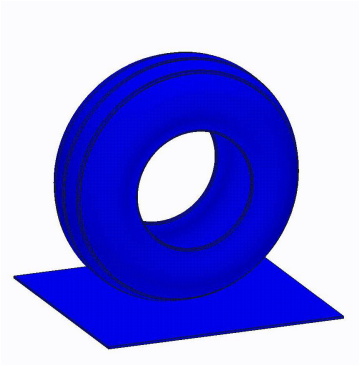
- ✓ Fit material parameters based on multiple experimental modes, covering complex deformation

scenarios.

✓ Support foam material volumetric deformation fitting, addressing simulation challenges for compressible materials (e.g., porous rubber).

**【Notes】**

- The data must cover the simulation strain range. For example, to predict 150% strain, test data ≥150% must be provided to avoid extrapolation errors.
- For small strains (<40%), Neo-Hookean is recommended. For large strains (300%-700%), Ogden or Arruda-Boyce is preferred.

| Rubber dust plug model | Tire inflation loading |
| --- | --- |
|  |  |

**【Where is it】**

Simulation >> **Material Library**

## 7.2.2.4 Added Composite Hashin Failure Criterion

The new version introduces the Hashin failure criterion module based on the classic Hashin theory framework, enabling high-precision simulation of progressive failure behavior in fiber-reinforced composites. It supports four typical failure modes:

Fiber Tension Failure: Triggered when tensile stress in the fiber direction exceeds the longitudinal tensile strength.

Fiber Compression Failure: Buckling or shear failure due to compressive stress in the fiber direction.

Matrix Tension Failure: Interfacial debonding or matrix cracking due to transverse tensile stress.

Matrix Compression Failure: Shear or extrusion failure due to transverse compressive stress.

**【What users can do】**

✓ Provides visualization of failure modes, distinguishing fiber/matrix damage regions and severity.

**【Where is it】**

Simulation >> Simulation Tree >> Laminate Properties >> **Failure Criteria**

### 7.2.2.5 Added Inertia Relief Function

A new inertia relief function is added in the linear static module to address rigid body displacement issues in unconstrained/under-constrained models. The core principle involves introducing an inertia force field that balances external loads, achieving self-equilibrium in the model.

**【What users can do】**

✓ Automatically calculates the centroid as a reference point or allows manual selection of any node/coordinate system for inertia force calculations.

✓ Converts rigid body acceleration response under quasi-static loads into equivalent inertia forces, avoiding stress distortions caused by the traditional pseudo support reaction method.

**【Notes】**

- Material density must be defined (as it is fundamental for inertia force calculations).
- Reference point selection principles:

Automatic mode: Recommended for homogeneous structures composed of isotropic materials.

- Manual mode:

For bolted structures: Select the bolt hole center.

For transmission systems: Select the gear meshing center.

【Where is it】

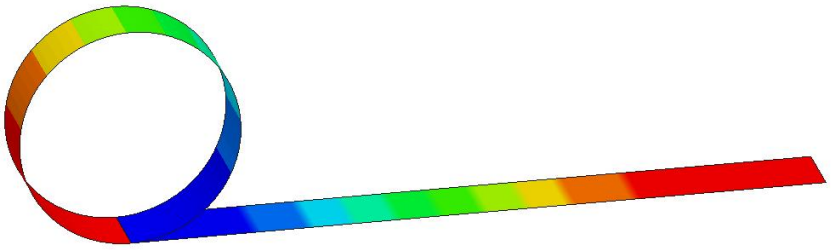
Simulation >> Statics >> **Inertia Relief**

## 7.2.2.6 Added First-order Large Deformation Shell

The new version supports large deformation analysis for first-order shell elements, including a plastic material constitutive model. It can handle buckling, post-buckling, and large deflection responses under complex working conditions. Compared to traditional higher-order shell elements, first-order elements reduce computational cost while maintaining accuracy.

**【What users can do】**

✓ Supports analysis involving contact, material nonlinearity (plasticity), and geometric nonlinearity (large displacement, large rotation), such as metal stamping and thin-walled structure buckling.



First-order shell large deformation

【Where is it】

Simulation >> Task Options >> **Geometric Nonlinearity**

## 7.2.2.7 Added Rigid Body Definition

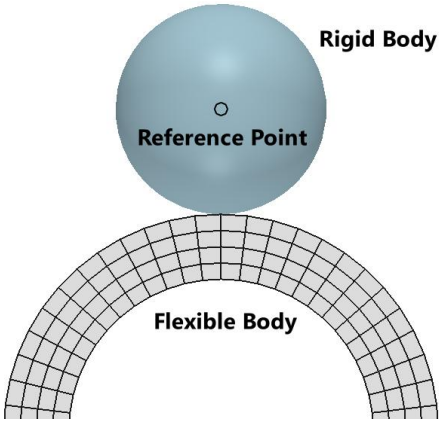
The new version introduces rigid body definition, allowing a part of the model to be defined as a rigid body. It also adds hinge constraints (rotational degree of freedom control), sliding constraints (axial degree of freedom release), and fixed constraints (full degree of freedom locking), enabling the simulation of complex multibody system motion relationships.

**【What users can do】**

✓ Simplifies models with rigid bodies, reducing the need for local mesh refinement and shortening simulation time for large assemblies.

**【Notes】**

- Rigid body regions can use coarse meshes, but the mesh quality of flexible body regions must be ensured.




**【Where is it】**

Simulation >> Connections >> **Rigid Body**

## 7.2.2.8 Added Timoshenko Beam Element

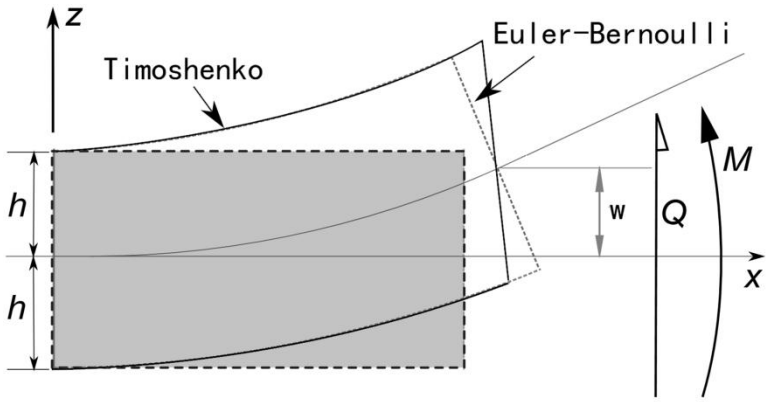
The new version introduces the Timoshenko beam element, which considers transverse shear deformation effects. It provides more accurate stress and deformation results for short and thick beams.

**【What users can do】**

✓ More accurate stress and deflection calculations for short beams, significantly reducing errors compared to Euler beams.

**【Notes】**

- For slender beams with a span-to-depth ratio greater than 20, switching to Euler beams is recommended to save computational resources.




【Where is it】

Simulation >> **Beam Element Properties**

## 7.2.2.9 Added Surface-to-Surface Contact Method for Bonded Contact

The new version adds a surface-to-surface contact method for bonded contact, providing a more accurate contact processing approach. Compared to the traditional node-to-surface contact algorithm, it offers higher computational accuracy and stability, particularly in cases of large deformation or complex contact regions.

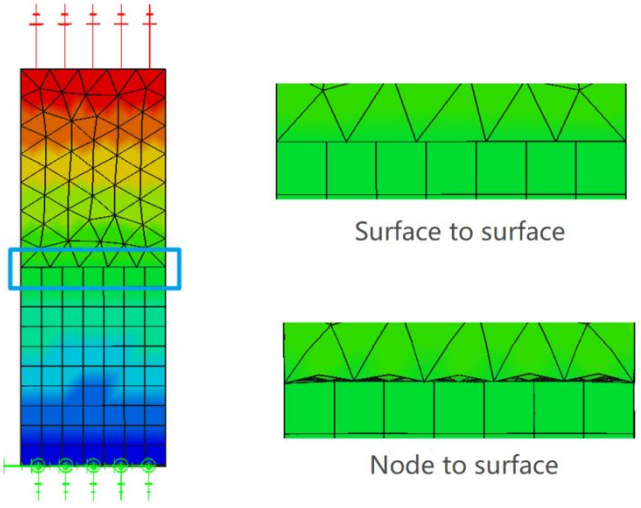
**【What users can do】**

✓ Eliminates stress jumps at the node level seen in node-to-surface methods, ensuring a smooth transition of contact pressure distribution.

✓ Prevents contact failure due to master surface penetration into the slave surface, addressing issues like mesh misalignment and missed penetration detection in traditional algorithms.

**【Notes】**

• The surface-to-surface method requires more computational resources than node-to-surface; it is recommended to enable it primarily in critical contact regions.




The simulation result improved by using surface to surface contact

**【Where is it】**

Simulation >> Contact >> Bonded Contact >> **Contact formulation**

## 7.2.2.10 Optimize Contact Function

The new version enhances the contact function, allowing users to define additional contact parameters:

- **Ramp interference**: Interference or clearance can be applied in stages to avoid non-convergence of the solution due to sudden load changes in the initial contact stage. For example, in interference fit analysis, linear growth of contact offset can replace traditional stepwise loading, improving convergence stability.

- **User defined penalty factor**: Users can manually define the normal contact stiffness scaling factor, providing a flexible balance between accuracy and efficiency.

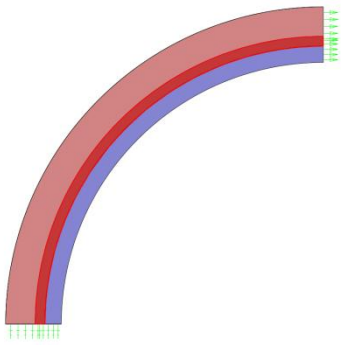
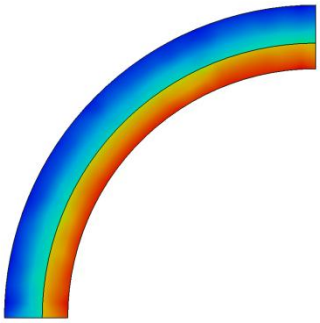
**【What users can do】**

✓ Avoid sudden changes in initial penetration or gaps through ramp interference and reduce the number of iterations.

✓ Users can adjust penalty parameters based on material properties (e.g., elastic modulus, Poisson's ratio) and mesh size to balance penetration depth and computational resources.

【Notes】

- It is recommended to start with a lower penalty parameter and gradually increase it until contact stress stabilizes.

| Initial interference between the inner and outer rings | Shrink fit is achieved through progressive interference |
| --- | --- |
|  |  |

【Where is it】

Simulation >> **Contact**

## 7.2.2.11 Optimize Harmonic Analysis Algorithm

The new version optimizes the harmonic response analysis algorithm, supporting the simultaneous definition of multiple independent frequency/phase harmonic excitation loads. This allows the simulation of structural dynamic responses under complex conditions, such as multi-source vibration and phase-difference excitation. Additionally, a residual vector method for force loading has been introduced, correcting modal truncation errors and significantly improving accuracy in the mid-to-high frequency range. This is particularly beneficial for scenarios with dense modes or insufficient modal count. In terms of efficiency, the new version of harmonic response analysis has significantly improved the calculation effect, and the calculation efficiency for large-scale problems has increased dozens of times.

【What users can do】

✓ Supports multi-source vibration scenarios, such as motors and wind power equipment, allowing simultaneous definition of base excitation (e.g., acceleration) and local force excitation.

✓ Uses the residual vector method to compensate for modal truncation errors, reducing the

required number of modes.

✓ Enables efficient harmonic response analysis for large-scale assemblies, meeting complex industrial-scale model requirements.

**【Notes】**

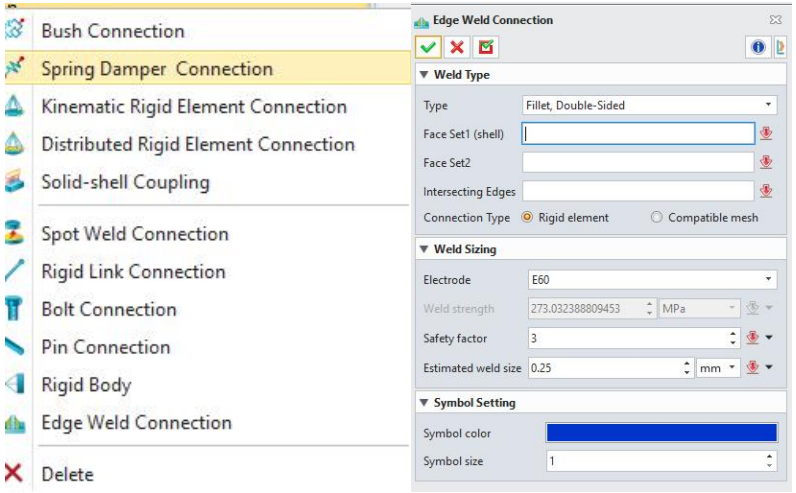
• It is recommended to first perform modal analysis to determine the structure's natural frequency range and avoid excessive frequency bandwidth, which could waste computational resources.

**【Where is it】**

Simulation >> **Harmonic Response Analysis**

## 7.2.2.12 Added the Edge Weld Connection Function

The new version has added five new welding types: fillet single and double-sided welding, fillet lap welding, and bevel single and double-sided welding. It supports users to determine the weld strength based on different electrode types, set corresponding safety factors, and preset weld dimensions. During post-processing, the required weld dimensions can be calculated based on the set weld strength, helping users verify the weld strength on the opposite side. And set reasonable weld seam dimensions according to actual needs.



Edge weld connection settings

**【What users can do】**

- ✓ Support the selection of different welding types.
- ✓ By setting the corresponding safety factors, presetting the weld seam dimensions, and using the weld seam strength and the actual force conditions to calculate the required weld seam dimensions in reverse.

**【Notes】**

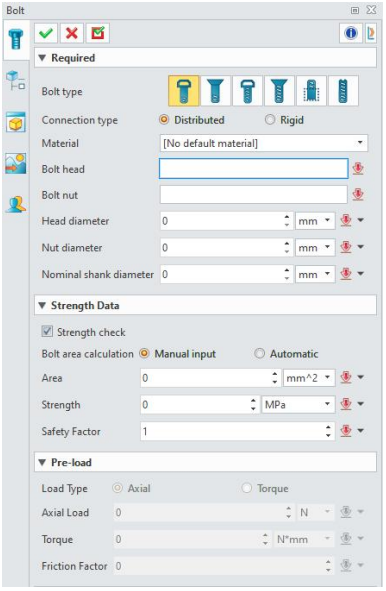
- Pay attention to the type of connection. When choosing a compatible grid, the type of face group 2 must be a shell element.

**【Where is it】**

Simulation >> Interaction >> Edge welding connection

## 7.2.2.13 Added the Function of Bolt Connection

The new version has added various types of bolt connections. Users can set corresponding parameters such as bolt material, threaded surface form, nut diameter, and handle diameter according to their needs. It supports a strength check function, allowing manual/automatic setting of the bolt area, strength, and safety factor, and adding prestress to the bolts.




bolt connection setting

**【What users can do】**

- ✓ Support the selection of different bolt types and set the corresponding geometric parameters according to different bolt types.
- ✓ Support the check of bolt strength by setting bolt strength and safety factors.

**【Where is it】**

Simulation >> Interaction >> Bolt connection

## 7.2.2.14 Added Multi-step(two steps) Analysis

ZWSim Structural adds multi-step(two-steps) analysis, designed for complex engineering scenarios, supporting multi-stage simulation needs. Users can independently set load conditions, contact states, boundary constraints, and solver parameters in different analysis steps, enabling full-process simulations from static to dynamic and thermal-structural coupling.

**【What users can do】**

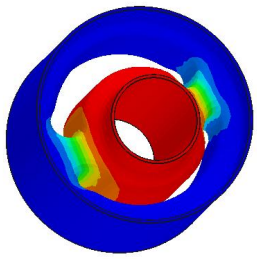
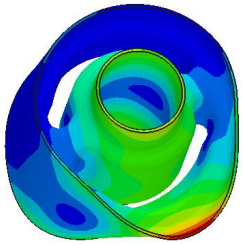
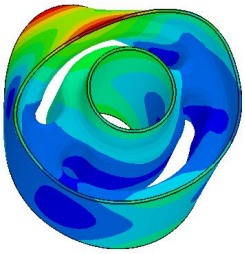
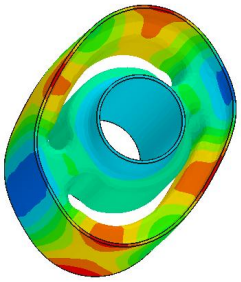
- ✓ Pre-stressed Modal Analysis: Combines static preload with dynamic vibration characteristic

evaluation.

- ✓ Thermal-Structural Coupling Analysis: Simulates material expansion deformation and stress distribution in high-temperature environments.
- ✓ Stepwise Assembly Simulation: Supports multi-step loading and constraint adjustments, making it suitable for verifying the mechanical behavior of complex assemblies.
- ✓ Nonlinear Load-Unload Cycles: Iterative simulations of nonlinear processes, such as material plastic deformation and contact separation.

**【Notes】**

- The steps must be planned according to physical logic (e.g., heat transfer should precede structural deformation).
- **Load Inheritance**: Dynamic analysis steps must explicitly define load inheritance methods from previous steps (e.g., fixed or time-varying loads).

| | | | |
|---|---|---|---|
|  |  |  |  |
| Prestressed load analysis | Prestressed modal analysis | | |

**【Where is it】**

Simulation >> New Structural Simulation >> Multi-step Analysis

## 7.2.2.15 Added Large Deformation Timoshenko Beam Function (ZW3D 2026 SP1)

The new version of the software features a comprehensive upgrade to the Timoshenko beam element, adding support for large-deformation geometric nonlinear analysis. Building upon the Timoshenko beam implemented in the previous version—which operated under the assumption of small deformations—this release introduces a new solver algorithm tailored for large-deformation scenarios. This enables engineers to accurately simulate beam structural behavior accounting for shear deformation across a broader range of engineering applications.

**【What can be done】**

- ✓ For beam structures with a small slenderness ratio (length-to-height ratio) (L/h < 10), shear deformation effects become significant, and traditional Euler-Bernoulli beam theory severely underestimates deflection. This feature enables more accurate prediction of deflection and stress distribution in short, thick beams under large deformation conditions.
- ✓ It can simulate the static behavior of beam structures under finite deformation, including large deflection and large rotation effects, accurately calculating the deformation and stress state of the structure under ultimate load.

**【Notes】**

- The material density property (foundation for inertia force calculation) must be defined. An appropriate beam theory should be selected based on the beam's slenderness ratio. For slender beams (L/h > 10), the Euler-Bernoulli beam theory, which is computationally more economical, may continue to be used. For short, thick beams (small L/h), composite beams, or problems requiring high-frequency vibration analysis, the Timoshenko beam element is recommended.

**【Where is it】**

Simulation >> Beam Element Property >> Timoshenko beam

## 7.2.2.16 Added Period Symmetry Constraint (ZW3D 2026 SP1)

The new version introduces periodic symmetry constraints for efficiently handling geometries with

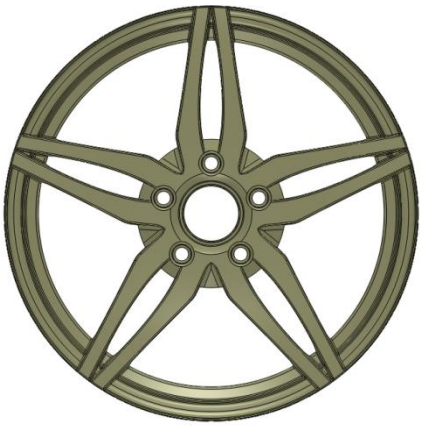
rotational symmetry. This feature allows users to accurately simulate the mechanical behavior of an entire structure by defining symmetric boundary conditions, requiring only a complete model of one symmetric sector. Periodic symmetry constraints are particularly suitable for static, modal, and frequency response analyses of rotating machinery (such as turbine blades, fans, gears, etc.) and periodic lattice structures. Its core principle leverages the structure's symmetry to reduce the full-model problem to a single sector problem, significantly lowering computational resource consumption while maintaining computational accuracy.

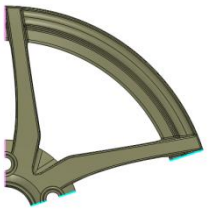
**【What can be done】**

- ✓ For structures with circumferentially uniform distributions such as fans, impellers, and propellers, users need only create a basic sector model for a single blade and its corresponding hub section (e.g., a 6-blade fan requires only a 60° sector). The software automatically simulates the overall structural dynamic response through periodic symmetry constraints, reducing computational effort to 1/n (where n is the number of blades). This approach is particularly suitable for analyzing large rotating components.
- ✓ After analysis, individual sector results can be automatically expanded to a complete 360° model, providing an intuitive visualization of the overall structure's vibration patterns.

**【Notes】**

- Periodic symmetry constraints are applicable only to problems where geometry, materials, and boundary conditions exhibit periodic symmetry. If the structure contains local asymmetric features (such as damaged blades or non-uniform supports), a full model must be employed.
- The angle between the two boundaries (starting face and ending face) of the base sector is 360° /n (where n is the integer number of sectors).





Periodic symmetry constraints

**【Where is it】**

Simulation >> Interaction >> Constraint Equation

## 7.2.2.17 Added Harmonic Response Analysis Based on the Direct Method (ZW3D 2026 SP1)

The new version of the Harmonic Response Analysis module introduces direct solution technology, enabling direct solution of second-order dynamic differential equations in the frequency domain without relying on modal shape extraction and superposition. Through full matrix operations, it calculates responses such as displacement and stress at each frequency point, making it particularly suitable for handling complex problems involving asymmetric system matrices, such as those arising from non-proportional damping and acoustic-solid coupling.

**【What can be done】**

✓ The direct method eliminates the need for preliminary modal analysis, sparing users from concerns about selecting principal degrees of freedom or determining modal order, thereby simplifying the analysis setup process. Simultaneously, by directly solving the full model equations, it avoids errors potentially introduced by modal truncation (i.e., using only partial low-order modes) in the modal superposition method, particularly aiding in capturing the influence of high-order modes.

【Notes】

- The direct method, due to its direct processing of the entire system matrix, typically requires more computer memory than the modal superposition method, especially when solving large-scale models. Users are advised to ensure sufficient physical memory configuration based on model scale (number of nodes, degrees of freedom) to prevent solution termination caused by insufficient memory.

【Where is it】

Simulation >> Harmonic Response Analysis >> Task Option >> Direct Method

### 7.2.2.18 Added General Contact Functionality (ZW3D 2026 SP1)

The new version introduces a universal contact feature that enables users to automatically detect and manage contact state changes throughout the analysis process by selecting all potentially contacting surfaces at once, eliminating the need to manually define specific contact pairs.

This feature is particularly suited for complex assemblies or scenarios with numerous contact surfaces and unpredictable contact relationships. Compared to traditional contact pair methods, universal contact significantly streamlines the preprocessing workflow and enhances modeling efficiency for complex contact problems.

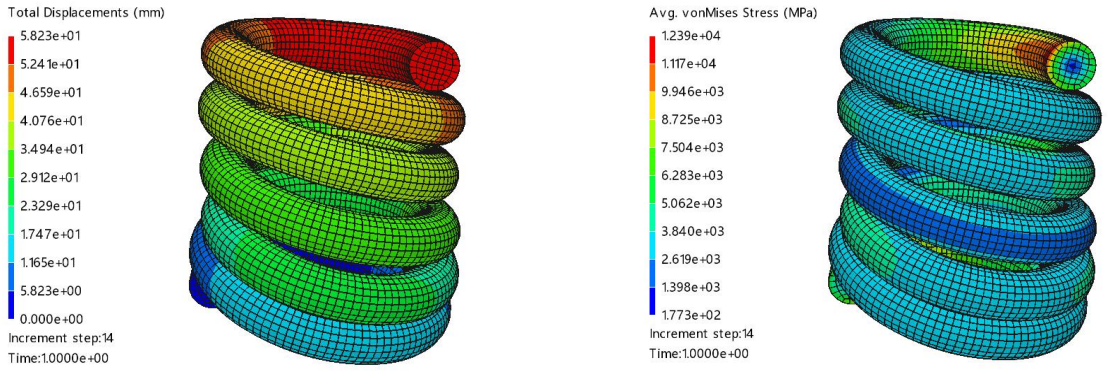
【What can be done】

✓ For complex assemblies with numerous potential contact surfaces—such as precision transmission mechanisms, electronic connectors, and vehicle suspension systems—the universal contact feature automatically identifies and calculates all possible contact interactions. Users need not define contact pairs individually; simply specify components or surfaces that may engage in contact, and the software automatically handles contact relationships, significantly reducing preprocessing time and setup complexity.

✓ In analyses involving large deformations (such as seal compression, rubber part bending, wire crimping, etc.), General Contact effectively handles self-contact between objects and their own surfaces, as well as dynamic changes in contact relationships.

【Notes】

- While universal contact offers convenience, it cannot entirely replace traditional contact pair methods. For simple models with relatively well-defined and stable contact relationships, or scenarios requiring precise control over specific contact interfaces, contact pair methods may prove more advantageous.




Spring Self-Contact Analysis Based on General Contact

【Where is it】

Simulation >> Interaction >> General Contact

## 7.2.2.19 Support for Drop Analysis of Hyperelastic Material Structures (ZW3D 2026 SP1)

The new version resolves the issue of inaccurate simulation of hyperelastic materials (such as rubber and silicone) in drop analysis. By introducing hyperelastic constitutive models, it enables more realistic prediction of stress-strain responses and rebound behavior during drops for components like seals, shock absorbers, and flexible protective sleeves, helping engineers optimize product impact resistance.

【Where is it】

Simulation >> Material >> Hyperelastic Material

### 7.2.2.20 Added New Temperature-Related Materials (ZW3D 2026 SP1)

The new version supports temperature-dependent materials, enabling users to define temperature-varying curves for key material parameters such as elastic modulus, Poisson's ratio, thermal expansion coefficient, thermal conductivity, and specific heat capacity. This allows material properties to exhibit different values at different temperatures, significantly enhancing the computational accuracy and reliability of structural behavior simulations in high-temperature or low-temperature environments.

In practical engineering applications, the mechanical and thermal properties of most materials undergo significant changes with temperature variations. This capability enables engineers to simulate the actual performance of products under variable temperature conditions with greater realism.

**【What can be done】**

- ✓ When performing structural analysis involving temperature fields (such as brake disc thermal stress analysis or electronic chip thermal expansion analysis), this feature precisely accounts for material property variations with temperature. This yields stress distributions and deformation patterns that more accurately reflect real-world conditions, eliminating calculation errors caused by using room-temperature material parameters.
- ✓ In analyses involving large deformations (such as seal compression, rubber part bending, wire crimping, etc.), General Contact effectively handles self-contact between objects and their own surfaces, as well as dynamic changes in contact relationships.

**【Where is it】**

Simulation >> Material Library >> Linear elastic material

### 7.2.2.21 Added Auto Stabilization Feature (ZW3D 2026 SP1)

An automatic stabilization feature has been added to effectively address convergence issues in nonlinear analysis. When numerical instability occurs during simulation—such as iteration divergence caused by local underconstraint, contact discontinuity, or material softening—the stabilization function

automatically applies a small artificial damping force. This suppresses rigid-body displacement or high-frequency oscillations, enabling the solver to smoothly navigate critical states and ultimately obtain a balanced solution.

The core principle involves introducing a "pseudo-damping" term related to unstable degrees of freedom into the system stiffness matrix. This damping value automatically decays as the solution process stabilizes, thereby preventing significant interference with the final physical results.

**【What can be done】**

✓ In the analysis of assemblies involving complex contacts—such as multi-body contacts, large slips, and separation-recontact — the automatic stabilization function effectively suppresses numerical oscillations caused by abrupt changes in contact states. This prevents analysis interruptions due to iterative oscillations, thereby enhancing the success rate of simulations for complex assemblies;

✓ For models potentially containing minor unconstrained degrees of freedom (e.g., slight incompleteness in constraints imposed based on symmetry), the automatic stabilization feature automatically suppresses rigid-body motion in these degrees of freedom, enabling convergence without requiring users to precisely adjust boundary conditions.

**【Notes】**

- The damping factor is the key parameter controlling the magnitude of artificial damping. The initial value should adopt the recommended value automatically calculated by the software. An excessively low factor may fail to effectively suppress oscillations; conversely, an excessively high factor will cause results to deviate from physical reality, manifesting as an "overly stiff" response or abnormal energy dissipation;

- After enabling the automatic stabilization function, the pseudo-strain energy in the output file must be checked. Typically, this energy should not exceed 1%-5% of the total internal energy. If the proportion is too high, it indicates that the automatic stabilization function has significantly affected the results. In such cases, the physical validity of the results should be carefully evaluated, or the stabilization factor should be adjusted and the calculations rerun;

- Automatic stabilization should be regarded as a numerical tool to aid convergence, not as a substitute for physical reality. If the model itself exhibits significant rigid-body displacement or insufficient constraints, the model should first be inspected and corrected.

Three-point bending simulation based on automatic stabilization function

**【Where is it】**

Simulation >> Nonlinear Static >> Task Option >> Automatic Stabilization

## 7.2.2.22 Added Reduced Integration Element algorithm (ZW3D 2026 SP1)

The reduced integration element function has been added, and both the stiffness method and the enhanced strain method are supported, which can effectively balance the calculation accuracy, efficiency and stability.

**【What can be done】**

- ✓ Reduced integration elements are typically more computationally efficient than fully integrated elements of the same order, as they feature fewer integration points and require less computational effort for forming the element stiffness matrix and performing integration. This significantly reduces solution time and enhances analysis efficiency when handling large-scale models；
- ✓ For structures primarily subjected to bending, reduced-integral elements can effectively avoid the shear self-locking issues commonly encountered with fully-integral linear elements.

**【Notes】**

✓ The use of reduced integral elements carries the risk of hourglass patterns, manifesting as irregular, jagged deformations in the mesh. Hourglass energy monitoring is essential. Typically, the ratio of pseudo-strain energy to total internal energy should be kept below 5−10%. If this ratio is excessively high, it indicates significant hourglass pattern influence, potentially rendering the results unreliable.

**【Where is it】**

Simulation >> Simulation Tree >> Right-click entity >> Entity Properties

## 7.2.3 Post-processing

### 7.2.3.1 New Post-processing Module

The post-processing functionality has been separated from the pre-processing interface to form a dedicated results analysis interface, avoiding efficiency issues caused by mixed functionalities. In the previous version, the simulation workflow and results analysis shared the same toolbar, while the current version distinguishes between two independent modules: 'Simulation Workflow' and 'Post-Processing Results'. This provides clearer grouping and categorization of result tools, such as view management, results analysis, and report generation, making user operations more intuitive and efficient.

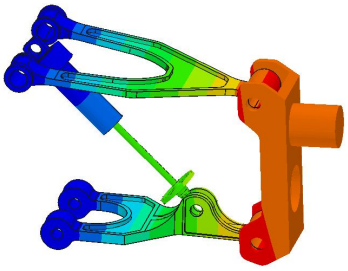
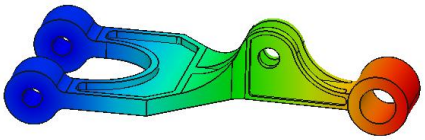
The ribbons of the new post-processing module

**【Where is it】**

Simulation >> Analyze >> **Enter Post-processing**

### 7.2.3.2 Added Partial Display Function for Contour

The new version supports the partial display function for contours, allowing users to generate and display contours only for local mesh regions during the post-processing stage.

| Full Display of Contour | Partial Display of Contour |
| --- | --- |
|  |  |

**【Where is it】**

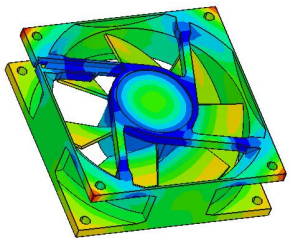
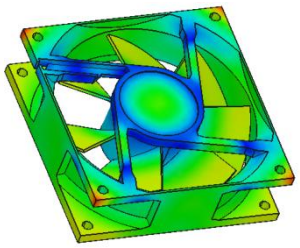
Result >> General >> Partial Show/Hide

### 7.2.3.3 Added Discrete Contour Display Function

The new version supports the discrete display format of contours, allowing continuous contours to be converted into discrete color scales, which intuitively present areas of value jumps in simulation results through different color blocks. With discrete contours, it can clearly display non-uniform distributions and significant variations in discrete data within the simulation results.

**【What users can do】**

✓ By using discrete color scales, gradient abrupt regions (such as stress concentration or abrupt temperature changes) are clearly visualized, facilitating the rapid identification of critical anomalies.

| Discrete Contour | Smooth Contour |
| --- | --- |
|  |  |

**【Where is it】**

Result >> Toolbar >> **Discrete Display**

### 7.2.3.4 Added Analysis Result/Frame Function

The new result/frame switching feature allows users to quickly switch results at the post-processing stage based on incremental steps (such as iterative steps in nonlinear analysis) or time frames (such as time nodes in transient dynamics, vibration response analysis).

**【What users can do】**

✓ Rapidly switch to key frames and conduct multi-dimensional comparative analysis with tools like contours.

Rapid Results Switching Tool

**【Where is it】**

Result >> Contour Control >> **Result/Frame**

### 7.2.3.5 Added Post-processing Calculator Function

The new version introduces a post-processing calculator function, offering users a flexible tool for customized result analysis. It supports operations including addition, subtraction, multiplication, division, exponentiation, logarithms, and trigonometric functions, allowing combined calculations on existing field variables such as stress, strain, and displacement.

**【What users can do】**

✓ Users can define new custom result variables by applying flexible mathematical expressions to existing field variables.

✓ The calculation results can be directly applied to create graphical outputs such as contour plots and curve plots, enabling more intuitive data presentation.

**【Notes】**

- Make sure the entered mathematical formulas and expressions are accurate to prevent

calculation errors due to syntax issues.

- Prior to performing calculations, verify that the units and dimensions of all field variables are consistent to ensure the correctness of the results.

**【Where is it】**

Result >> Result Data >> **XY Plot Calculate**

Result >> Result Data >> **Contour Calculate**

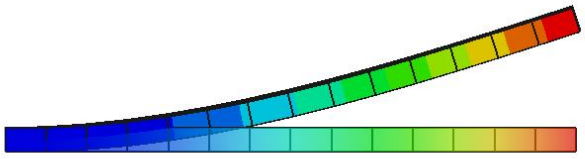
### 7.2.3.6 Added Deformed/Undeformed Contour Display Function

The new version enables the display of deformed contours and the simultaneous display of deformed and undeformed contours, offering a clear visualization of geometric changes before and after deformation and the distribution of simulation results.

**【What users can do】**

✓ The magnitude, direction, and dynamic changes in deformation distribution can be intuitively observed.

✓ The deformation trend can be verified to ensure it conforms to physical expectations.



Simultaneously Display Deformed and Undeformed Contours

**【Where is it】**

Result >> Toolbar >> **Deformed Contour / Undeformed Contour / Deformed and Undeformed Contours**

### 7.2.3.7 Optimized Result Report Export Function

The new version improves the result report exporting function, emphasizing enhanced

professionalism, information completeness, and automation. It enables the automatic inclusion of key simulation configuration details, including analysis steps, load conditions, boundary conditions, and calculation parameters, to produce structured reports.

**【What users can do】**

- ✓ The function automatically extracts simulation parameters, including material properties, mesh density, and solver settings.
- ✓ Users can choose to export all or specific results on demand, minimizing redundant data and improving the relevance of the report.

**【Where is it】**

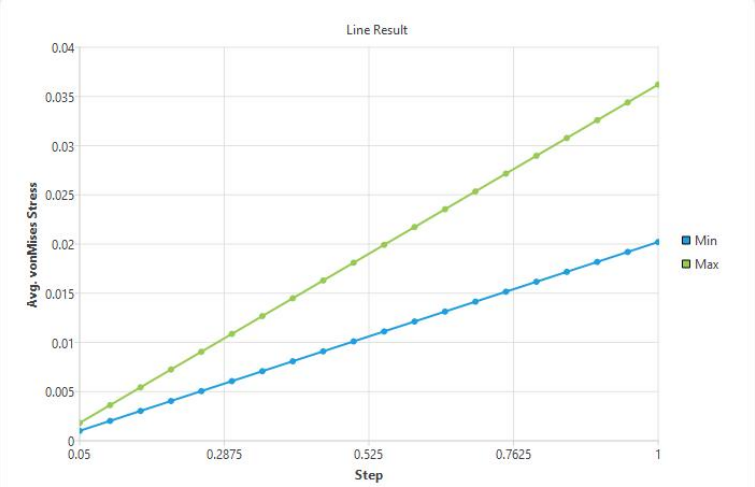
Result >> Result Data >> **Report**

### 7.2.3.8 Added XY Plot Function Based on Stepwise Maximum and Minimum Values

The new version enhances the XY plot function, enabling the generation of curve graphs based on stepwise maximum and minimum values. It automatically extracts extreme value data (such as maximum stress, minimum displacement, and temperature peaks) from each simulation step and dynamically visualizes parameter trends through 2D curves, helping users quickly identify critical points.

**【What users can do】**

- ✓ By comparing the differences in extreme value curves before and after optimization, users can quantify the practical impact of enhancement measures such as lightweight design and material substitution.
- ✓ Users can rapidly pinpoint critical moments of structural failure risk by analyzing extreme value curves.




The step-max/min plot of a nonlinear analysis result

**【Where is it】**

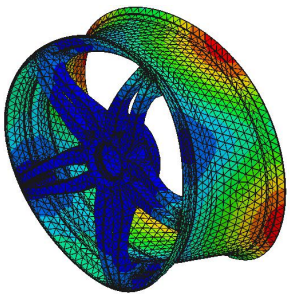
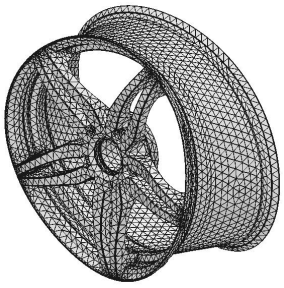
Result >> Result Data >> New XY Plot >> **Max/Min**

## 7.2.3.9 Added Export Deformed Mesh Function

The new version adds the export deformed mesh function, enabling users to save the post-simulation deformed state as a mesh data file. This exported mesh data can be reused to redefine analysis conditions and perform secondary simulation validation in the software.

**【What users can do】**

✓ The deformed mesh can be imported into other physics field modules to enable secondary analysis in applications such as thermal-mechanical coupling.

| Deformation Result | Export Deformed Mesh |
| --- | --- |
|  |  |

**【Where is it】**

Result >> Result Data >> **Export Deformed Mesh**

## 7.2.3.10 Added Export Image Function

The new version improves the image export feature, enabling users to directly generate and export animated GIF files within the post-processing interface without the need for third-party software. It provides customizable resolution settings to meet the quality requirements of different scenarios.

**【What users can do】**

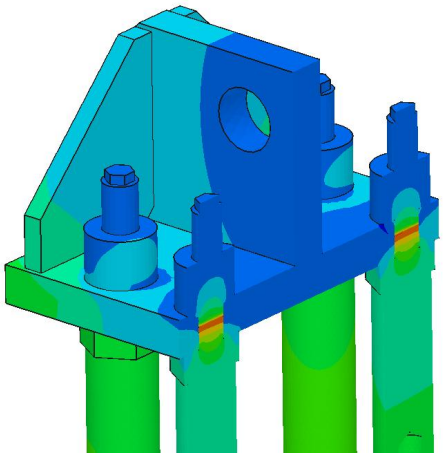
- ✓ Intuitively displays the dynamic response process of structures (such as vibration modes and collision transients.
- ✓ Exports GIF files of the dynamic response process of structures (such as vibration modes and collision transients), allowing users to directly embed the files into reports like Word or PPT.

**【Where is it】**

Result >> Result Data >> **Export Image**

## 7.2.3.11 Added Mesh Plane Sectioning Feature (ZW3D 2026 SP1)

The new version introduces a mesh plane sectioning feature. This functionality enables users to define sectioning planes and visualize smooth cross-sectional results within the mesh model, significantly enhancing the intuitiveness and accuracy of simulation result analysis. Unlike previous versions where section results were displayed as discrete element boundaries, the new feature employs advanced data interpolation and smoothing algorithms to generate continuous, smooth contour plots of physical field distributions—such as stress, strain, or temperature—on any specified section plane.



Smooth plane cutting results

【Where is it】

Result >> General Settings >> Sectioning

## 7.2.3.12 Added Dynamic Stiffness Calculator Feature (ZW3D 2026 SP1)

Dynamic stiffness is a key physical quantity characterizing a structure's resistance to dynamic deformation. Unlike static stiffness, it is a frequency-dependent complex number where the magnitude represents the structure's ability to resist deformation at specific frequencies, while the phase reflects the delay relationship between excitation and response. The new version introduces a dynamic stiffness calculator feature, enabling users to conveniently compute the structure's dynamic stiffness curve and key metric values after completing harmonic response analysis.

**【What can be done】**

✓ The post-processing dynamic stiffness calculator supports the average method, exponential method, and area method. Users can select the appropriate method based on specific analysis objectives and industry standards；

✓ This feature integrates dynamic stiffness calculation into the software's post-processing environment, eliminating the need for users to manually export frequency response function data to external programs for complex formula conversions and calculations. After completing harmonic response analysis, users simply specify the response curve in the calculator to obtain dynamic

stiffness results with a single click.

**【Where is it】**

Harmonic Response Analysis >> Results >> IPI Calculator

## 7.3 Low Frequency Electromagnetic Simulation

The ZW 3D low-frequency electromagnetic simulation plug-in adopts a high-precision finite element (FEM) solution algorithm and supports two-dimensional and three-dimensional static magnetic field, transient magnetic field, electrostatic field, and DC conduction electric field solution types. It can help users complete electromagnetic field distribution, performance parameter simulation and optimization analysis of various types of motors, transformers, actuators, sensors, wireless charging and other electromechanical equipment and power equipment.

Applicable scenarios include various motors (permanent magnet motors, induction motors, etc.), transformers, reactors, lightning arresters, insulating bushings, actuators, electromagnetic sensors, wireless charging and other equipment.

### 7.3.1 Solver Capability Improvement

#### 7.3.1.1 Improved Efficiency in Solving Large-scale Simulation Tasks

Solution efficiency plays a key role in determining the solution and calculation capabilities of simulation software. ZW3D V2026 optimizes the calculation method and improves the solution stability and calculation efficiency of some cases. Better solution efficiency allows the software to be more powerful and efficient, helping users solve larger and more complex practical problems.

**【What users can do】**

- ✓ Improved solution efficiency allows the software to calculate larger and more complex simulation scenarios;
- ✓ Under the same problem scale, using the optimized version can significantly save computing time and improve the work efficiency of design simulation.

**【Notes】**

- During use, you need to pay attention to the scale of the problem being solved and the degree of matching between local computing resources;
- While computational efficiency is improved, memory usage remains unchanged.

**【Example】**

Through comparative calculations of multiple cases, the calculation efficiency has been significantly improved. The detailed comparative data is shown in the following table:

| Number | Unknowns | Improved efficiency |
|---|---|---|
| ① | 8963840 | 67% |
| ② | 29999380 | 58% |
| ③ | 98063705 | Improve crash |

**【Where is it】**

Part/Assembly Environment >> Simulation >> ZwMetas 3D >> **Electrostatic**

### 7.3.1.2 Dynamic Grid Computing Function

It supports dynamic mesh solution calculation, can adaptively update the mesh during the simulation process, and can solve motion problems such as motors.

**【What users can do】**

✓ Solve time domain dynamic motion problems;

✓ According to the different positions of the object's motion, the updated mesh is adaptively generated.

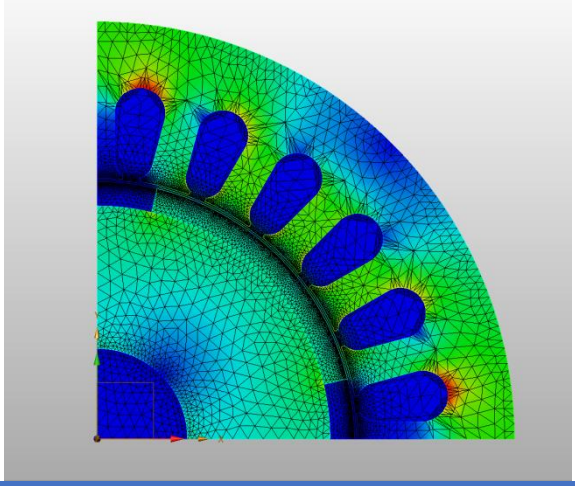
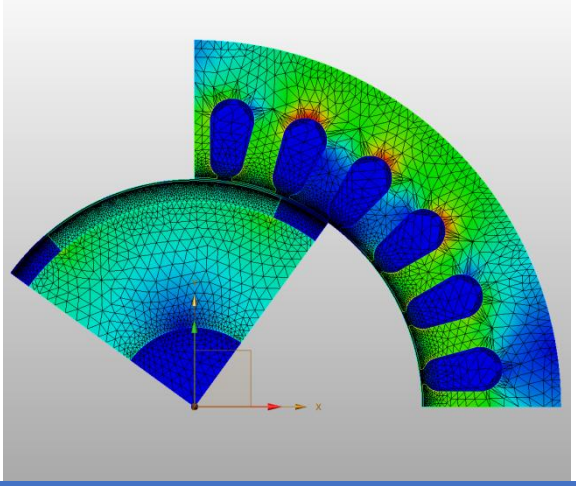
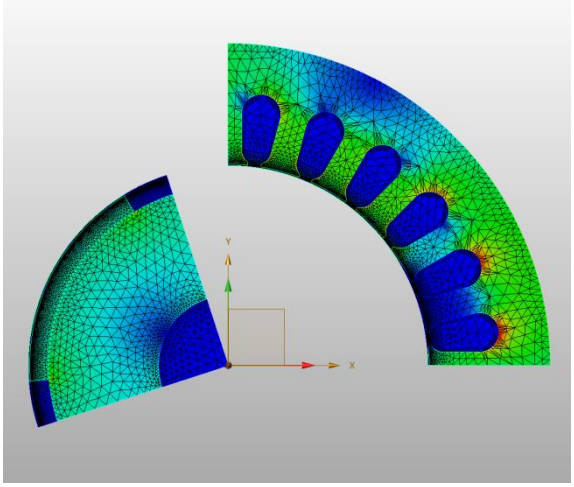
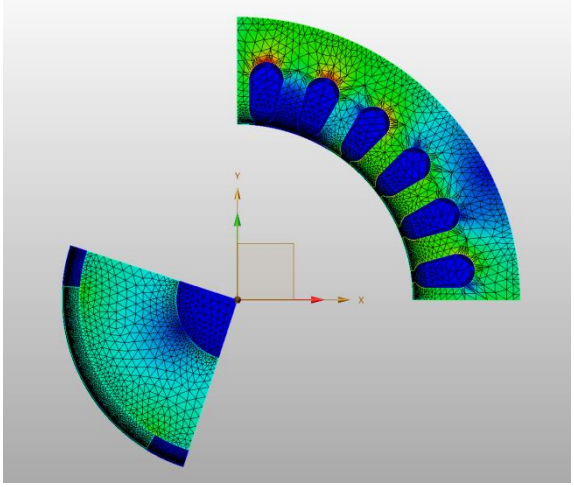
**【Notes】**

- All moving components must be within the slip boundaries set in the moving area;
- Pay attention to the reasonable setting of movement direction;
- Pay attention to the coordination between the time step and the movement speed.

**【Examples】**

During the solution process of the motor's rotational motion simulation, the mesh for the next

solution can be automatically generated based on the current mesh, and the entire solution's dynamic mesh generation and calculation can be completed in this way.

| Moment 1 | Moment 2 |
| --- | --- |
|  |  |
| **Moment 3** | **Moment 4** |
|  |  |

**【Where is it】**

Part/Assembly Environment >> Simulation >> **ZwMetas 2D >> Transient (Cartesian,XY)**

Part/Assembly Environment >> Simulation >> **ZwMetas 2D >> Transient (Cylindrical about Z)**

## 7.3.2 Post-processing Optimization

### 7.3.2.1 Improved Post-processing Display Capabilities

As the scale of solution calculation increases, the post-processing cloud display and rendering steps will also take a lot of time. ZW3D V2026 version has optimized the post-processing display model, reducing the scale of data storage and greatly improving display efficiency under the same display quality.

**【What users can do】**

- ✓ Display for large-scale unknown quantity calculation problems;
- ✓ Improve display speed under the same computing scale.

**【Notes】**

- Double-click to open the cloud map display normally;
- Pay attention to the matching degree between the post-processing display scale and the local computer hardware resources.

**【Examples】**

Through a simulation case of an electrostatic field with 8 million unknowns, the post-processing cloud map generation time is compared. The results show that the ZW3D V2026 version has significantly improved for different post-processing results. The detailed comparison data is shown in the following table:

| Number | Result Type | Improved Efficiency |
|---|---|---|
| ① | Contour results | 70% |
| ② | Scalar cloud results | 65% |
| ③ | Vector cloud results | 67% |
| ④ | Screenshot cloud image results | 52% |

**【Where is it】**

Part/Assembly Environment >> Simulation >> Results >> **Field Results**

### 7.3.2.2 New Section Cloud Function

In a specific simulation scenario, it is necessary to observe the distribution and changes of the physical fields inside objects and on the interfaces between objects.

**【What users can do】**

✓ Physical field inspection inside objects;

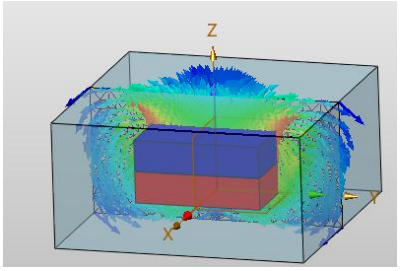
✓ The contour display on any defined section can be supported directly in the post-processing results.

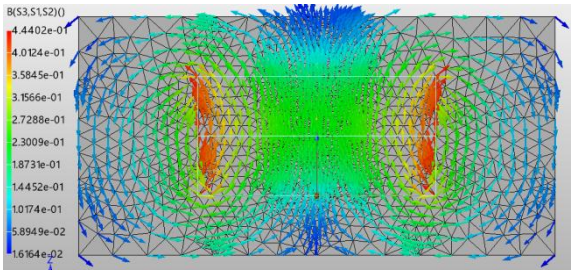
**【Notes】**

- The section of the cross-section cloud diagram is determined by adding the section normal vector to any point on the section;
- When viewing the cross-sectional cloud map, in order to obtain better results, it is usually necessary to select the simulated air domain as the viewing target;
- If the cross-sectional cloud map is blocked by objects and the visual effect is poor, you can use the wireframe display mode or the section view to adjust the view to achieve a better display effect.

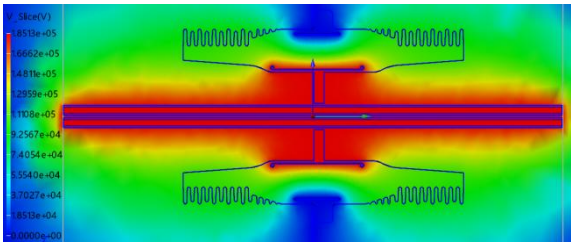
**【Examples】**

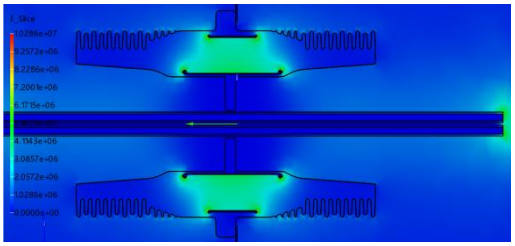
1) During the simulation of a permanent magnet, you can use the cross-sectional cloud graph function to view the magnetic field distribution state in the entire space on a certain cross section.




2) During the simulation of power switchgear, it is necessary to check the field distribution in the internal gas insulation space.





**【Where is it】**

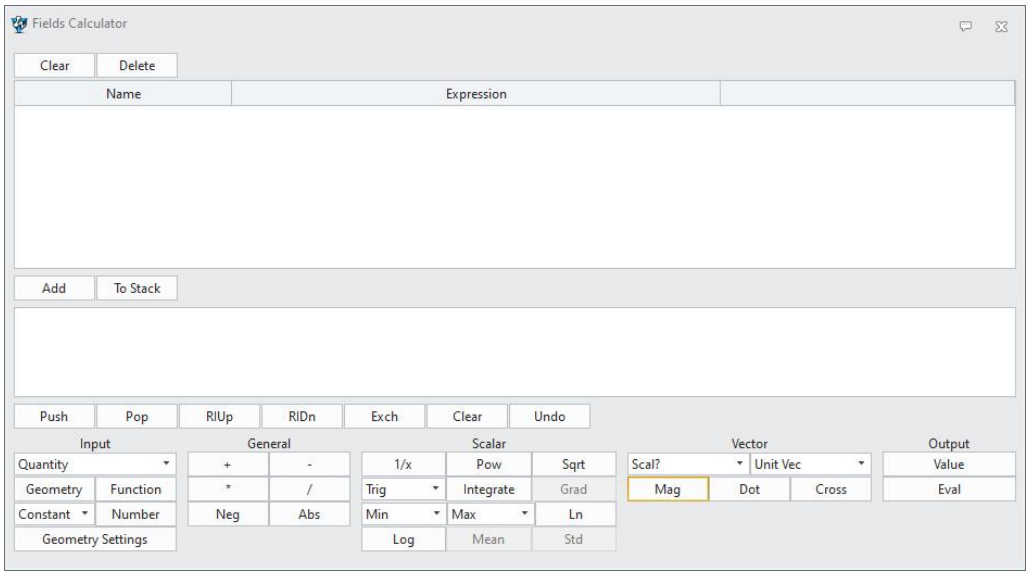
Part/Assembly Environment >> Simulation >> Results >> **Create Slice disp**

### 7.3.2.3 Added Field Calculator Function

It supports users to define the physical quantities displayed in post-processing, and perform arbitrary scalar and vector operations on basic post-processing physical quantities.

**【What users can do】**

- ✓ Define the physical quantities displayed in post-processing
- ✓ Various post-processing physical quantity calculations



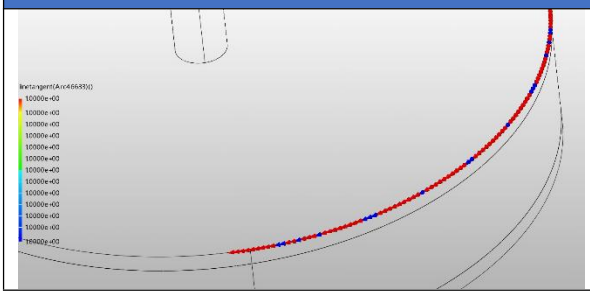
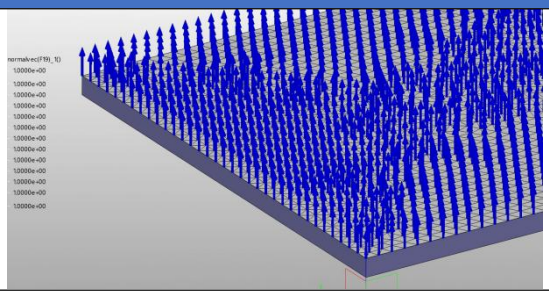

**【Notes】**

- When using field calculations, pay attention to the matching of operation variables and operators. For example, scalar operations such as addition, subtraction, multiplication and division cannot be used on vectors, and modulo, dot product and cross product cannot be used on scalars.

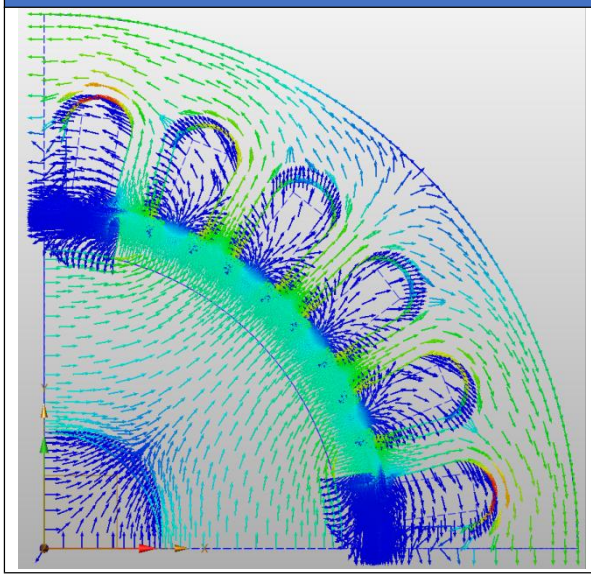
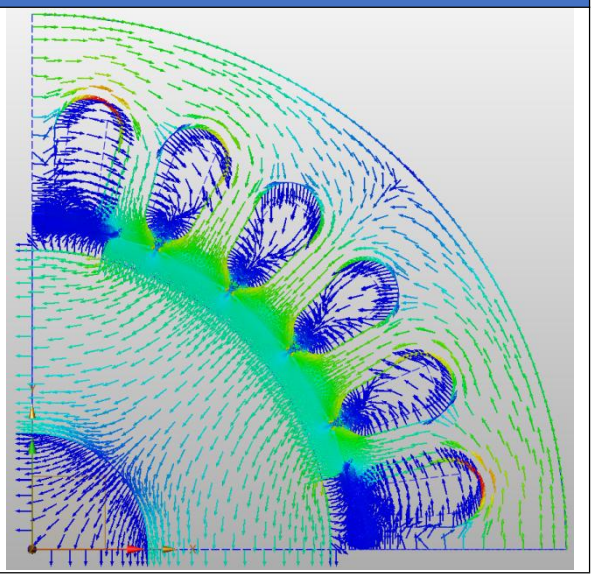
- After defining the field calculation components, when drawing the field component graph, you also need to pay attention to the matching between the defined field components and the geometric body. For example, if the custom component contains the calculation of the surface volume, then the component can only be drawn on the surface later. If the custom component contains the calculation of the curve tangent vector, then the component can only be drawn on the surface.

**【Examples】**

1) For example, the tangent unit vector and the normal unit vector of an arbitrary curve can be obtained by defining a field calculator.

| Tangential component of curve | Surface normal component |
| --- | --- |
|  |  |

2) For example, when the vector distribution of magnetic induction intensity VectorB is known, how can we know the distribution of the negative number of magnetic induction intensity vector -VectorB ? This can be achieved by defining the calculator to customize the method.

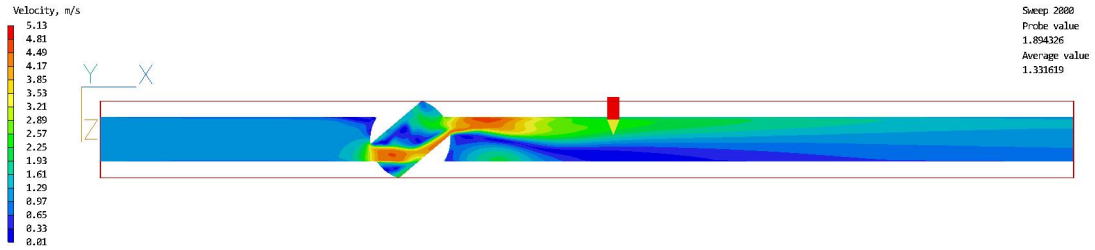
| Magnetic induction intensity vector | Negative magnetic induction intensity vector |
| --- | --- |
|  |  |

**【Where is it】**

Part/Assembly Environment >> Simulation >> Results >> **Calculator**

## 7.4 ★Fluid Simulation

A new fluid simulation module has been introduced, which provides functionalities for analyzing flow resistance and supports the evaluation of fluid flow characteristics in pipelines, channels, valves, and porous media. It can simulate energy losses caused by factors such as friction and turbulence, and predict key parameters such as pressure distribution, velocity field changes, and temperature variations. It supports the analysis of pressure loss in pipeline systems, evaluation of flow characteristics in complex channels, calculation of heat exchange efficiency, and fluid flow simulation in porous media. This can be applied to industrial pipeline optimization design, improving fluid transport efficiency, performance evaluation of valves and fluid control devices, as well as fluid optimization in electronic device heat dissipation and cooling systems.




Flow distribution inside the pipeline valve

Pressure distribution within the porous medium

### 7.4.1 New pre-processing functionality added

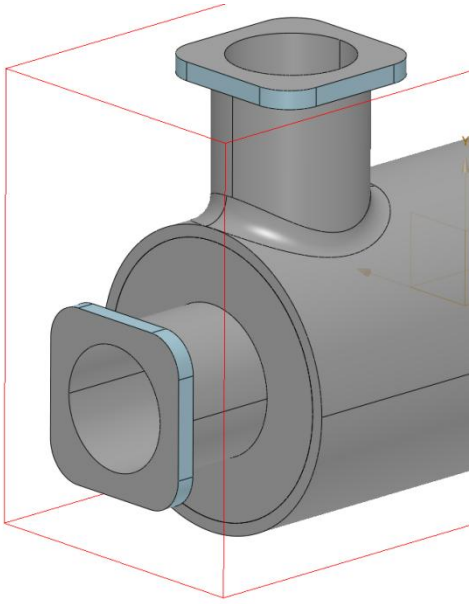
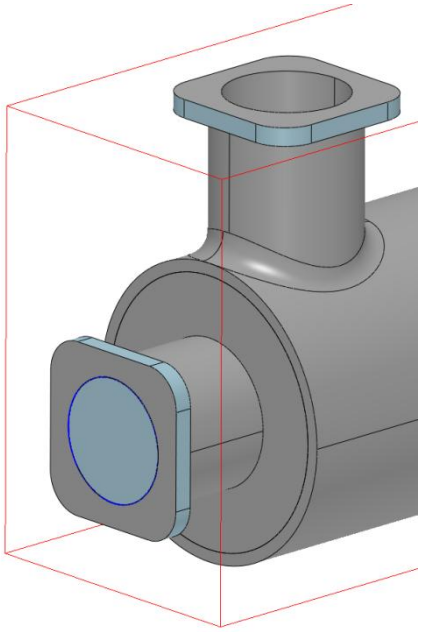
#### 7.4.1.1 New automatic cap creation feature added

Supports creating end caps for all holes within the boundary of the selected plane, and allows

customization of the end cap thickness.

**【What users can do】**

✓ Supports one-click creation of end caps for analyzing fluid flow within completely enclosed volumes.

| Before creating the cap | After creating the cap |
| --- | --- |
|  |  |

**【Notes】**

- Ensure the quality of the geometry, with no broken surfaces or other dirty geometries.

**【Where is it】**

Simulation >> New Fluid >> Geometry-Process >> **Cover Holes**

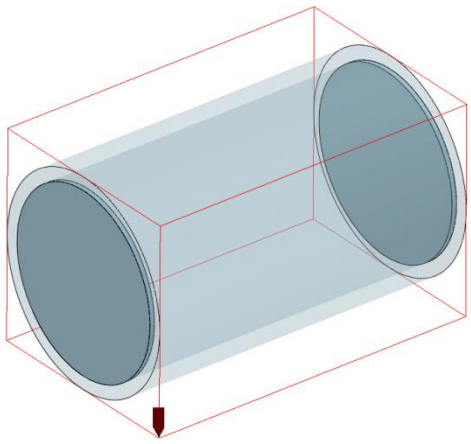
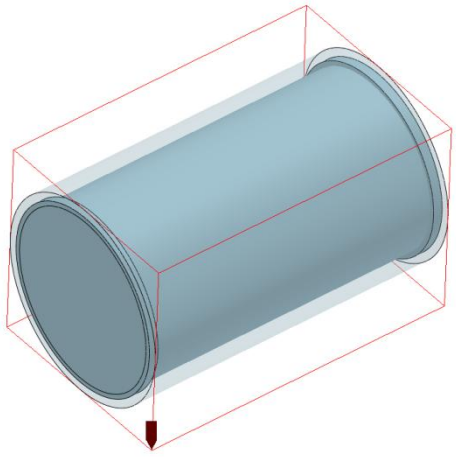
### 7.4.1.2 New fluid domain extraction feature added

Select a closed fluid region formed by the base geometry and end caps, with adjustable end cap thickness and side offset distance.

**【What users can do】**

✓ Supports defining fluid subdomains, which differ from the default fluid material of the

computational domain.

| Before extracting the fluid region | After extracting the fluid region |
| --- | --- |
|  |  |

**【Notes】**

- Fluid regions of different fluid types must be separated by solid regions.
- The geometry must be closed and have high geometric quality requirements.

**【Where is it】**

Simulation >> New Fluid >> Geometry-Process >> **Extract Inner Tubes**

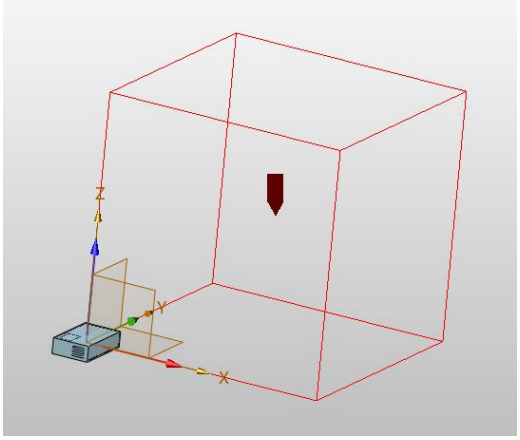
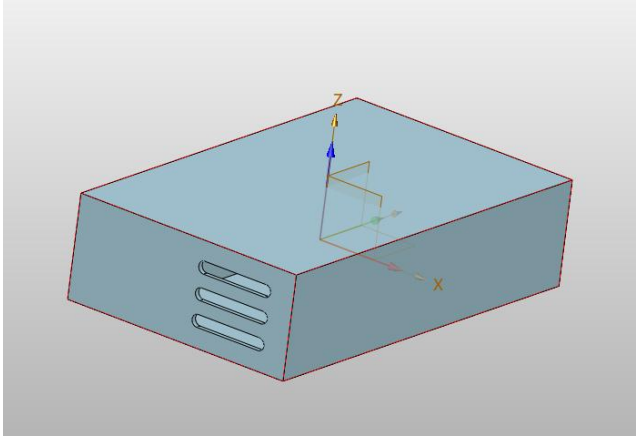
### 7.4.1.3 New adjustment of the domain

To avoid insufficient computational domain size, the domain should be reset. You can choose to automatically reset the domain, fitting the geometry size, or manually adjust the domain size.

**【What users can do】**

✓ Supports automatic or manual adjustment of the domain size.

✓ The size of the domain can be adjusted based on the scaling ratio and distance, or by directly setting the origin position and dimensions of the domain.

| Before adjusting the domain size | After adjusting the domain size |
| --- | --- |
|  |  |

**【Notes】**

- Fluid regions of different fluid types must be separated by solid regions.
- The geometry must be closed and have high geometric quality requirements.

**【Where is it】**

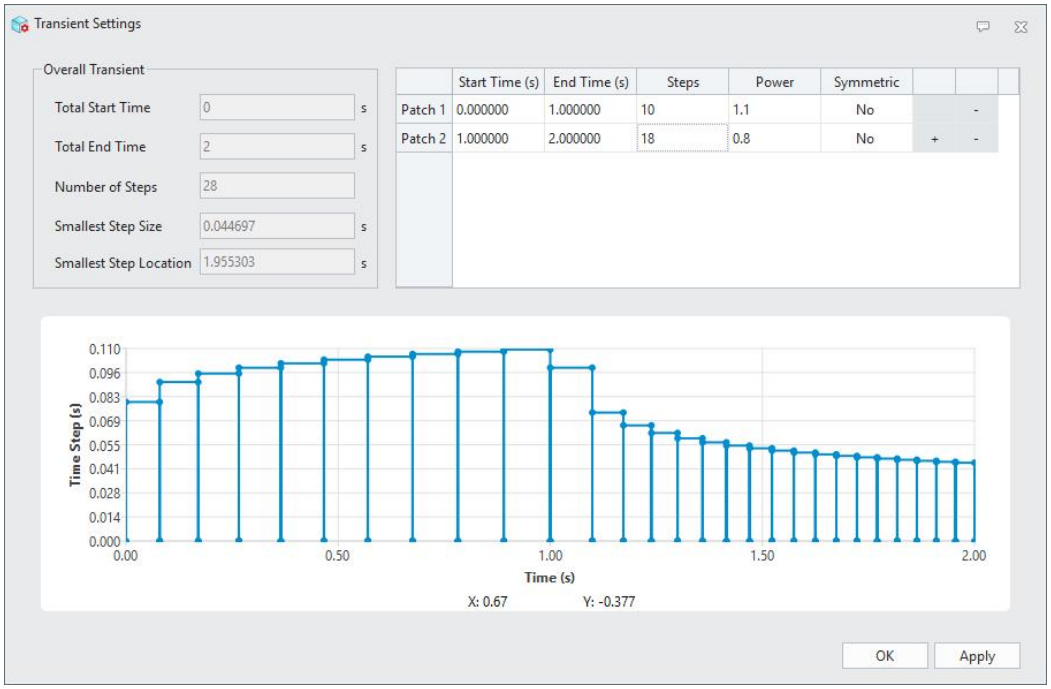
Simulation >> New Fluid >> Configuration >> **Domain**

### 7.4.1.4 Add Transient Analysis

For transient problems, the final solution will be time-stepped based on the initial conditions for the specified problem.

**【What users can do】**

- ✓ Supports non-uniform changes in total time and time step size within individual phases.
- ✓ Visualization of transient time steps.




**【Notes】**

- The minimum step size and minimum step position are displayed only and cannot be adjusted.
- During transient analysis, the intermediate step result storage feature must be enabled, with customizable save frequency.

**【Where is it】**

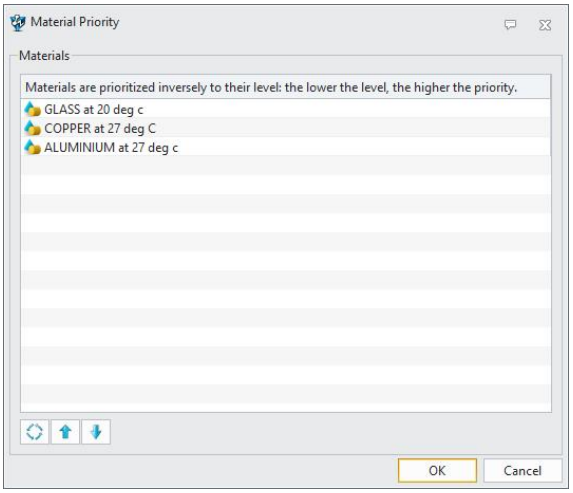
Simulation >> New Fluid >> Configuration >>Model Set >>Transient Solution >> **Click To Edit**

## 7.4.1.5 Add Material Priority Settings

For overlapping areas, if there are definitions of different materials, the material type for the overlapping area needs to be determined through material priority settings.

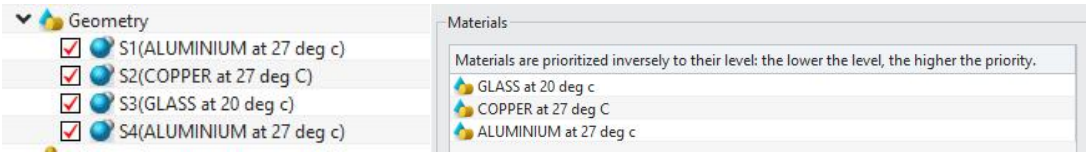
**【What users can do】**

- ✓ All materials used in the model are included in the list.
- ✓ Use the up and down arrows to adjust the material priority order; the lower the position, the higher the priority.
- ✓ Materials sorting can be reset with one click.




**【Notes】**

- If multiple models use the same material, only one material will appear in the material list.




**【Where is it】**

Simulation >> New Fluid >> **Material Priority**

### 7.4.1.6 Add Fan Model

A fan is a type of flow boundary condition, which can be set as a circular or rectangular plane. For circular fans, the rotation and direction of rotation can be considered.

**【What users can do】**

- ✓ Simulate circular and square fans.
- ✓ The fan's PQ curve can be set.
- ✓ The rotation direction of the fan can be set.

**【Notes】**

- Fan entity selection can only select faces, not solids.

【Where is it】

Simulation >> New Fluid >> **Fan**

### 7.4.1.7 Add Flow Resistance Model

Flow resistance is a volumetric property that can be used to simulate porous medium materials (such as sponge, grid), narrow channel modules (such as heatsink), etc., by setting the drag coefficient to achieve equivalent pressure drop.

**【What users can do】**

- ✓ Set the viscous drag coefficient.
- ✓ Set the inertial drag coefficient.
- ✓ Set the porosity.

**【Notes】**

- For objects set as flow resistance, the material must be selected as fluid material, and solid materials cannot be selected.
- The flow resistance of the body and surface is based on different calculation methods; please choose carefully.

【Where is it】

Simulation >> New Fluid >> **Flow Resistance**

### 7.4.1.8 Add Inlet Boundary

The inlet is a type of flow boundary condition, serving as a mass source entering the fluid domain, and is divided into two types: inlet (along the coordinate axis) and angled inlet. The types of inlet can be velocity, volumetric flow rate, and mass flow rate. Additionally, users can set the inlet's density, temperature, pressure, external turbulence conditions, and more.

**【What users can do】**

- ✓ Set velocity, volumetric flow rate, mass flow rate, and normal velocity.

✓ Set the inlet with an angle.

**【Notes】**

- The geometry of the inlet needs to select a face, while the geometry of the angled inlet needs to select a body.

**【Where is it】**

Simulation >> New Fluid >> **Inlet**

### 7.4.1.9 Add outlet boundary

The outlet is a type of flow boundary condition, which is divided into two types: outlet (along the coordinate axis) and angled outlet. The outlet velocity type can be user-defined or In-cell. Additionally, users can set external pressure, external temperature, and external turbulence conditions for the outlet.

**【What users can do】**

✓ Set pressure and velocity outlet.

✓ Set the angled outlet.

**【Notes】**

- The geometry of the outlet needs to select a face, while the geometry of the angled outlet needs to select a body.

**【Where is it】**

Simulation >> New Fluid >> **Outlet**

### 7.4.1.10 Add Contact Resistance (ZW3D 2026 SP1)

The user specifies the thermal contact resistance at the interface between two solids, or between a solid and a fluid.

In the view area, select the surface where the thermal contact resistance is to be defined, or select an individual surface in the simulation tree to directly enter the value of the surface thermal resistance.

**【What users can do】**

✓ Set Contact Resistance.

**【Where is it】**

Simulation >> New Fluid >> **Contact Resistance**

### 7.4.1.11 Add Source (ZW3D 2026 SP1)

Create heat sources for certain components. By creating heat sources, you can analyze the transfer and distribution of heat generated by the heat sources within the system.

**【What users can do】**

✓ The entity type can be selected as an entity or a box.

✓ The source model can be selected as total heat, fixed temperature, or momentum source.

**【Where is it】**

Simulation >> New Fluid >> **Source**

### 7.4.1.12 Add Planar Thermal (ZW3D 2026 SP1)

Used to specify surface heat sources on solid surfaces in contact with a fluid, as well as on interfaces between solids.

**【What users can do】**

✓ In the graphics area, select the surface where the surface heat source is to be defined, or select an individual surface in the simulation tree for setup.

✓ You can choose from total power, fixed temperature, or heat flux. The data type can be set as a constant or a curve.

**【Notes】**

- In transient simulations, time-dependent curves can be edited.

**【Where is it】**

Simulation >> New Fluid >> **Planar Thermal**

### 7.4.1.13 Add Two-Resistor (ZW3D 2026 SP1)

Used to simplify the modeling of small electronic packaging components in electronic devices. It simulates the heat transfer process between heat sources (such as chips and electronic components) and the surrounding environment.

**【What users can do】**

- ✓ Supports selecting entities or creating boxes, and setting the corresponding normal direction.
- ✓ Node power type can be selected as constant or curve, then set the node-to-shell thermal resistance JT and node-to-substrate thermal resistance JB, and set the bottom layer thickness.

**【Notes】**

- The bottom layer thickness must not exceed the thickness of the two-resistor.

**【Where is it】**

Simulation >> New Fluid >>**Two-Resistor**

### 7.4.1.14 Add Electric Power (ZW3D 2026 SP1)

Electrical contact conditions can be specified either at the interface between two conductive solids or at the external boundary of a conductive solid. The possible types of electrical conditions include current and voltage.

**【What users can do】**

- ✓ Supports the creation of power sources. The power source settings help predict the voltage and current distribution of the circuit under different operating conditions. The power source type can be set to either voltage or current. After selecting the corresponding surface, specify the voltage or current value.

**【Notes】**

- When selecting the current type, the direction of current flow must be specified—choose either the positive or negative direction.

**【Where is it】**

Simulation >> New Fluid >>**Electric Power**

### 7.4.1.15 Add Electrical Resistance (ZW3D 2026 SP1)

Contact resistance can be specified on the surface of a conductive solid. In this case, the current passing through the resistance will generate the corresponding Joule heat.

**【What users can do】**

✓ Supports specifying either a direct resistance value or a resistivity combined with the material thickness

**【Where is it】**

Simulation >> New Fluid >>**Electric Resistance**

### 7.4.1.16 Add Monitor Point (ZW3D 2026 SP1)

Monitor point help users track and analyze key parameters in real-time during the simulation process, thereby better understanding the system's performance and behavior.

**【What users can do】**

✓ supports creating monitor points to monitor the required data, with monitor point types selectable as points or entities.

**【Where is it】**

Simulation >> New Fluid >>**Monitor Point**

### 7.4.1.17 Add New Physical Model

The physical model is used to set options for the type of solution scheme, PDE equation type, radiation model, turbulence model, gravity, and other options.

**【What users can do】**

✓ Supports simultaneous consideration of fluid flow and conjugate heat transfer, or cases with only flow.

✓ Supports Laminar flow model, LVEL, Standard KE, Chen-Kim KE, RNG KE, Kw SST, and Low Re Kw

SST turbulence.

✓ Supports custom gravity magnitude and direction.

✓ Supports Immersol and P1-T3 radiation models.

**【Notes】**

- Select the appropriate turbulence model based on the actual physical scenario.
- After changing the turbulence model, the corresponding inlet turbulence parameters will change; please make sure to modify them.

**【Where is it】**

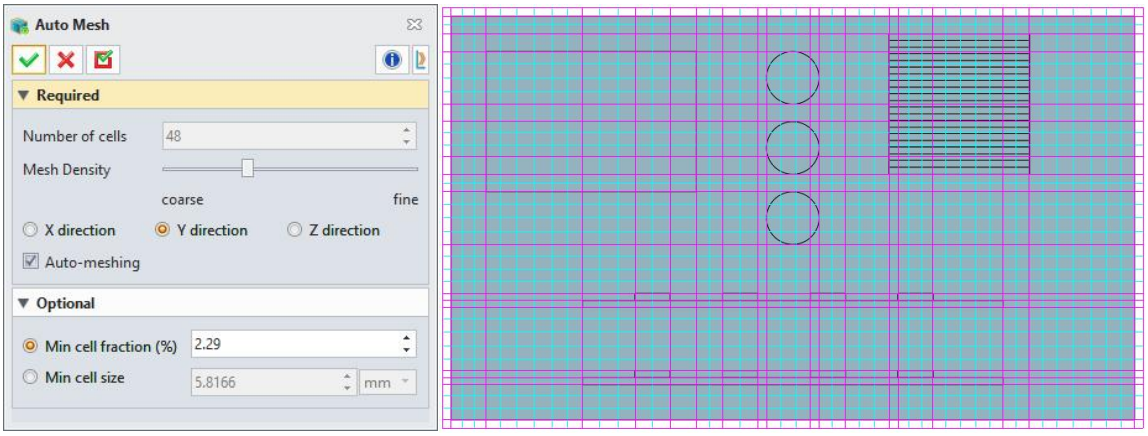
Simulation >> New Fluid >> Configuration >> **Physical Model**

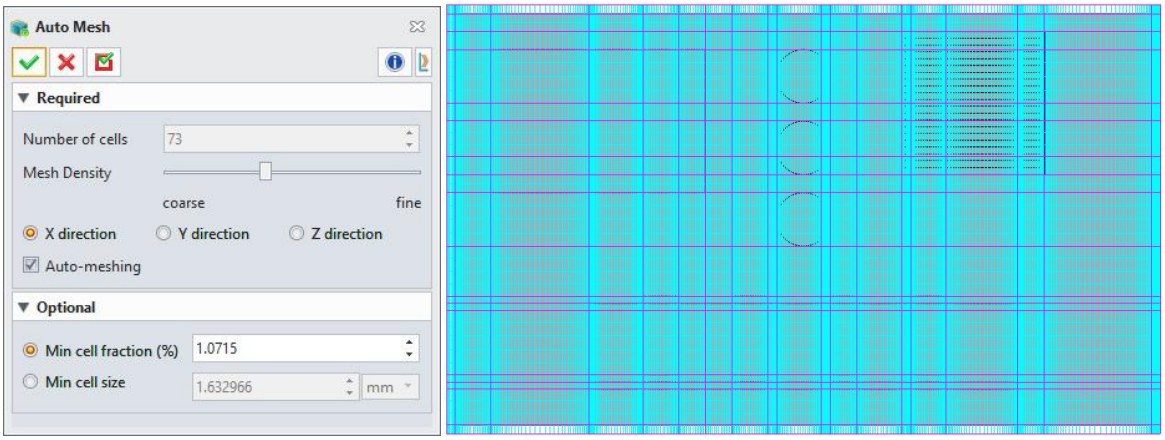
## 7.4.1.18 New automatic mesh division feature

Used to control the process of automatically constructing the mesh, specifying the density of the global mesh, which can be adjusted by dragging a slider to change the number of meshes.

**【What users can do】**

✓ Individually adjust the mesh density in three directions.






Mesh refinement

**【Where is it】**

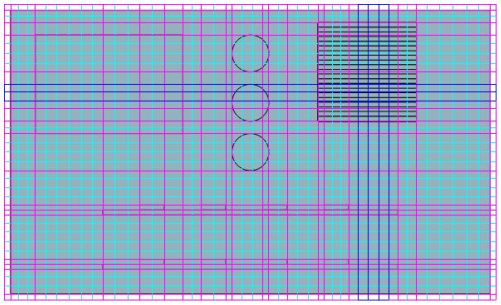
Simulation >> New Fluid >> Mesh Generation>> **Auto Mesh**

## 7.4.1.19 New mesh editing feature

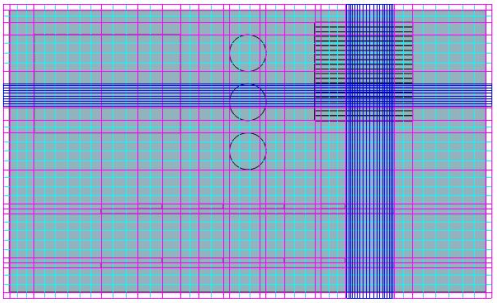
Manually adjust the number and distribution of meshes within a single area, locally refine, and enhance the computational accuracy of critical areas of the model.

**【What users can do】**

- ✓ Select different cutting element methods SPARSOL and PARSOL.
- ✓ Customizable mesh tolerance to eliminate the impact of small gaps on the mesh.
- ✓ Supports mouse clicks in the view area to select different mesh regions for editing.
- ✓ Supports custom mesh quantity and distribution methods for any specified area.



Before Adjustment



After Adjustment

【Where is it】

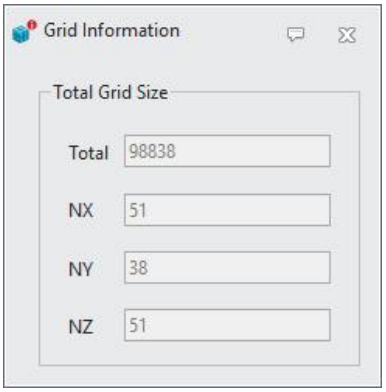
Simulation >> New Fluid >> Mesh Generation >> **Edit Grid**

### 7.4.1.20 New Mesh Statistics Functionality

Statistics on the total number of model meshes, as well as the number of meshes in the X/Y/Z directions.

**【What users can do】**

✓ The number of grids in all directions and the total number of grids in the whole domain can be counted




【Where is it】

Simulation >> New Fluid >> Mesh Generation >> **Grid Information**

## 7.4.2 New Post-Processing Functionality

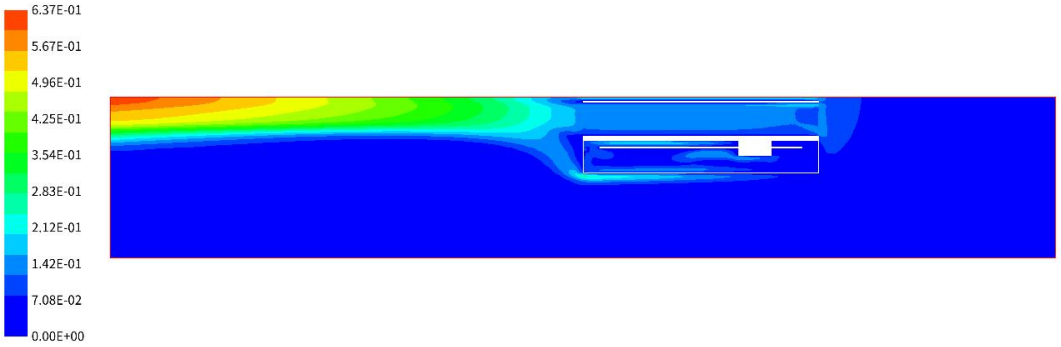
### 7.4.2.1 New Planar Contour Functionality

In the post-processing of the simulation, it is necessary to observe the distribution and variation of the physical field on different planes at various locations.

**【What users can do】**

✓ Planar contour distribution under different variables.

✓ Supports contour display at any position parallel to the default plane or on any specified plane.




**【Notes】**

- After the calculation is completed, it automatically switches to post-processing; before the calculation, the post-processing module cannot be accessed.

**【Where is it】**

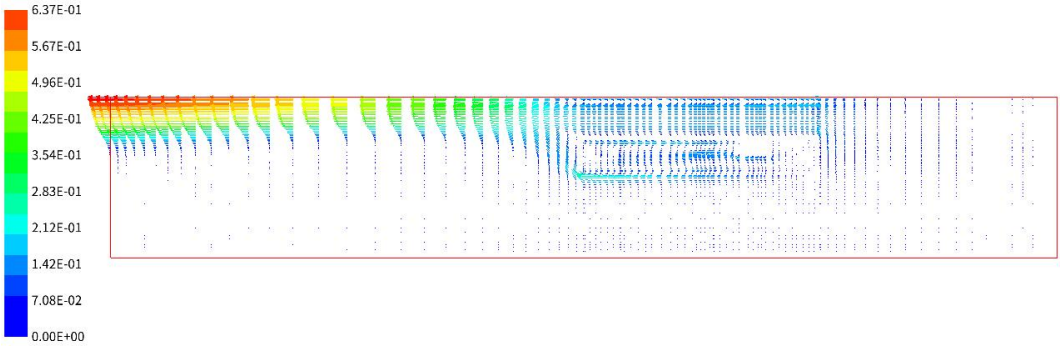
Post Processing >> Visualization >> **Plane contour**

### 7.4.2.2 New Planar Vector Functionality

In the post-processing of the simulation, it is necessary to observe the distribution and variation of the physical field on different planes at various locations.

**【What users can do】**

- ✓ Planar vector distribution under different variables.
- ✓ Supports vector display at any position parallel to the default plane or on any plane.
- ✓ Supports setting the line width of the vectors.
- ✓ Supports adjusting the arrow size.
- ✓ Supports planar arrows or 3D arrows.




**【Where is it】**

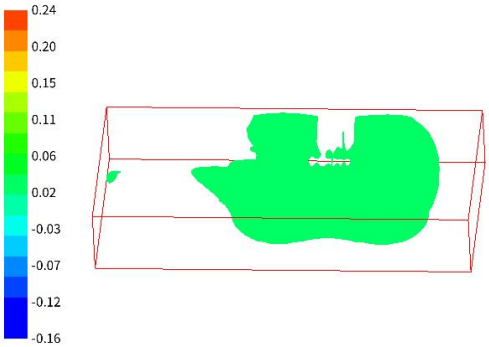
Post Processing >> Visualization >> **Plane Vector**

### 7.4.2.3 New Isosurface Functionality

In the simulation post-processing, it is necessary to observe the distribution of specific variable values.

**【What users can do】**

- ✓ Isosurface display under different variables.




**【Where is it】**

Post Processing >> Visualization >> **Iso-surface**

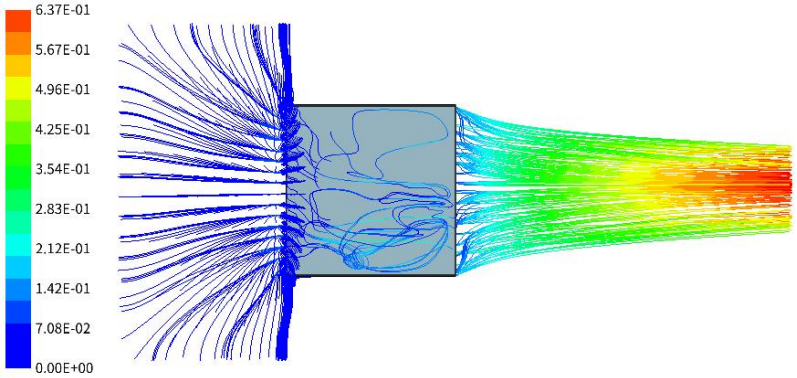
### 7.4.2.4 New Streamline Feature

In the simulation post-processing, the flow trajectory can be observed by creating streamlines.

**【What users can do】**

- ✓ Streamline display under different variables.

✓ Coloring can be based on the values of different variables.




**【Where is it】**

Post Processing >> Visualization >> **Streamline**

### 7.4.2.5 New Data Statistics Feature

In the simulation post-processing, parameters such as mass flow rate and power at the boundaries of the computational domain and objects can be statistically analyzed.

**【What users can do】**

✓ By default, the flow rate at the boundaries of the computational domain is statistically analyzed.

✓ Different objects can be selected for relevant data statistics.

**【Where is it】**

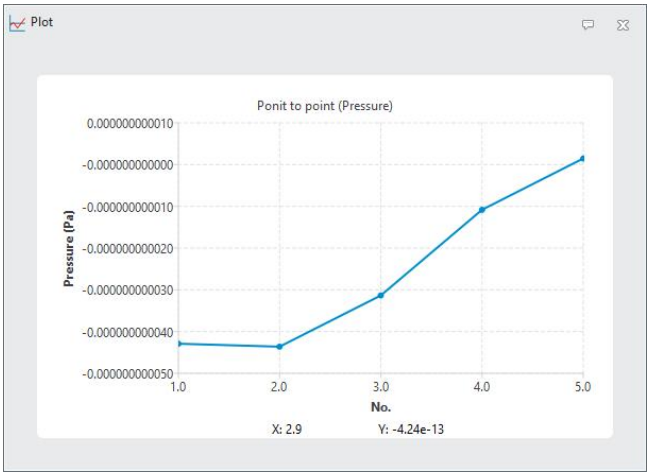
Post Processing >> Visualization >> **Statistics**

### 7.4.2.6 New 2D Plotting Feature

In the simulation post-processing, two points can be selected, the number of points between the two can be determined, and the corresponding variable can be selected to plot the curve of the variable's change with position.

**【What users can do】**

✓ Variable curves that change with position.




**【Where is it】**

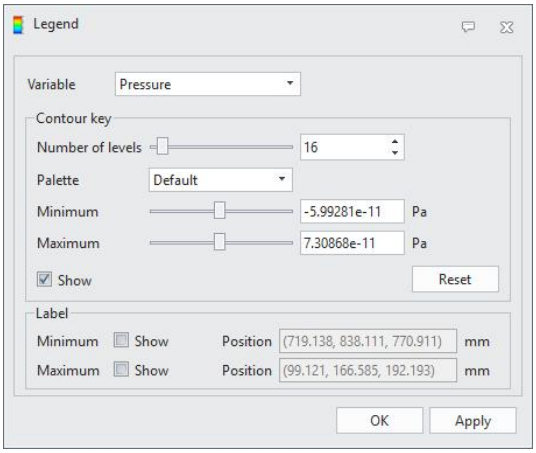
Post-Processing >> Visualization >> **Graph**

## 7.4.2.7 New legend feature

In the simulation post-processing, customize the legend display.

**【What users can do】**

- ✓ Different levels of legends.
- ✓ Adjusting the legend range.
- ✓ Display maximum and minimum values.




**【Where is it】**

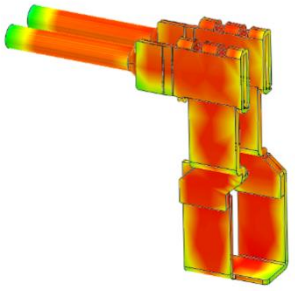
Post Processing >> Visualization >> **Legend**

## 7.4.2.8 New Surface Contour feature (ZW3D 2026 SP1)

Display the variation of physical quantities on the model surface in the form of a contour plot, where color gradients visually represent local differences.

**【What users can do】**

- ✓ Select the corresponding solid to create a surface contour plot.



**【Where is it】**

Post Processing >> Visualization >> **Surface Contour**

# 8 PDM

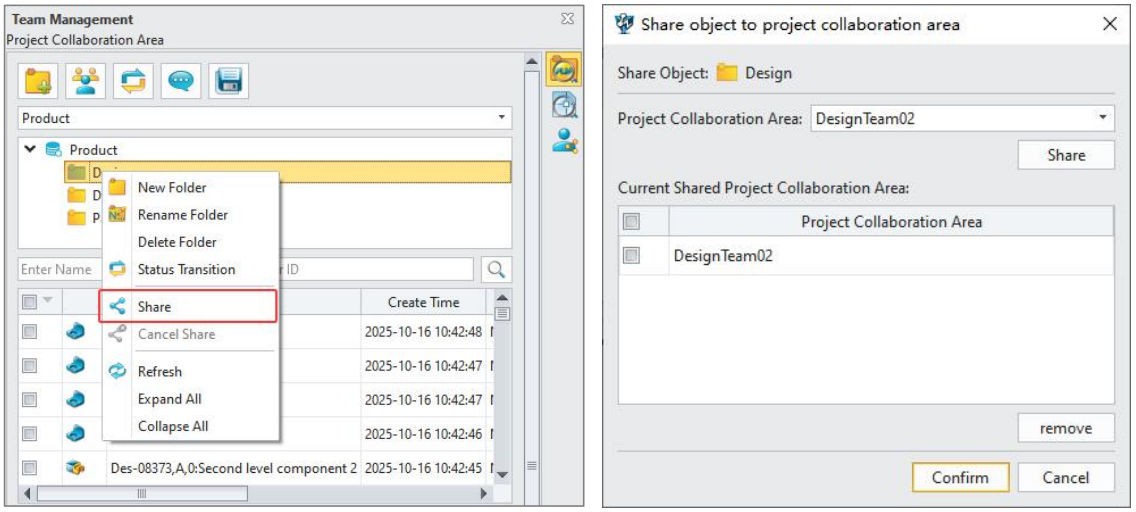
## 8.1 Project Collaboration Area Management（ZW3D 2026 SP1）

### 8.1.1 Add Project Collaboration Area Data Share Feature

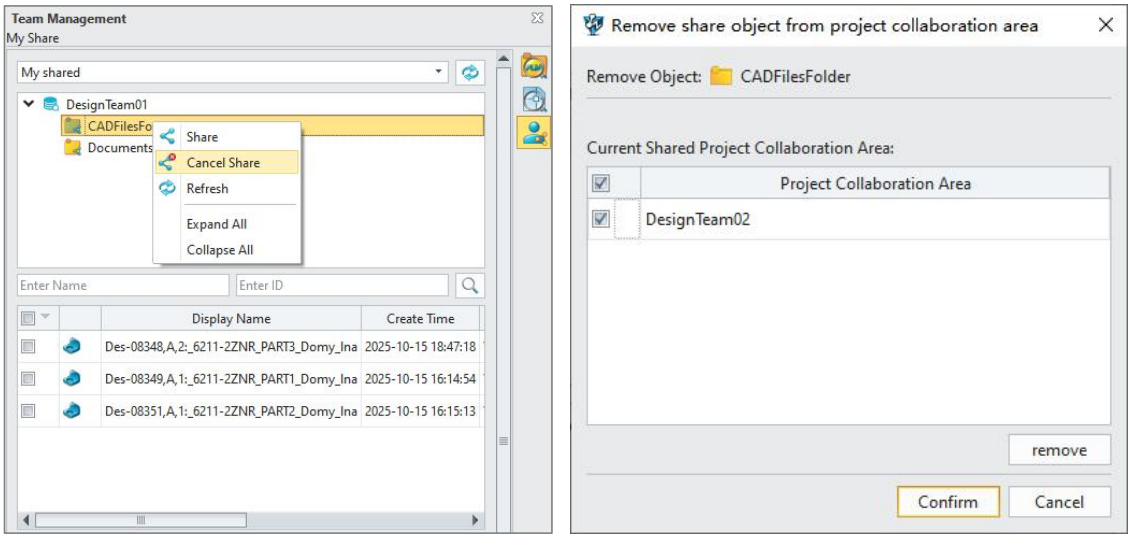
In the collaborative mode, supports sharing project collaboration area data (including designs, parts and documents) with other project collaborative teams by folder to achieve collaboration between collaborative teams.

**【What users can do】**

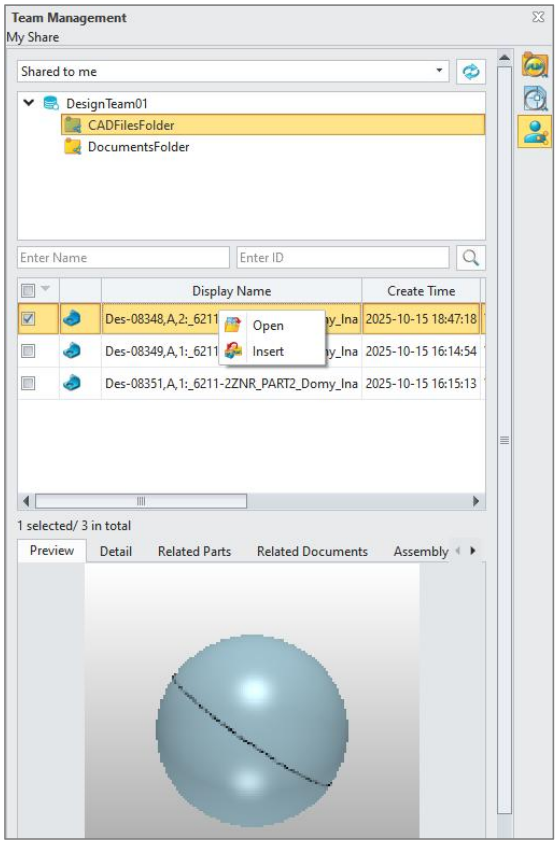
✓ The project collaboration area administrator can share the data of a folder in the collaboration area with other collaboration teams, so that members of other collaboration teams can view and use the data within the collaboration area.




✓ Shared folders are allowed to be removed from sharing. Once removed, other members of the collaborative team will no longer be able to view the data in that folder.

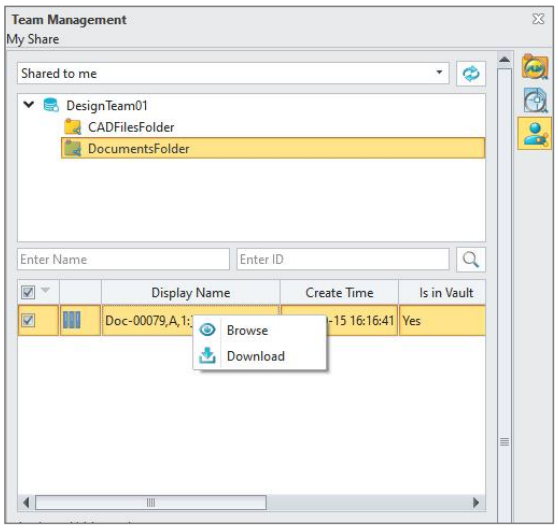



✓ The design object shared with me can have its model file inserted into the assembly being designed, or opened for viewing.




✓ The document object shared with me can be viewed for its document content, or downloaded

and saved.




**【Where is it】**

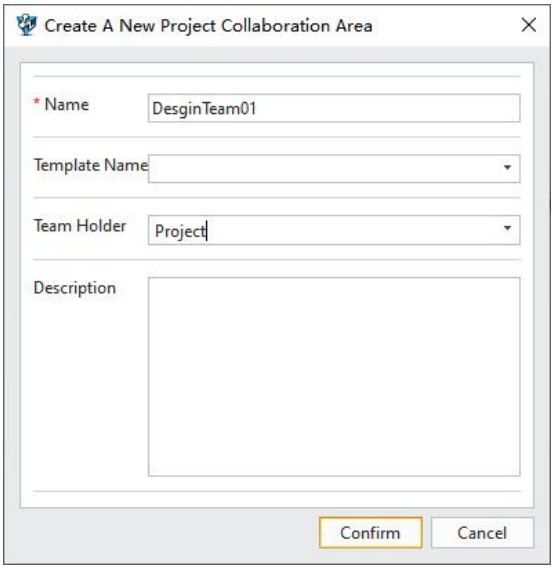
ZW3D >> ZWTeammate >> **Project Collaboration Area**

## 8.1.2 Optimize Relationship between Collaborative Teams and Project Teams

In collaborative mode, it is allowed to form multiple collaborative teams within the project team to complete the design of each team, and to conduct more granular management through collaborative teams.

**【What users can do】**

✓ When creating a collaborative team, you can choose to associate the project team with it to record the relationship between the collaborative team and the project team.




**【Where is it】**

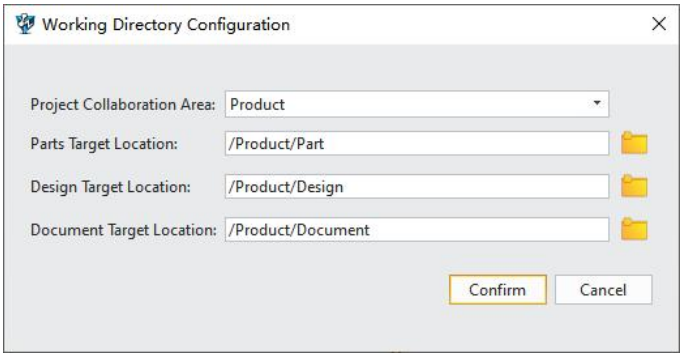
ZW3D >> ZWTeammate >> Project Collaboration Area >> **Create Collaboration Area**

## 8.1.3 Optimize Workspace Management

Workspace management is designed to solve the problem of collaborative design among engineers. The new version adds a personal workspace directory mirror on the server side to ensure that modifications in the design can be synchronized and saved at any time, making it convenient for engineers to design on different computers. At the same time, using local workspace cache can avoid frequent communication with the backend and reduce operational lag.

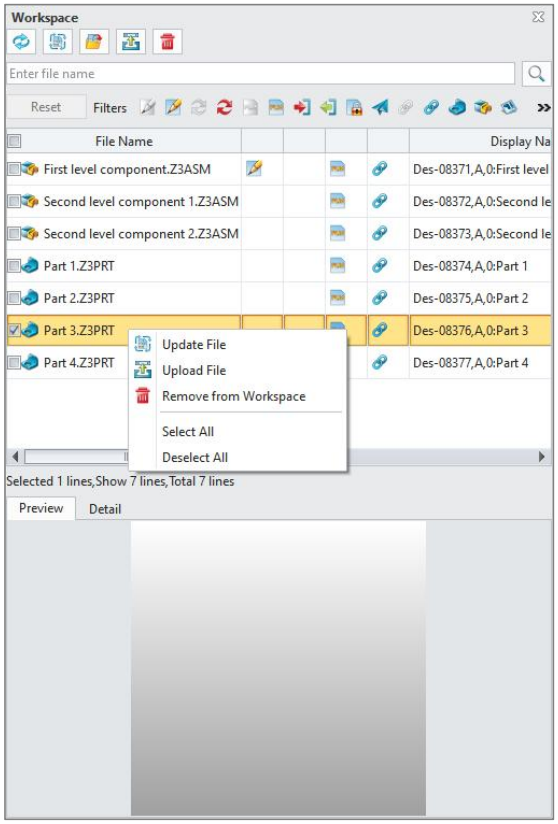
**【What users can do】**

✓ Automatically generate a local personal workspace by project collaboration area or main folder, and switch with the selected project collaboration area ormain folder.

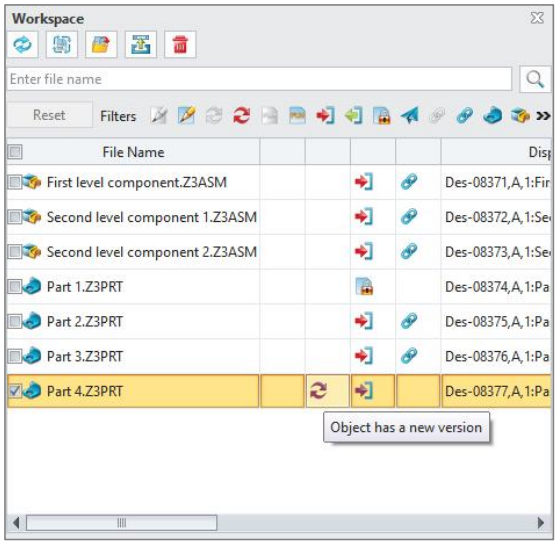



✓ The server has added a personal workspace mirror that supports synchronizing personal workspace data across multiple devices.

✓ Support separate management of personal workspace files, such as upload file, update file, and remove.




✓ Timely obtain the latest modification status of personal workspace data.



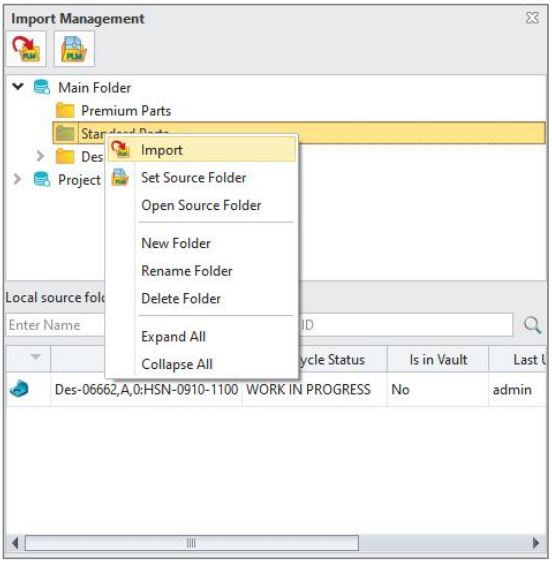

【Where is it】

ZW3D >> ZWTeammate >> **Configuration Manager**

ZW3D >> ZWTeammate >> **Workspace**

## 8.1.4 Adjust The Position of Importing Standard Parts

In collaborative mode, it supports importing standard parts into both the main folder directory and the project collaboration area directory.




【Where is it】

## 8.1.5 Add Non Collaborative Folder Working Mode

Some users do not need the complex and rigorous management method of collaborative management. They only need to put product data into a shared server and manage it through folders. In this scenario, Teammate supports switching to non collaborative mode. In non collaborative mode, all integrated objects (materials, designs, documents) are stored in the main folder and no longer provide access to the project collaboration area.

**【What users can do】**

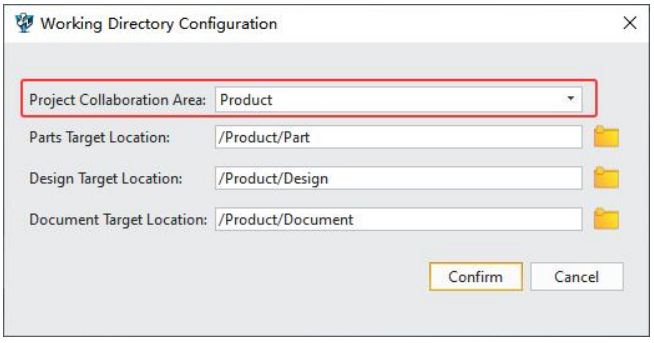
- ✓ Set the folder working mode through the preference FolderType. ProjectTeam represents only the main folder management mode, while GenTeam represents the project collaboration area management mode.

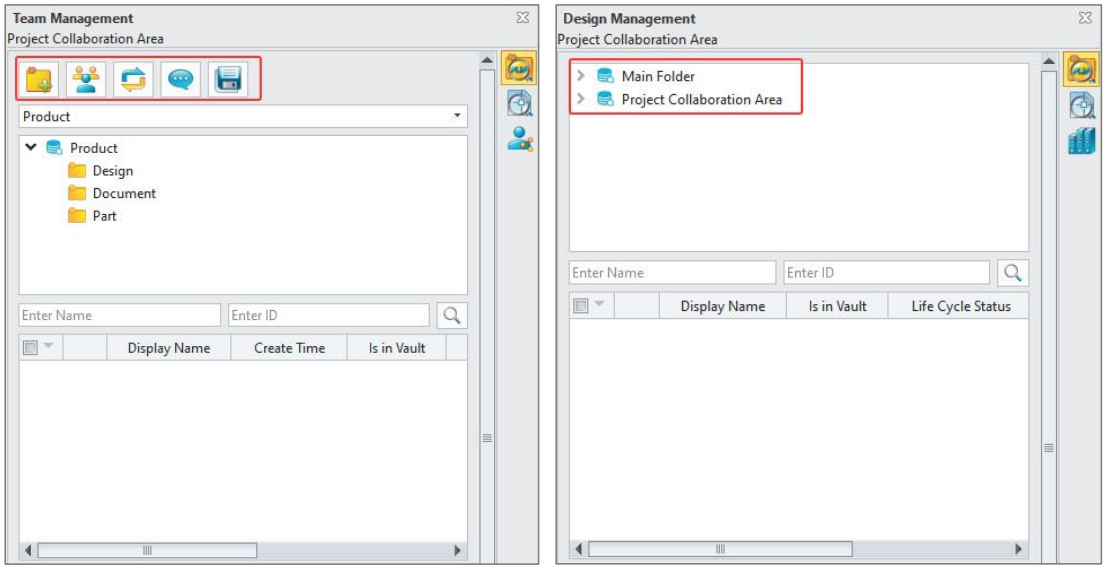
**【Notes】**

- Before applying the system, it is necessary to determine the working mode of the folder. If switching is required later, it may lead to the implementation of data migration.

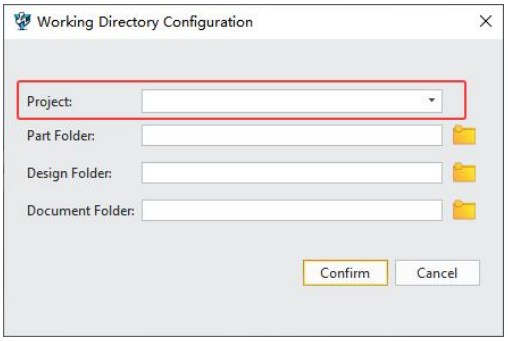
**【Example】**

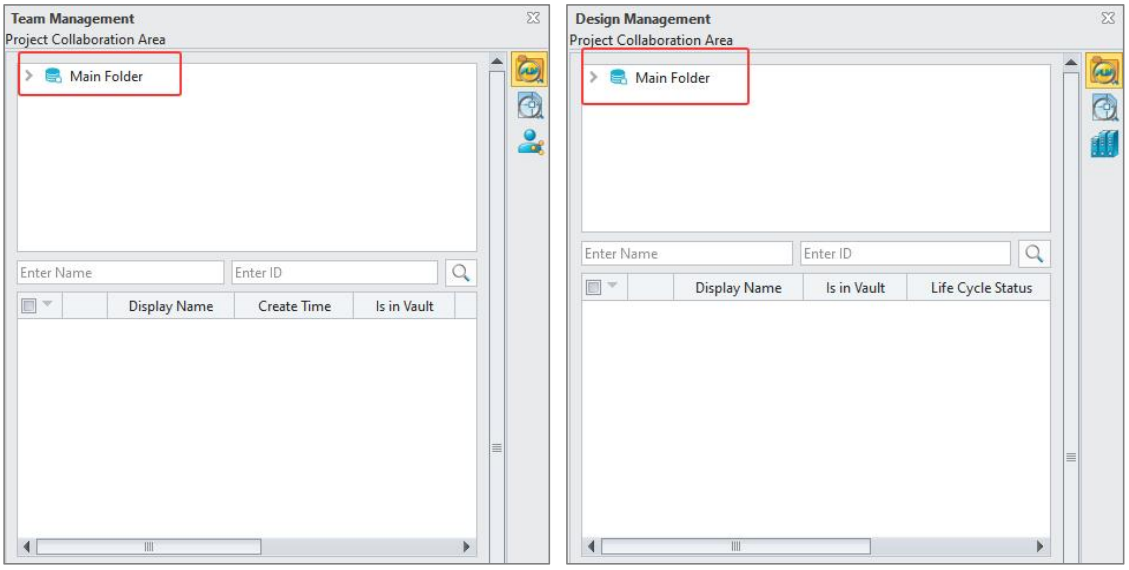
1) GenTeam working mode：






2) ProjectTeam working mode：






**【Where is it】**

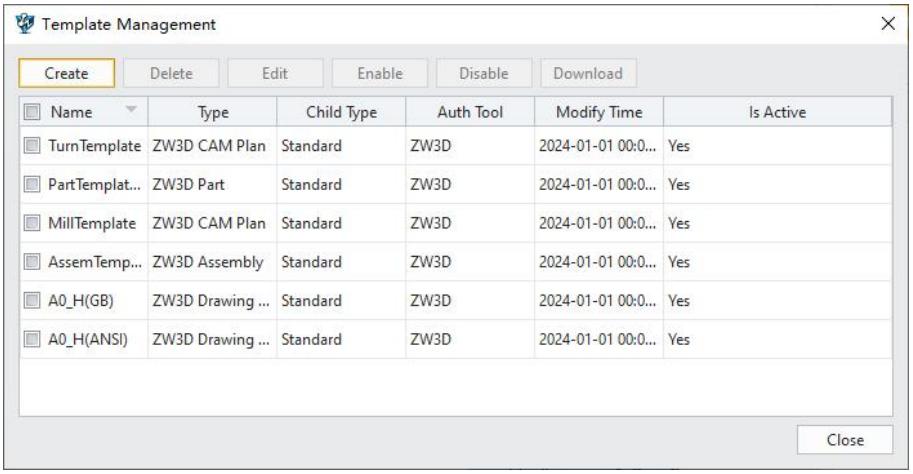
External browser environment >> Teammate management terminal >> System >> System Expansions >> **Preference Config**

## 8.2 Design Management

Similar to ZW3D's file creation workflow, ZWTeammate now supports creating parts, assemblies, or drawings using predefined templates with configurable attributes. Upon submitting 3D models to ZWTeammate, the system automatically extracts and maps template attributes.

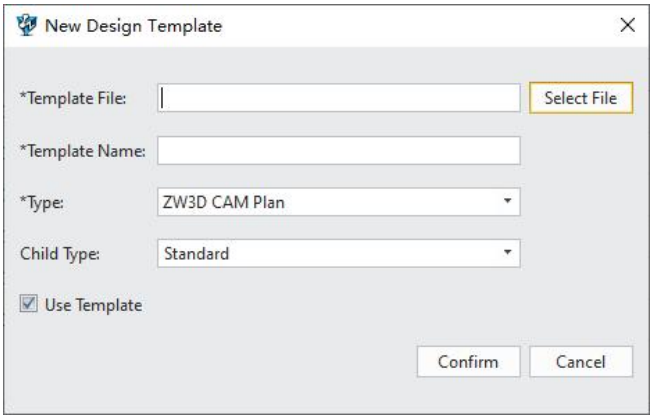
### 8.2.1 Template Management for Design

Add a design template management feature, including preset 3D universal templates, and configure the mapping between ZW3D template attributes and ZWTeammate attributes. The goal is to move locally maintained template files to online centralized management to prevent outdated local templates, ensuring template consistency and facilitating updates.

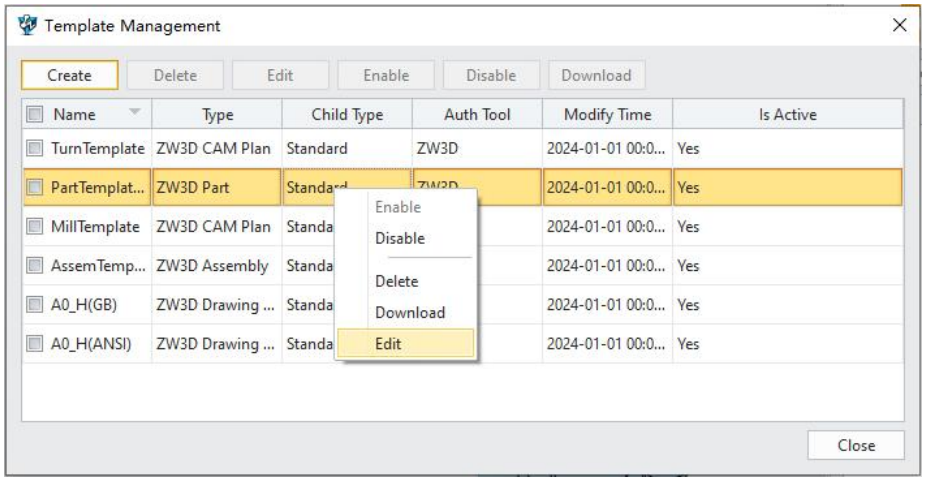



**【What users can do】**

✓ Users can easily add new design templates, either by creating them from existing templates or using model files, to meet personalized requirements.




✓ Manage templates via Delete, Edit, Enable, Disable, or Download.

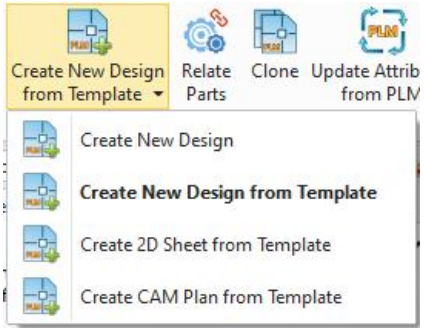



**【Where is it】**

ZW3D >> ZWTeammate >> **Template Management for Design**

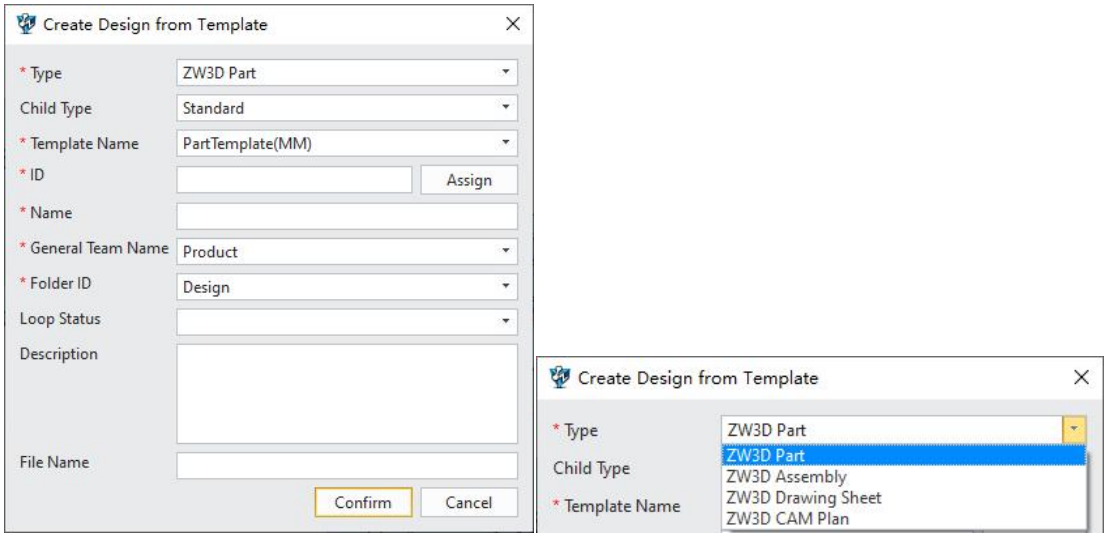
## 8.2.2 Create New Design from Template

New features include "Create New Design from Template", "Create 2D Sheet from Template", and "Create CAM Plan from Template". These options allow users to select standardized and up-to-date templates from the system to Create New Design from Template files, ensuring consistency in attributes, layers, and other critical information.




**【What users can do】**

✓ Create New Design from Template: Generate Parts, Assemblies, 2D Sheets, or CAM Plans standardized templates.




✓ Create 2D Sheet from Template: Supports creating a drawing file related with the current part or assembly model based on existing drawing templates in the system

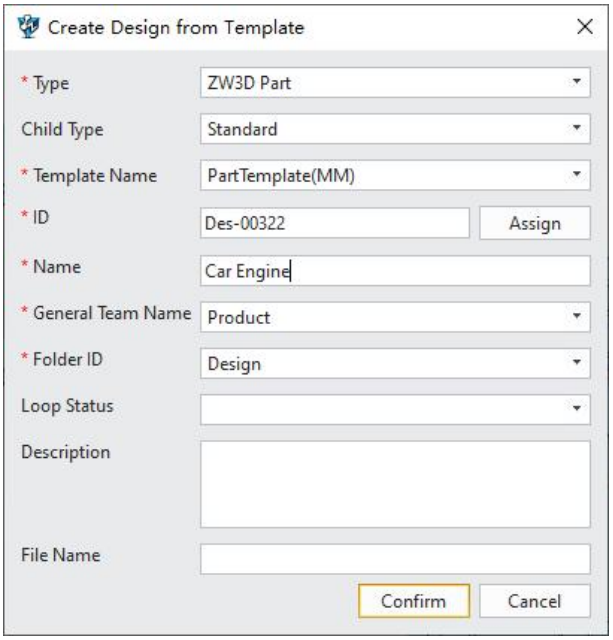
✓ Create CAM Plan from Template: Supports creating a machining plan file related with the current part or assembly model based on existing CAM plan templates in the system.

**【Notes】**

- The option to create 2D Sheet from template or CAM Plan from template is only available when a part or assembly model is already open.
- When creating a new design file from an existing template, a design object will be automatically generated by default.

**【Example】**

1) Select the file type and template name, enter the relevant attribute information for the design object, and click "Confirm" to complete the creation of both the file and the design object.




2) The available options for Type , Child Type, and Template Name are derived from the content maintained in the Design Template Management section.

**【Where is it】**

ZW3D >> ZWTeammate >> Object Creation/Update >> **Create New Design from Template**

ZW3D >> ZWTeammate >> Object Creation/Update >> **Create 2D Sheet from Template**

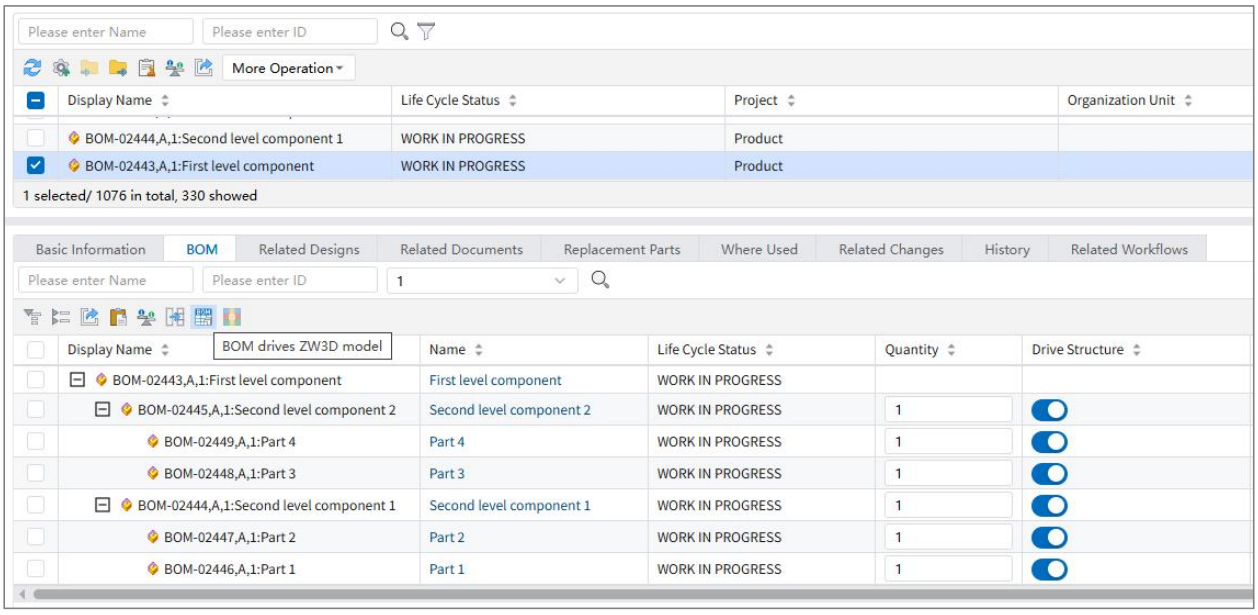
ZW3D >> ZWTeammate >> Object Creation/Update >> **Create CAM Plan from Template**

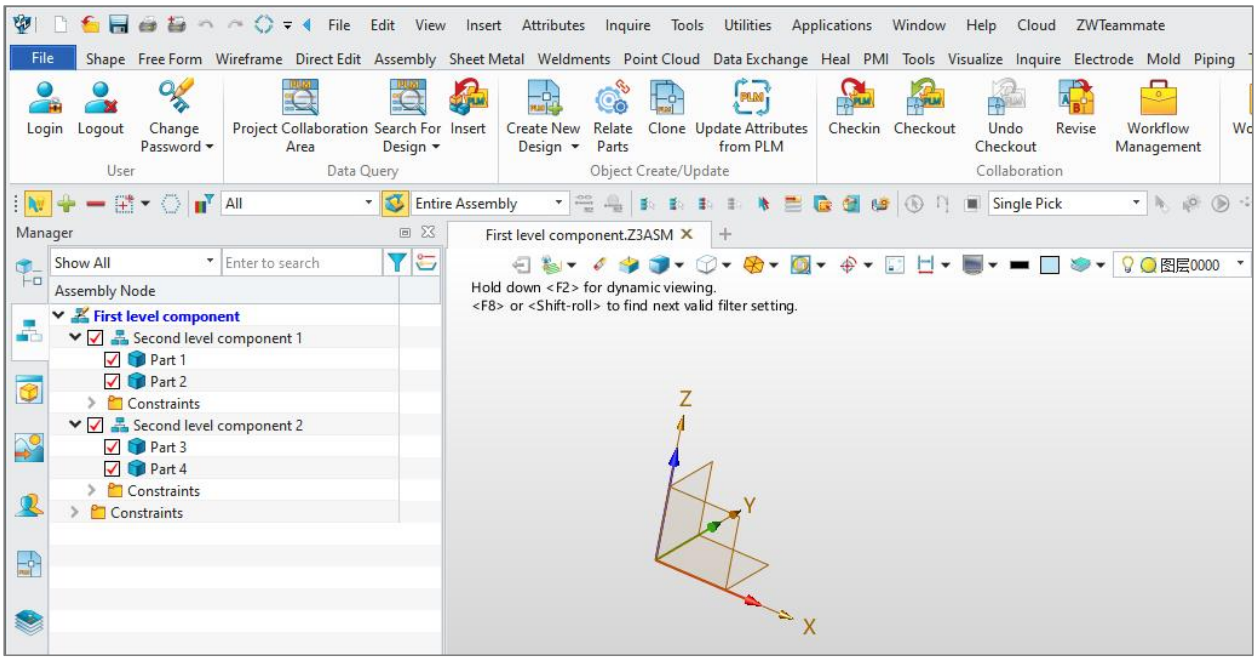
## 8.2.3 Add BOM Drives ZW3D Model Feature（ZW3D 2026 SP1）

Without starting ZW3D, the generation or update of model structure is driven by BOM data, and users can load and add components by opening the corresponding assembly model in ZW3D.

**【What users can do】**

✓ Insert one or more existing parts into the BOM, and by executing the "BOM drives ZW3D model" function, the model files of newly added parts can be automatically written into the higher-level assembly model.




**【Notes】**

- The parts involved in driving the model structure in the BOM must have been related with a design object(ZW3D model file).
- The attribute “quantity” of BOM line must be 1 to be used as a component of the driving structure. If it is greater than 1, the system will automatically turn off the driving structure switch

for that line.

- The function entrance of "BOM drives ZW3D model" can be turned on or off through the system preference BOM.driveDesign. Enabled.

**【Where is it】**

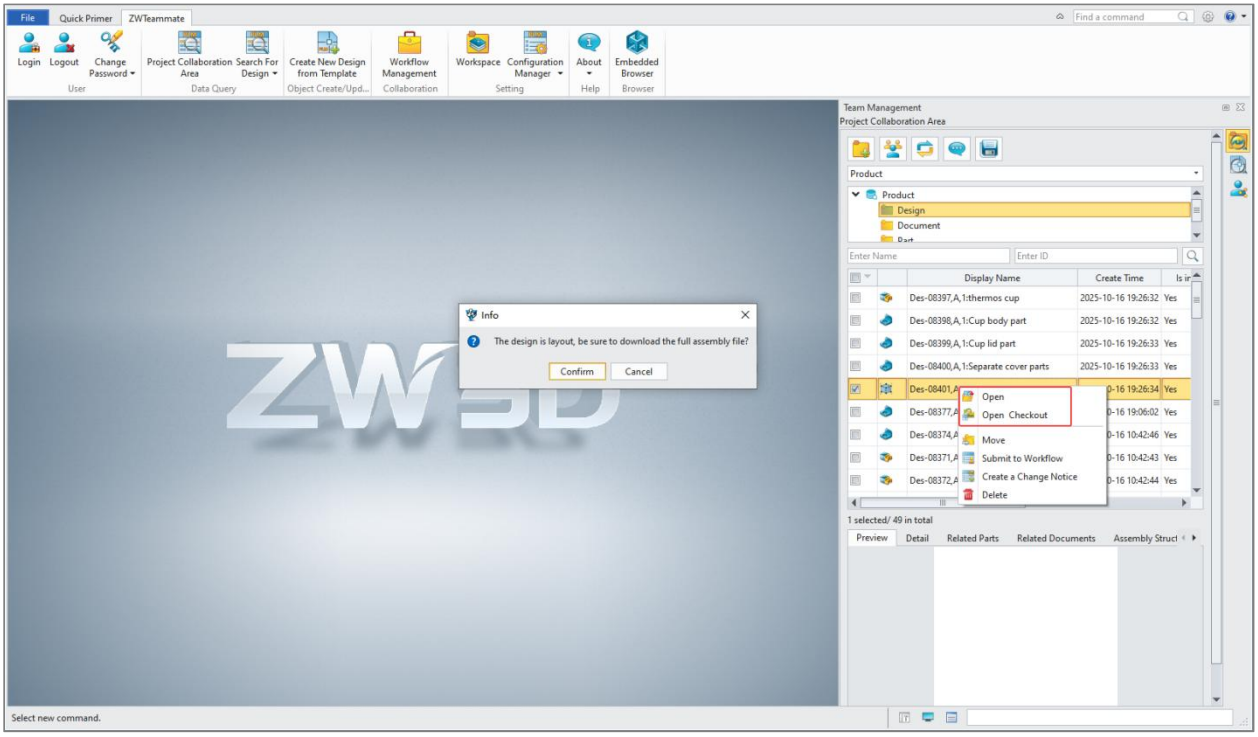
Browser environment >> Part >> BOM tab >> **BOM drives ZW3D model**

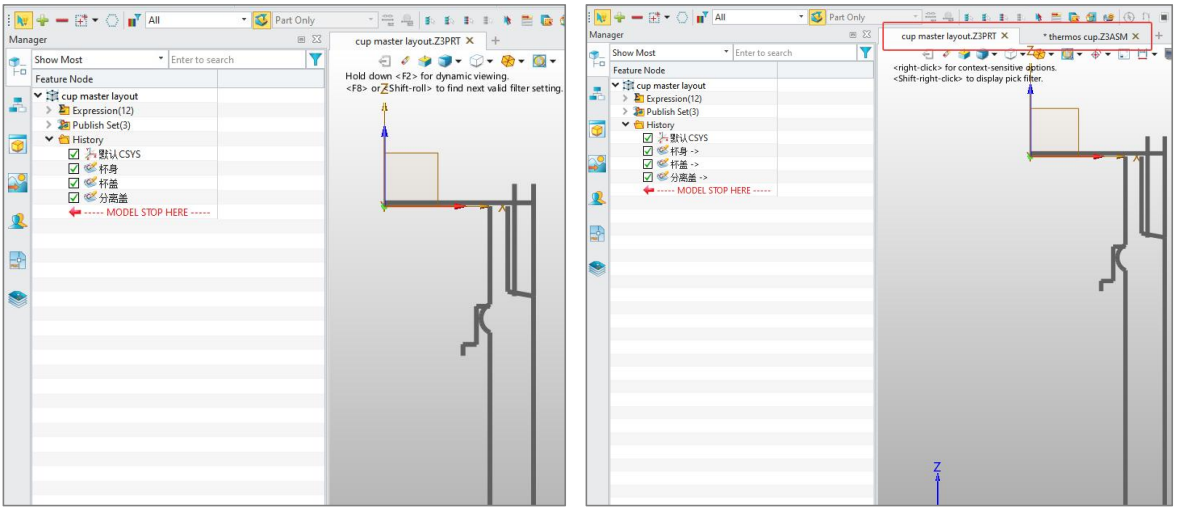
## 8.2.4 Optimize File Loading When Opening Master Layout（ZW3D 2026 SP1）

When opening or downloading the master layout from Teammate in the previous version, the assembly model was not loaded together, resulting in the loss of the reference relationship of the master layout. It was necessary to manually download the assembly once to solve the problem. The new version has improved this aspect. When users open the Teammate master layout in ZW3D, they will automatically obtain the higher-level assembly model and load it synchronously.

**【What users can do】**

✓ When opening the master layout, you can choose to open only the master layout (including the upper level assembly) or open the complete assembly (including all models).





【Where is it】

ZW3D >> ZWTeammate >> Project Collaboration Area >> **Open**

ZW3D >> ZWTeammate >> Project Collaboration Area >> **Open Checkout**

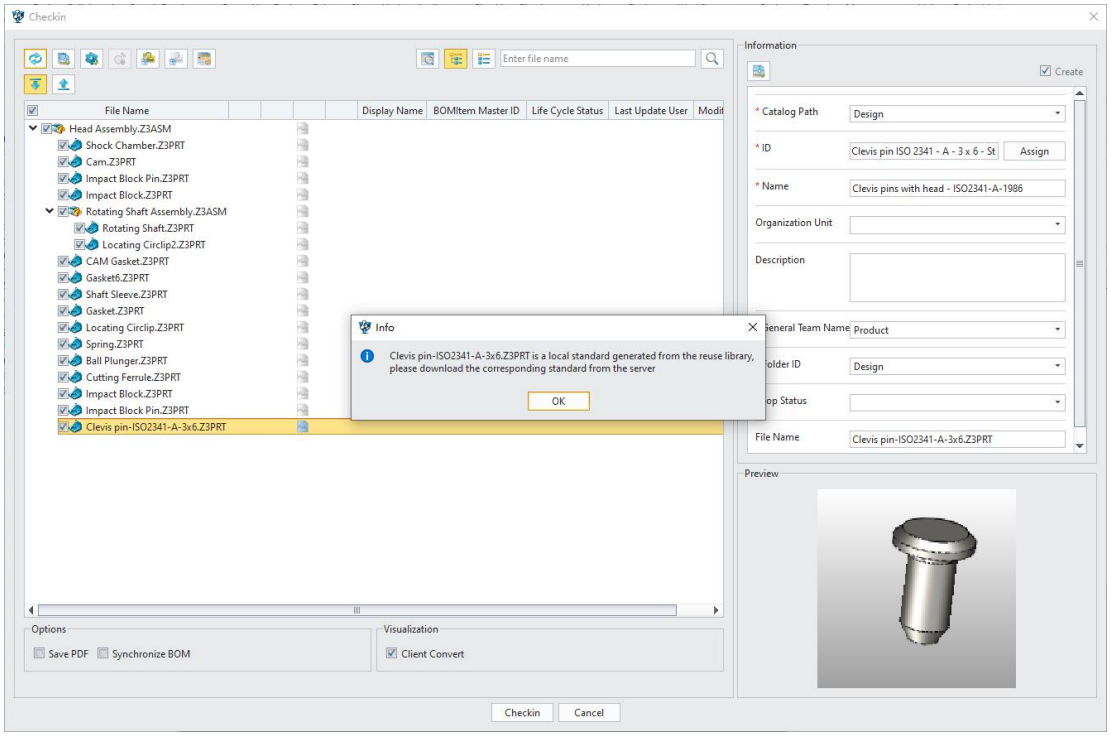
## 8.3 Reusable Resource Standardization

The system supports identifying local reusable library model files and performing targeted operations: a reuse identifier is added to the model to distinguish between local standard parts and server standard parts, preventing misuse.

**【What users can do】**

✓ If historical drawings are identified as local standard parts, the system will prompt and block the creation of new designs.

✓ If historical assembly structures are identified as containing local standard parts, the system will prompt and block automatic creation operations during the Checkin process.




✓ If a local standard part is inserted into an assembly, the system will issue a prompt.

**【Notes】**

- This option is configurable, allowing users to enable or disable the identification and blocking of local standard parts based on actual usage scenarios.
- The blocking operation only takes effect when logged into ZWTeammate.

## 【Where is it】

ZW3D >> ZWTeammate >> **Create New Design**

ZW3D >> ZWTeammate >> Checkin >> **Auto Create Design**

ZW3D >> Reuse Library >> **Select Standard Parts for Assembly Insertion**

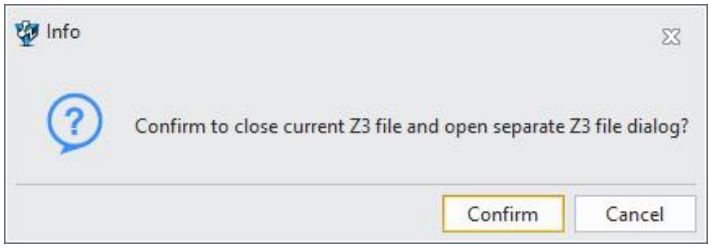
## 8.4 Z3 File Splitting

When the currently open file is identified as a Z3 format, a prompt should be added during the integration operation to inform the user that they must perform a file separation operation first.

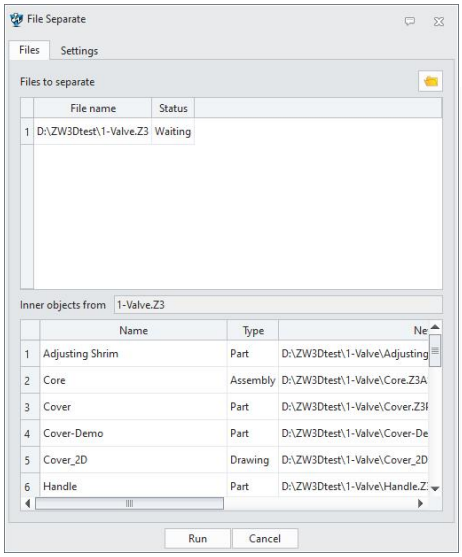
**【What users can do】**

✓ If the current model is identified as being in Z3 format, clicking "Create New Design" will prompt the user to enter the file separation interface.

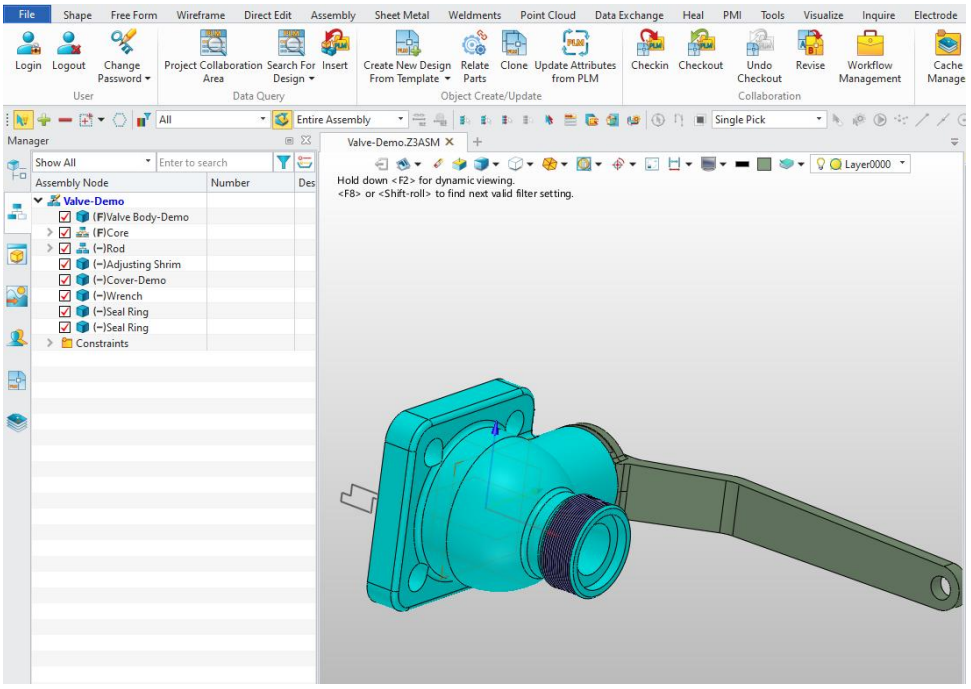
✓ If the current model is identified as being in Z3 format, clicking "Checkin" will prompt the user to enter the file separation interface.




✓ The system will use the standard file separation function in ZW3D to split the Z3 file.



✓ After separating the model files, proceed with the Create new design or Checkin operations on the integration side.




**【Notes】**

- During the file separation process for Z3 models, a new folder named after the model will be created in the current working directory, and all separated files will be placed in this folder.

**【Where is it】**

ZW3D >> ZWTeammate >> **Create New Design**

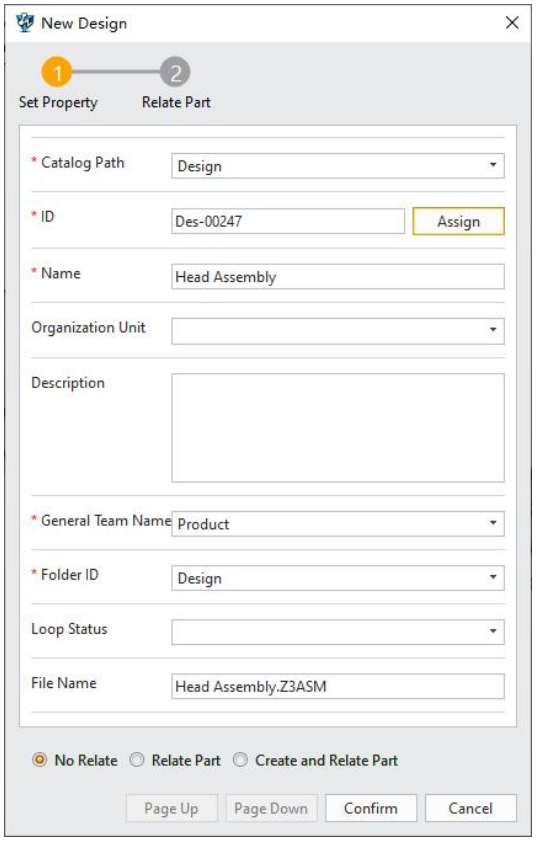
ZW3D >> ZWTeammate >> **Checkin**

## 8.5 Relate Parts

The Relate Parts feature allows linking design objects with parts. When either the design object or part is modified, the related information is automatically updated to ensure data continuity and consistency. By Relate Parts, the system can generate BOM (Bill of materials) based on the assembly structure and the relationships between design objects and parts, enabling automatic data synchronization and efficient BOM generation.

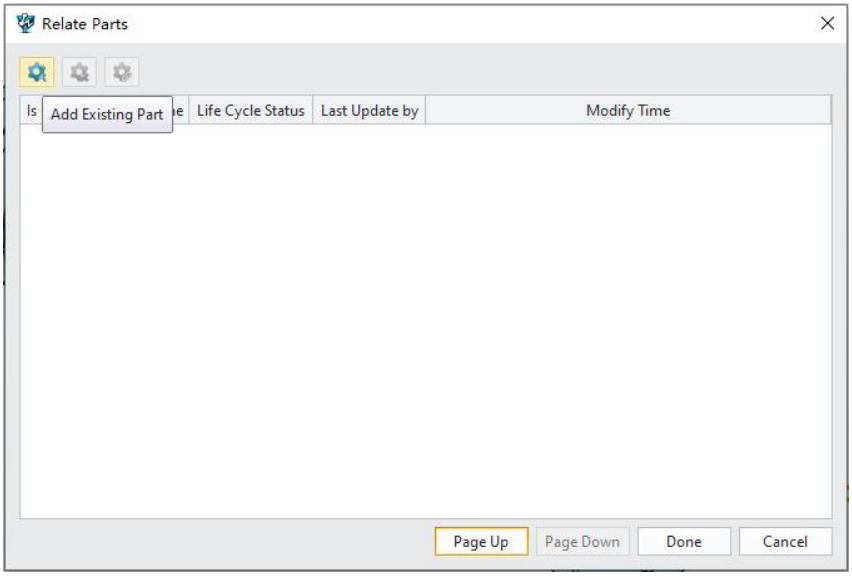
### 8.5.1 Synchronous Relate Part During Design Creation

When creating a design object, the system supports synchronously Relate Part.

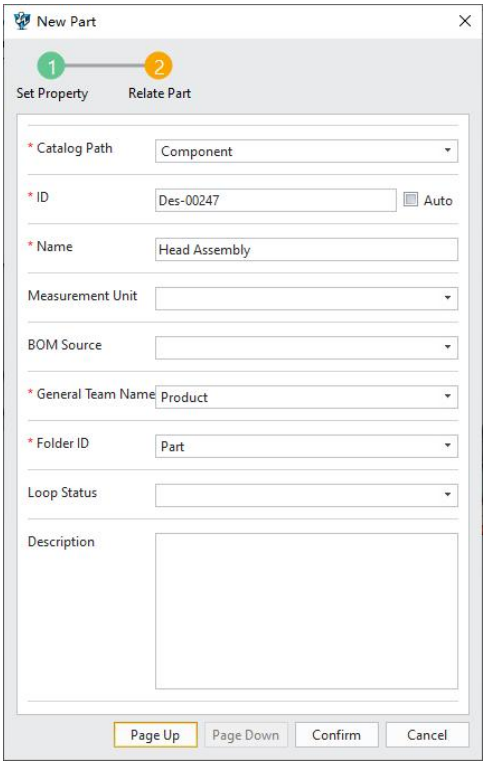



**【What users can do】**

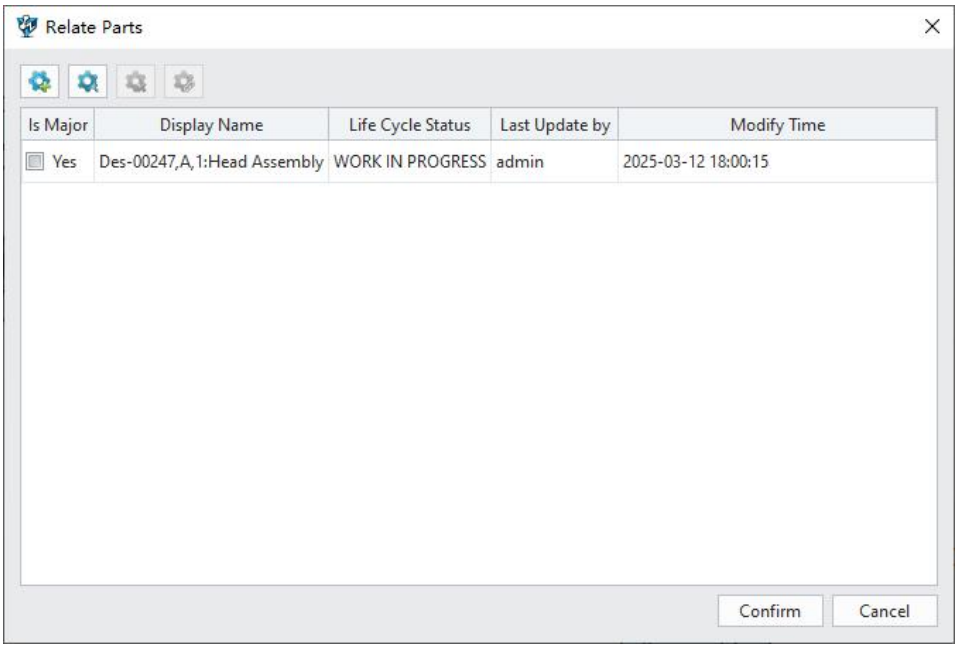
- ✓ Select "Relate Part" to search and link existing parts from the system.




- ✓ Select "Create and Relate Part" to create a new part and link it to the design object.




New Part




Relate Part

**【Notes】**

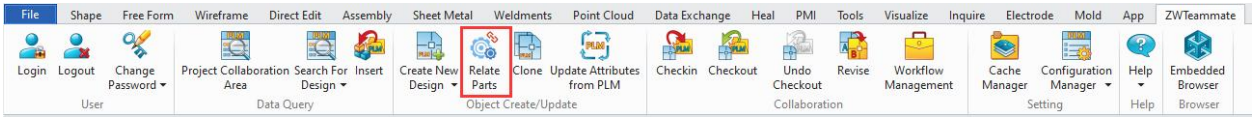
- When creating and Relate Part, if the object numbering rules have been configured in the system, users can check the "Auto" option. Upon saving the data, the system will generate a new number based on the predefined numbering rules.

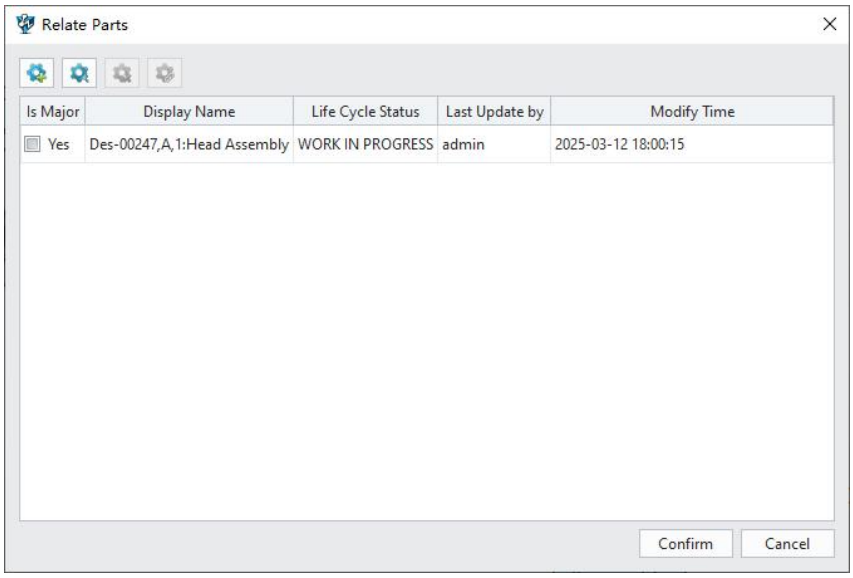
**【Where is it】**

ZW3D >> ZWTeammate >> Create New Design >>**Relate Part / Create and Relate Part**

## 8.5.2 Dedicated Relate Part Portal

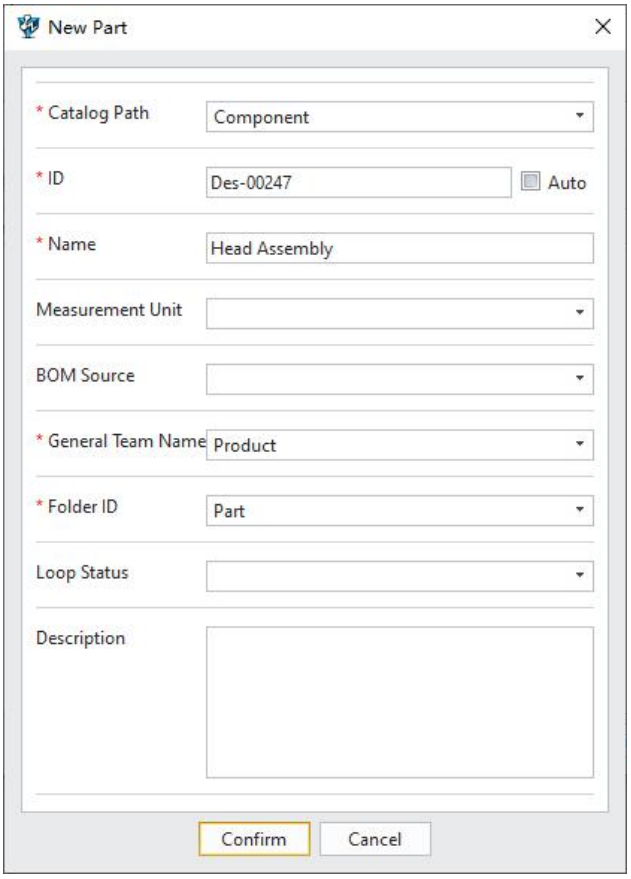
A dedicated "Relate Parts" entry has been added to facilitate managing the association between the current design object and parts.



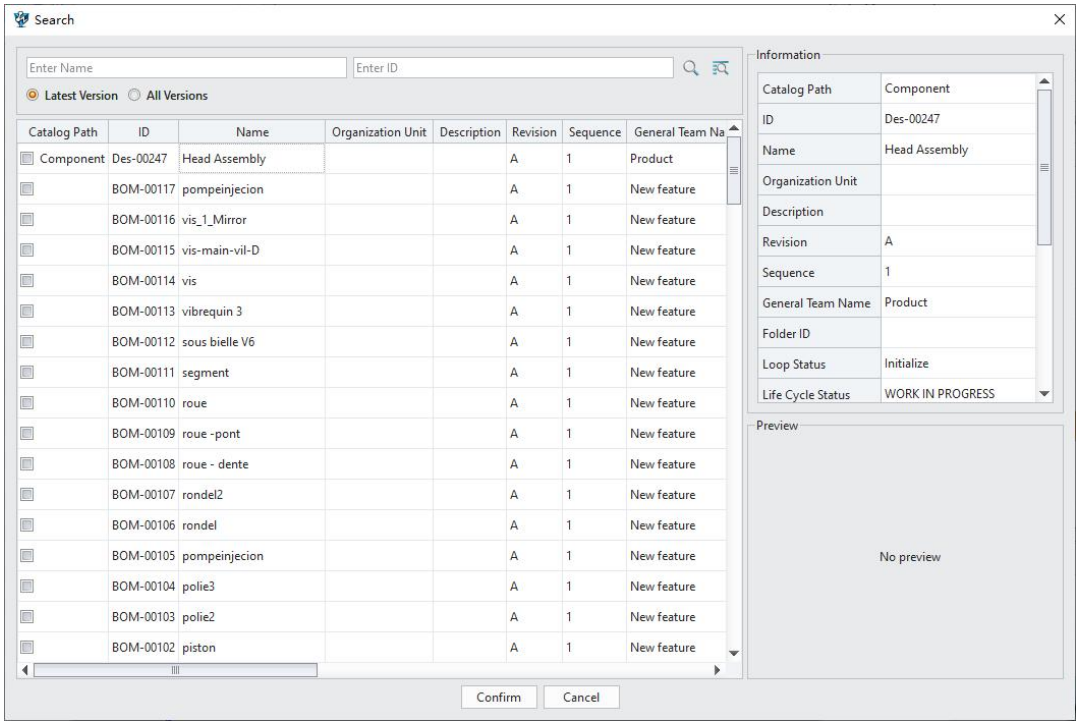




**【What users can do】**

✓ Add New Part: Create a new part and automatically associate it with the design object.



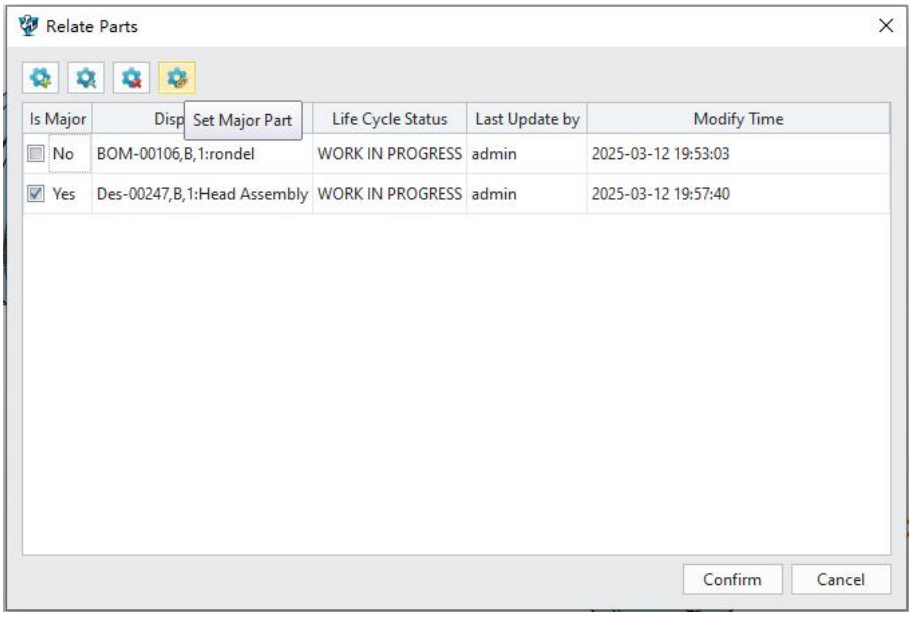

✓ Add Existing Part: Search for and link existing parts from the system to the design object.




- ✓ Remove Related Parts: Supports disassociating already linked part objects.
- ✓ Set Major Part: If multiple parts are related, users can designate a primary part object.

**【Notes】**

- Models or drawings created using the New File feature in ZW3D must first create a design object before Relate Part.
- Multiple part objects can be related, but only one major part object is allowed.

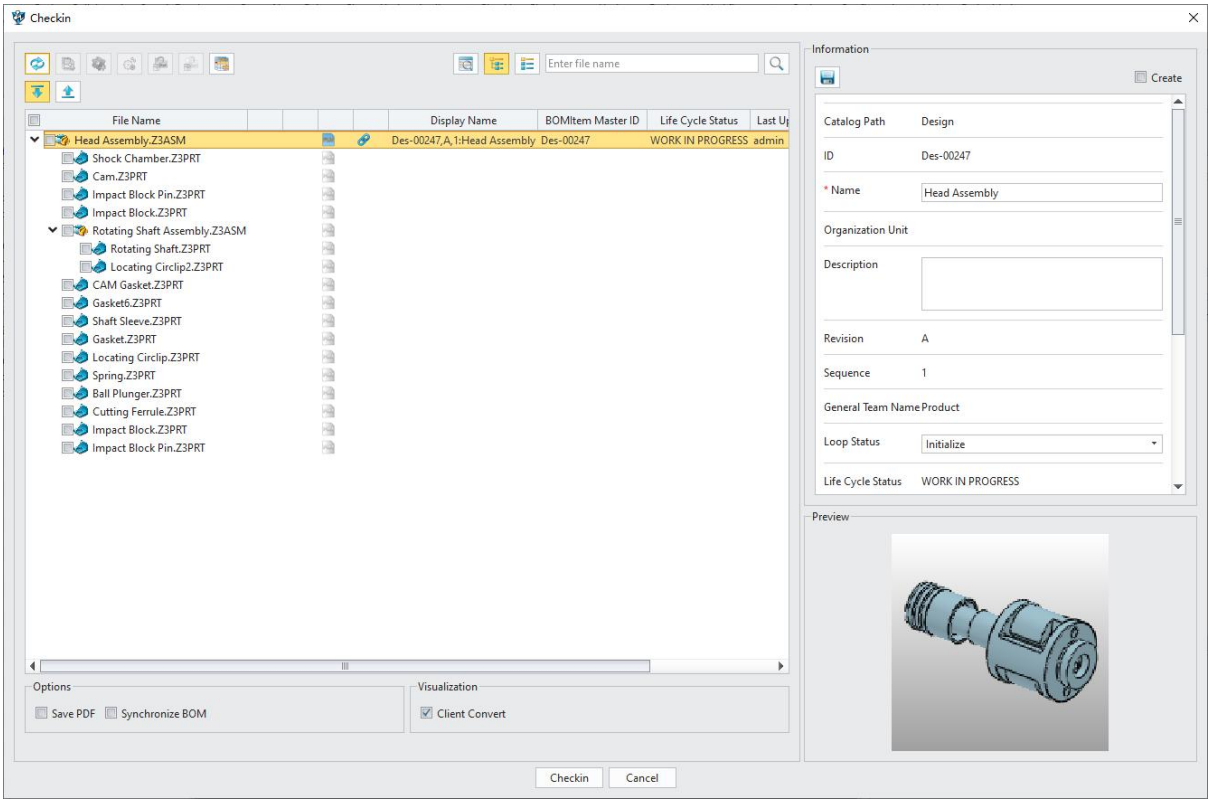



**【Where is it】**

ZW3D >> ZWTeammate >> **Relate Parts**

## 8.6 ★Check-in

The Check-in interface now supports saving the complete set of digital model files, including models, drawings, machining plans, external references, and sketches, to a specified path in the PDM system. Within the Check-in interface, users can also perform operations such as data creation, modification, Relate Parts, saving 2D PDF files, local visualization saving, BOM Compare, and BOM synchronization. This update primarily includes the following features:




## 8.6.1 Synchronize BOM

A new "Synchronize BOM" checkbox has been added to the Check-in interface, enabling the system to synchronize and update the BOM (Bill of materials) during the model Check-in process based on the latest assembly design structure.

Using the assembly structure and the primary part relationships of the design objects, the system will synchronously generate or update the BOM structure for the parts.

**【What users can do】**

✓ The BOM (Bill of materials) is automatically generated or updated with a single click based on the relationship between the assembly design structure and the main part object related with the design object, using the latest assembly design structure.

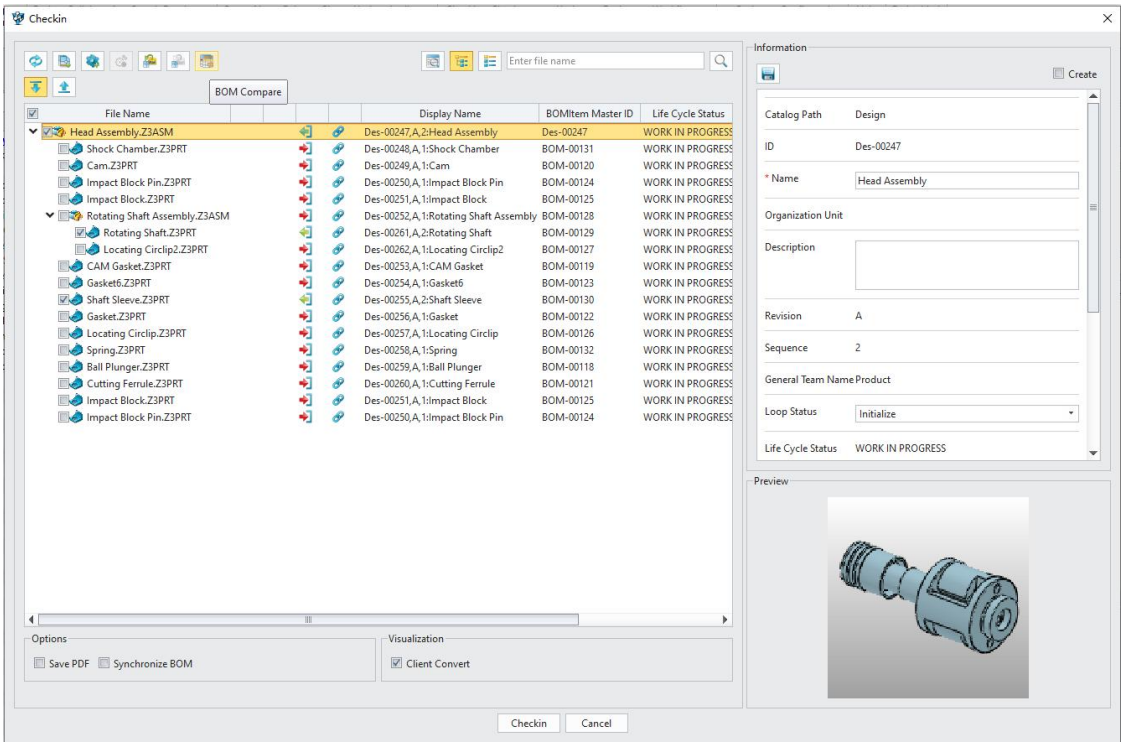
✓ Supports skeleton design scenarios: For models designed with skeletons, it supports generating an assembly structure with themaster layout, while excluding themaster layout during the generation of the part BOM structure.

【Where is it】

ZW3D >> ZWTeammate >> Checkin >> **Synchronize BOM**

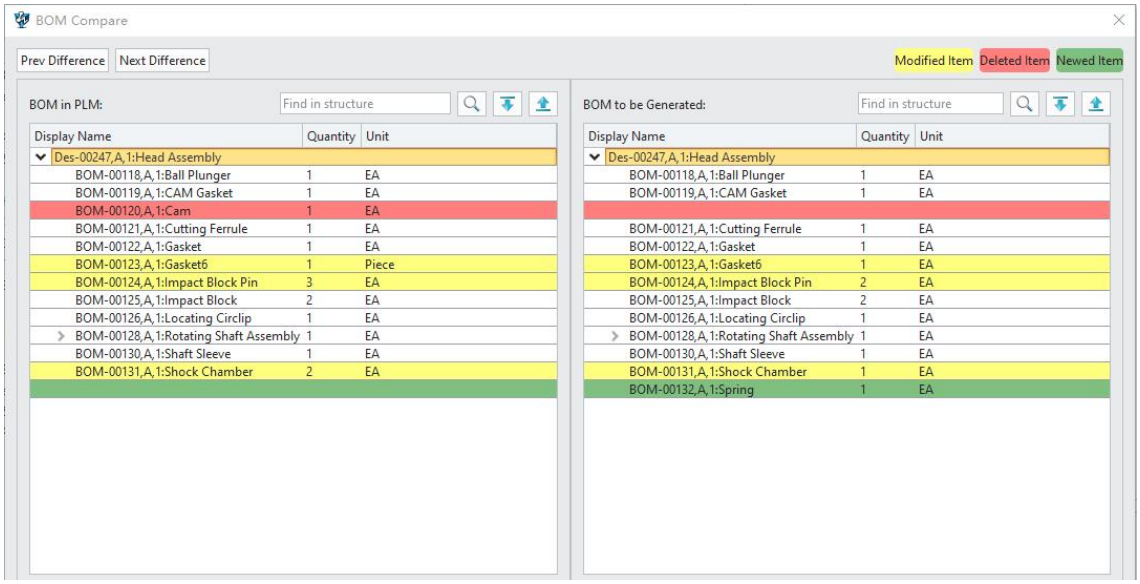
## 8.6.2 Add BOM Compare

A BOM Compare button is added to the Check-in interface, allowing for the comparison of the BOM (Bill of materials) to be generated with the existing BOM structure on the ZWTeammate side, highlighting any differences.

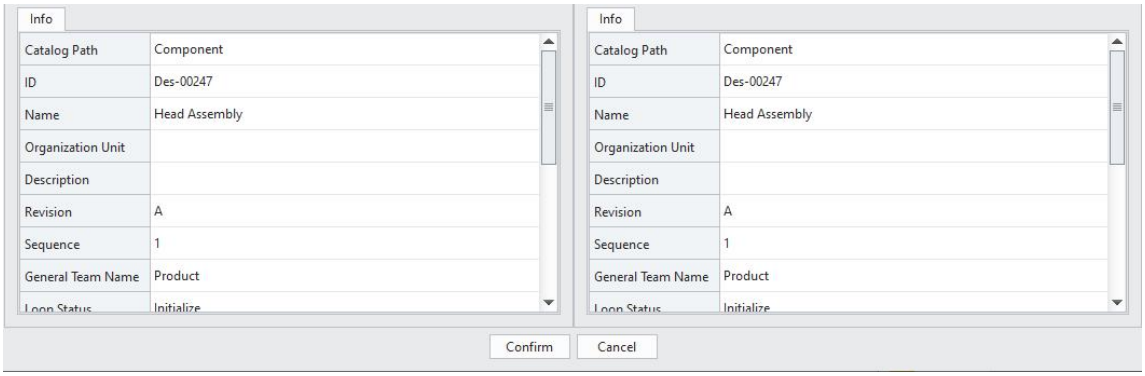



【What users can do】

✓ Compare the BOM structures before and after modifications, with updated content highlighted in different colors.




✓ Display basic information about the parts related with the part/assembly models.




**【Where is it】**

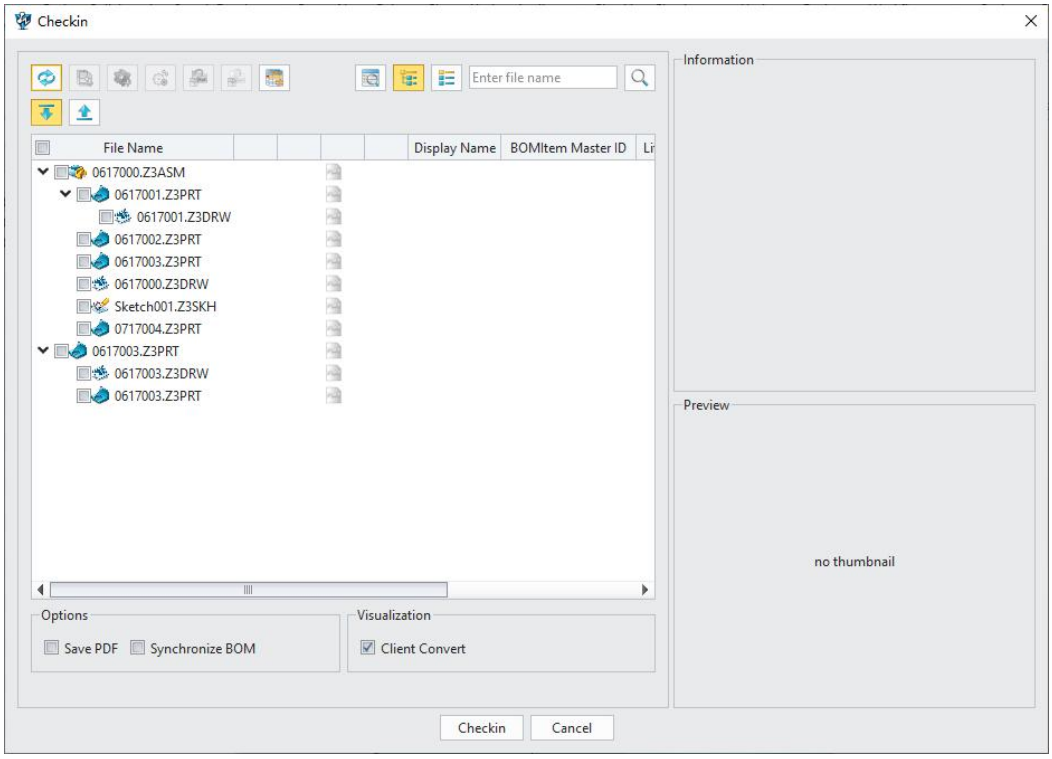
ZW3D >> ZWTeammate >> Check in >> **BOM Compare**

## 8.6.3 Check-in Derived Files

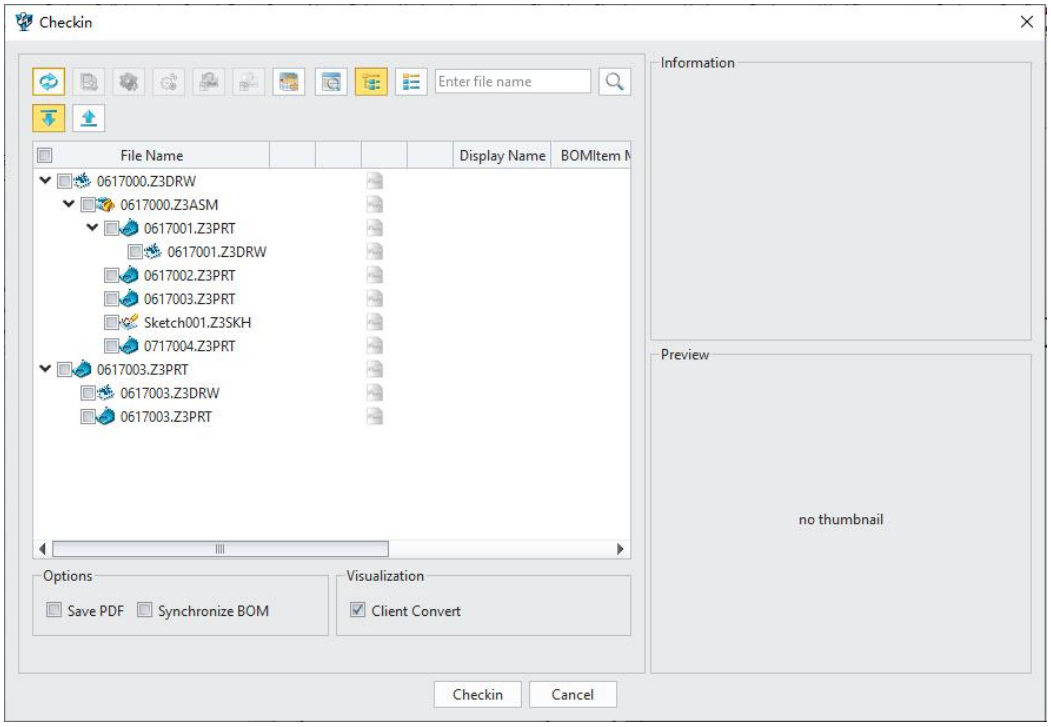
During the Check-in process, the system will also include derived files of the model, such as external parts, sketches, drawings, and cam plans, establishing reference relationships and checking them into the project collaboration area. This ensures the completeness and traceability of all critical design information.

**【What users can do】**

✓ When checking in part or assembly files, the system supports including related family tables, drawings, sketches, and external reference parts.




✓ When checking in drawings or cam plans, the system supports including related models, family tables, sketches, and external reference parts.




**【Where is it】**

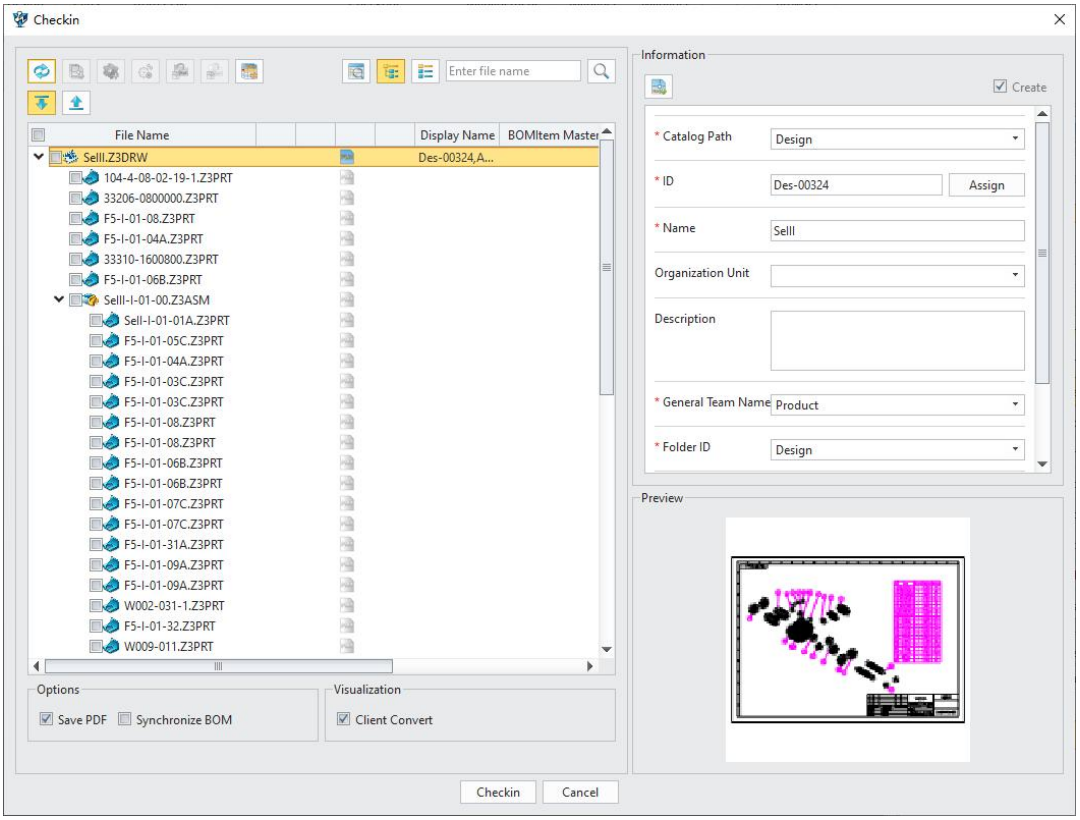
ZW3D >> ZWTeammate >> **Check-in**

## 8.6.4 Synchronous PDF Generation for Drawings

A new "Save PDF " checkbox has been added to the Check-in interface, allowing users to convert drawings into 2D PDF files and upload them synchronously during the model Check-in process.

**【What users can do】**

- ✓ Batch Conversion: Automatically convert and upload multiple drawings to PDF in bulk.




**【Notes】**

- Limitation: Only drawing files can be converted to PDF; 3D model files are not supported at this time.

**【Where is it】**

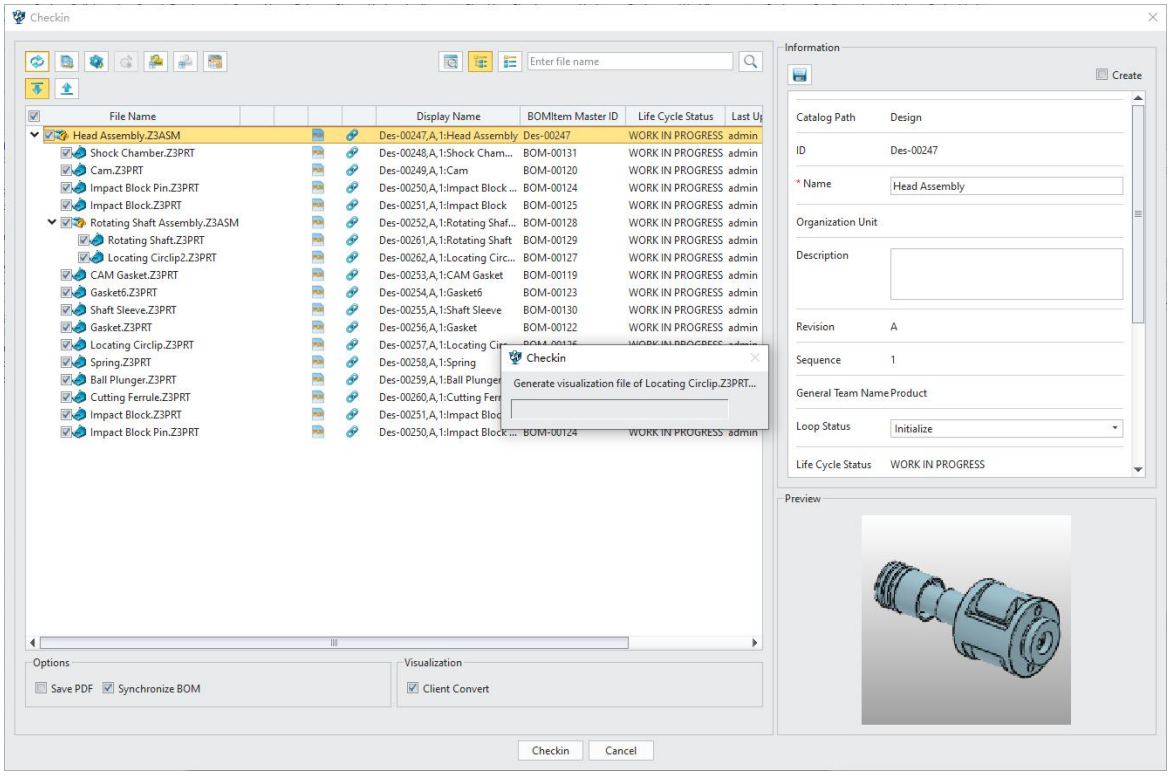
ZW3D >> ZWTeammate >> Check-in >> **Save PDF**

## 8.6.5 Local Visualization Conversion

The system supports converting models that are not yet checked in from the Checkin interface into HTML format for local visualization, which can then be uploaded along with the model files.

**【What users can do】**

✓ Process: When the "Client Convert" option is checked in the Check-in interface and the "Check-in" button is clicked, the system will convert the relevant unchecked-in models into HTML format and upload them to the PDM system along with the model files.




**【Notes】**

- Limitation: Family table objects are currently not supported for HTML visualization conversion.

**【Where is it】**

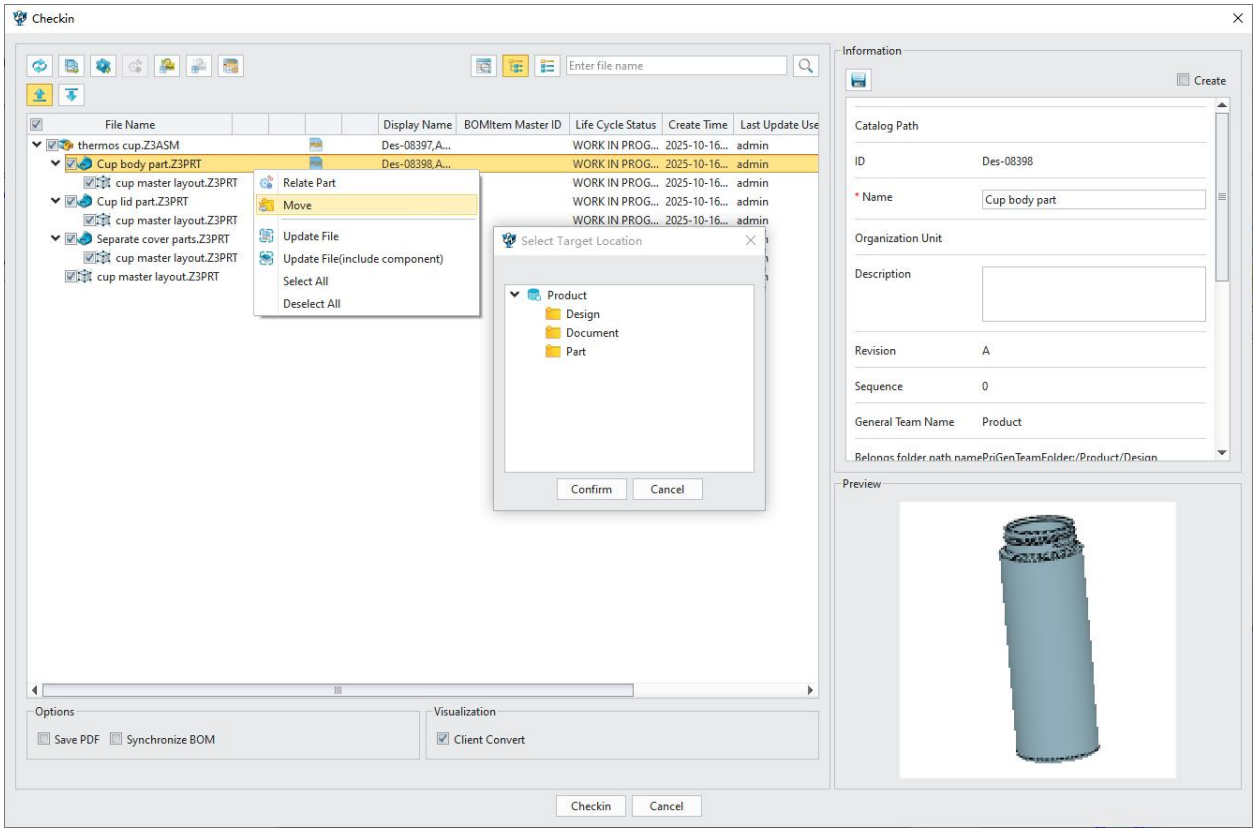
ZW3D >> ZWTeammate >> Check-in >> **Client Convert**

## 8.6.6 Add Modify Design Storage Location（ZW3D 2026 SP1）

When users check in designs, they will change their storage location in Teammate according to management needs.

**【What users can do】**

✓ When checking in, select the folder design that needs to be changed, perform the "move" operation, select the target folder, confirm and complete the changes.




**【Notes】**

- Only allowed to move within the same project collaboration area.

**【Where is it】**

ZW3D >> ZWTeammate >> Checkin >> **Move**

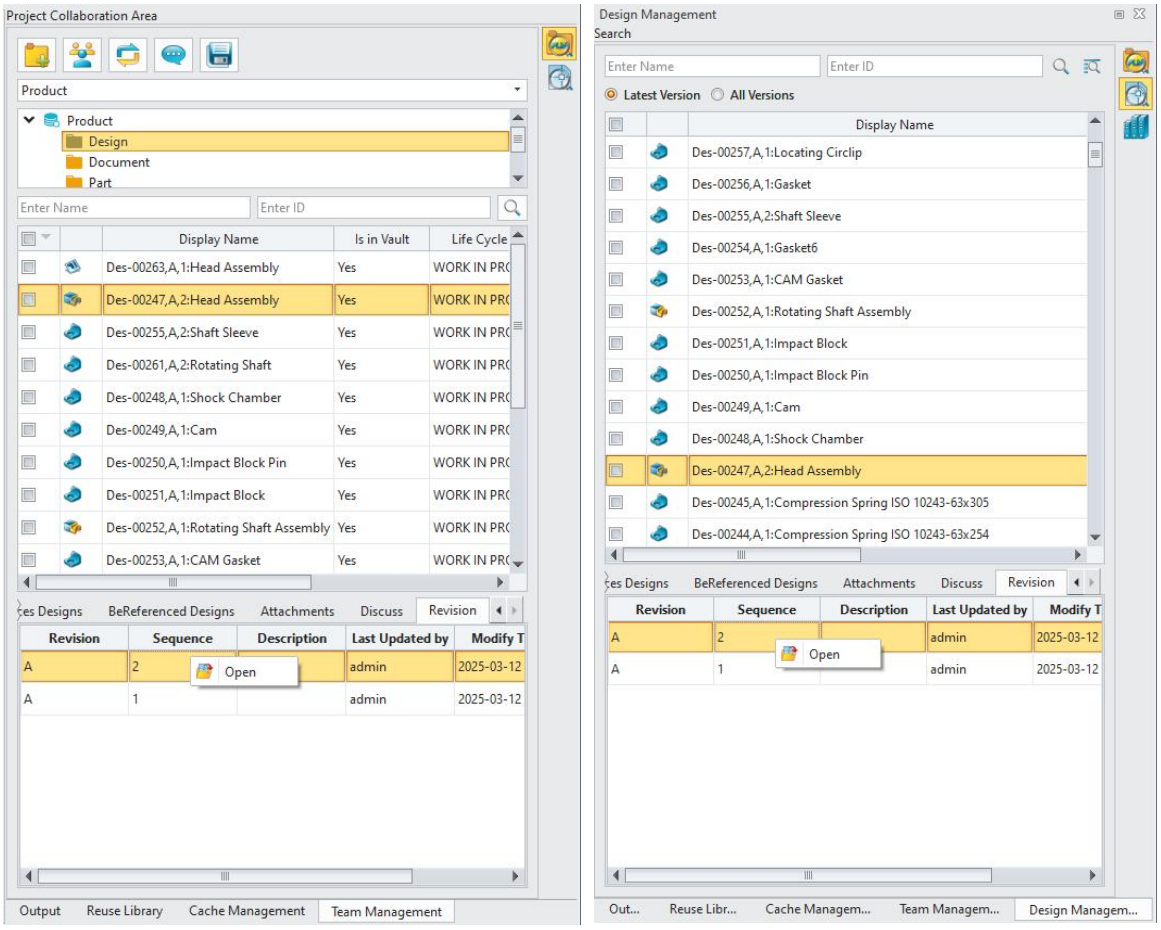
## 8.7 Version Management

### 8.7.1 View and Download Historical Minor Versions

The system supports viewing and downloading minor version files of historical drawings.

**【What users can do】**

✓ In the relevant search panel, right-click on an object or a specific version and select "Open" to download the complete set of digital model files to the local system and open the model in the ZW3D workspace.

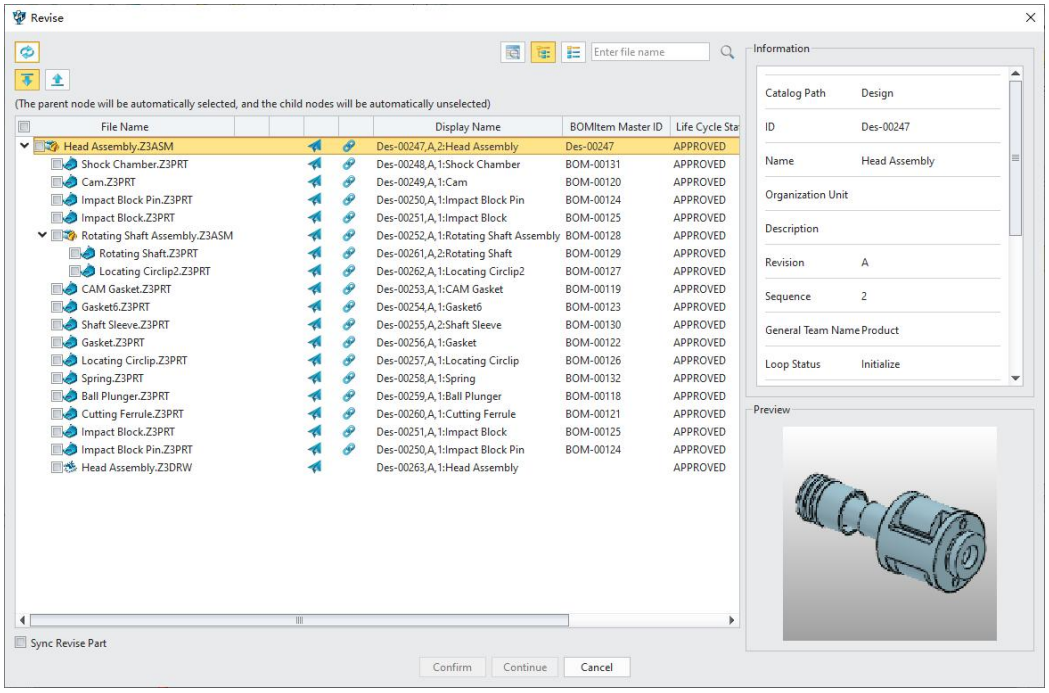



**【Where is it】**

ZW3D >> ZWTeammate >> Project Collaboration Area / Search for Design >> Object's "Revision" Tab >> **Right-click "Open"**

## 8.7.2 Optimized Revision

In the PDM system, if digital model files have been released, they no longer support Checkin or check-out operations. To make modifications, users must upgrade the version, a process facilitated by the revision feature.




**【What users can do】**

✓ Supports configuring relationship types to retain during revisions, including: structures, attachment relationships, part relationships, and folder relationships.

✓ Releases modification permissions for published files through the revision process.

✓ Allows one-click revision and version upgrades for selected or all model files.

✓ During revision, supports synchronously revising parts and re-establishing associations in the new version.

**【Where is it】**

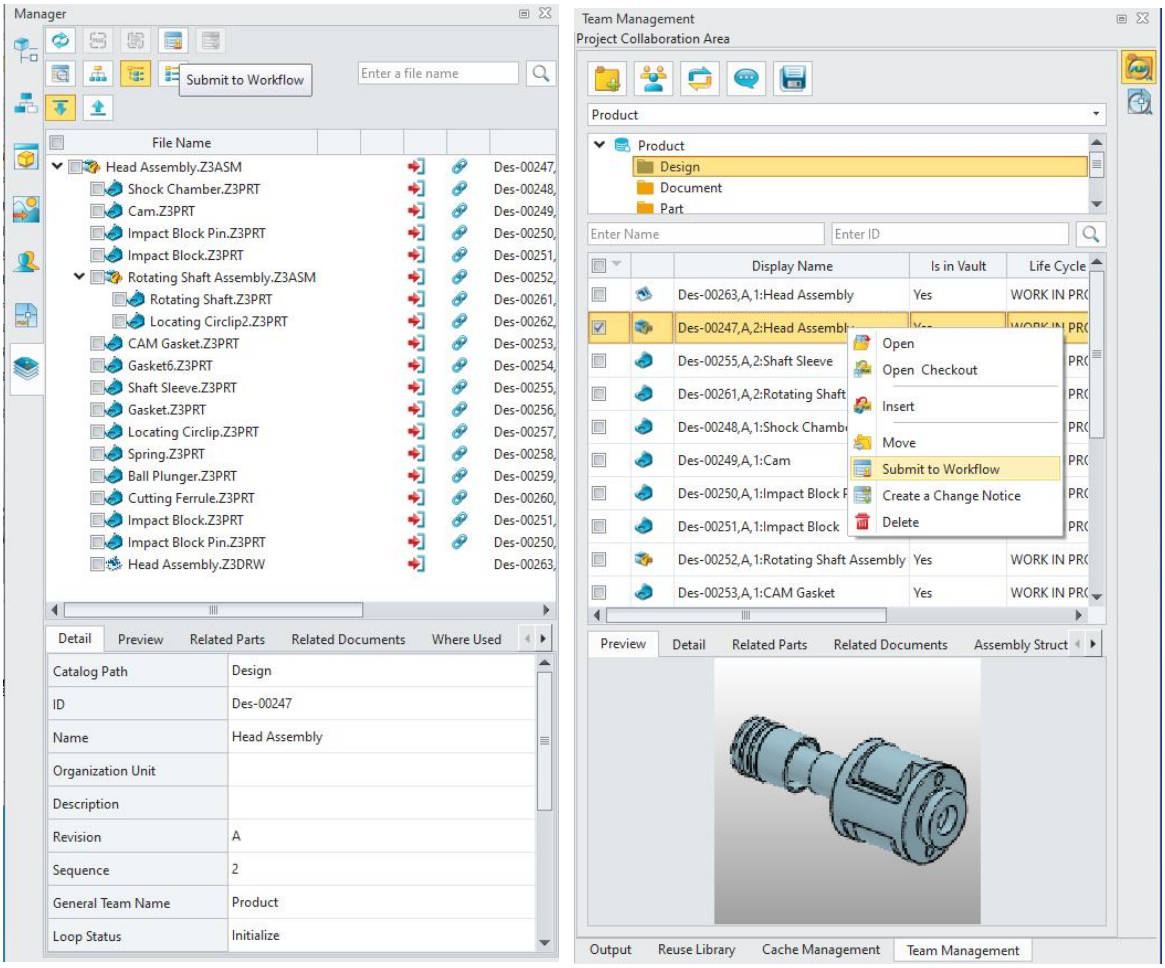
ZW3D >> ZWTeammate >> **Revise**

## 8.8 Process Management

Supports initiating processes and handling tasks in ZWTeammate (via the embedded browser). After process approval, the system performs silent attribute updates on ZW3D models.

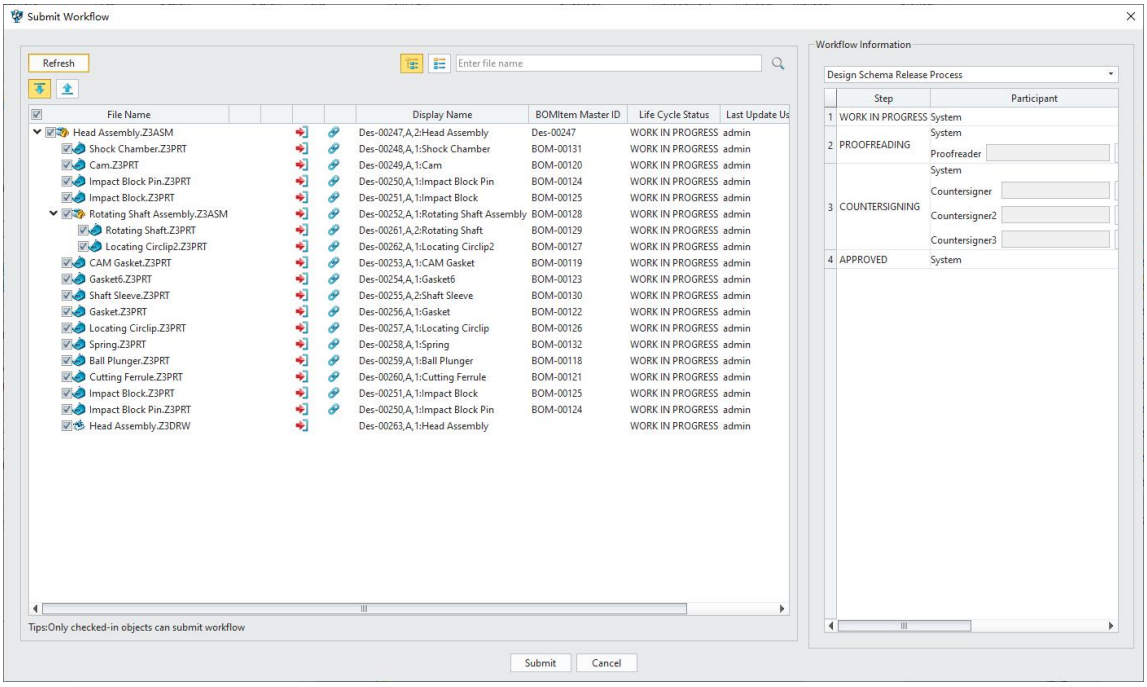
## 8.8.1 Submit to Workflow

Supports initiating approval workflows for design, part, and document objects from the Design Property, Design Structure, and various query panels.




**【What users can do】**

✓ Design Property / Design Structure Panel: Submit Workflow based on the complete structure of the current model.




✓ Project Collaboration Area Panel: Initiates approval workflows for a single selected object (the workspace includes design, part, and document objects, and due to differing workflows, multi-selection initiation is not supported).

✓ Search for Design / Search for Parts/Search for Document Panels: Submit Workflow in bulk for selected query results without traversing lower-level sub-components or relationships.

**【Where is it】**

ZW3D >> ZWTeammate >> **Design Property/Design Structure/Project Collaboration Area/Search for Design/Search for Parts/Search for Document Panels**

## 8.8.2 Silent Attribute Writing

Supports writing PDM attributes into assembly model properties, part model properties, and drawing properties without launching ZW3D.

**【What users can do】**

✓ Scope of Attribute Writing: Includes part attributes and design attributes, updated based on the attribute mapping and update direction configured in the settings file.

✓ Difference Comparison: Before writing, the system compares differences and only updates the file if changes are detected to prevent altering the file's MD5 checksum.

✓ Error Handling: If attribute writing fails, a failure event is logged in the "Error Message" section for review, allowing retries.

**【Notes】**

• Activation: Silent attribute writing is not enabled by default and can be activated based on project requirements.

**【Example】**

1) Add an event in the workflow (typically after the final approval node) to update the PDM attributes into the main file model properties of the approved design object.

2) After converting a part code from a temporary to an official number, update the "part Number" attribute in the model.

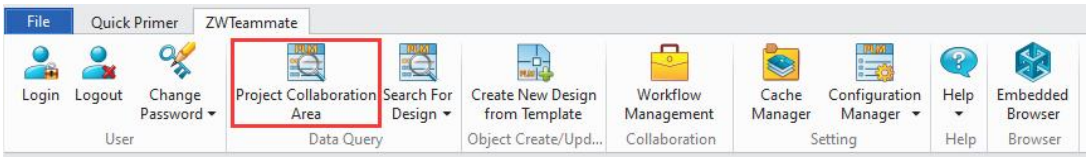
**【Where is it】**

ZW3D >> ZWTeammate >> **Workflow Management**

## 8.9 UI Adjustments and Usability Enhancements

### 8.9.1 Dedicated Project Collaboration Area Entry

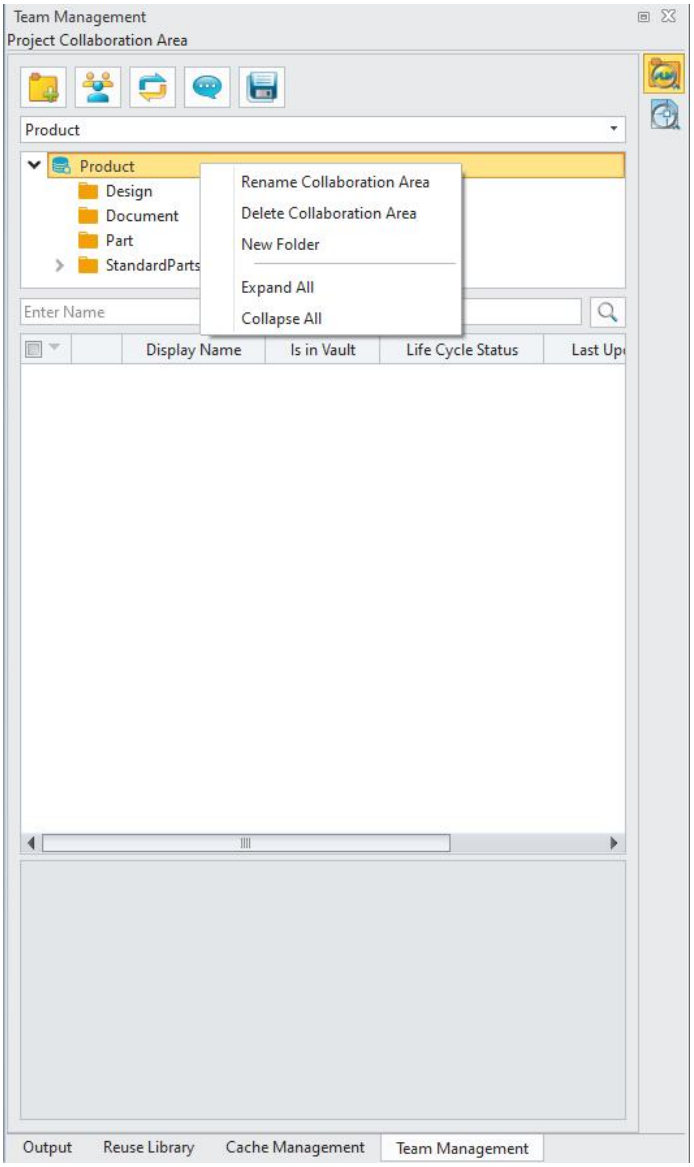
The Project Collaboration Area is built on group collaboration within a shared data repository, managing work-related data. Users can identify effective workspace configurations through experience, save them as templates, and reuse these templates to create new Project Collaboration Areas in the future.




**【What users can do】**

✓ Supports simultaneous management of design, part, and document data (combining queries for design, part, and document).

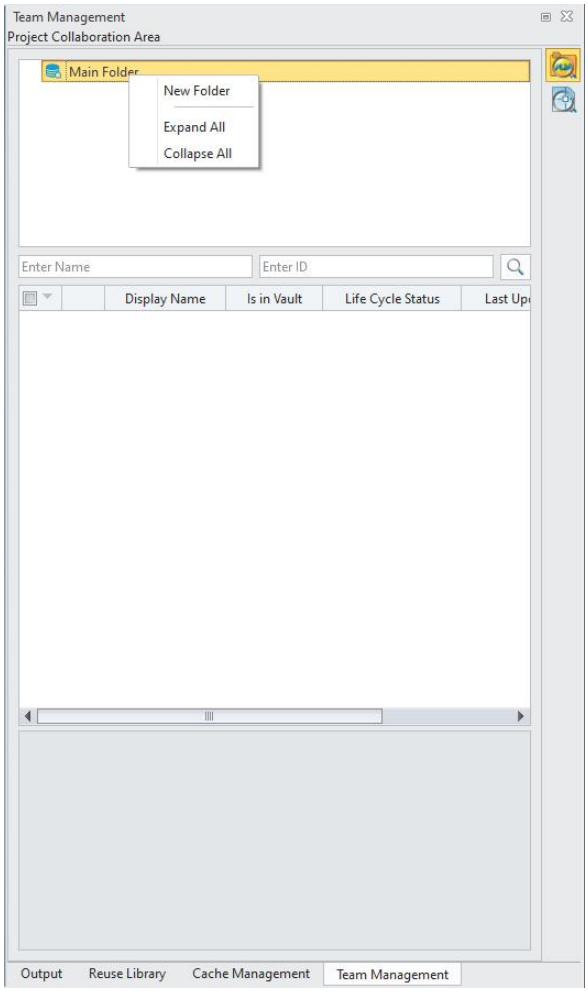
✓ Centralizes operations related to the Project Collaboration Area, such as creating new collaboration area, team management/role authorization, status transition, discussions, and saving as templates, within a unified interface.




**【Notes】**

- When users operate via a web browser, data creation must be synchronized between the web and integrated environments. In this case, the system uses a "Main Folder Only" configuration, where all data is created in the main folder.

- When users operate solely in the integrated environment, the Project Collaboration Area mode is enabled, allowing data to be managed according to the Project Collaboration Area structure.
- If the configuration file defines the integrated folder's working mode as "ProjectTeam" (Main Folder Only), the Project Collaboration Area panel will only display the main folder, and Project Collaboration Area-related settings will be disabled.



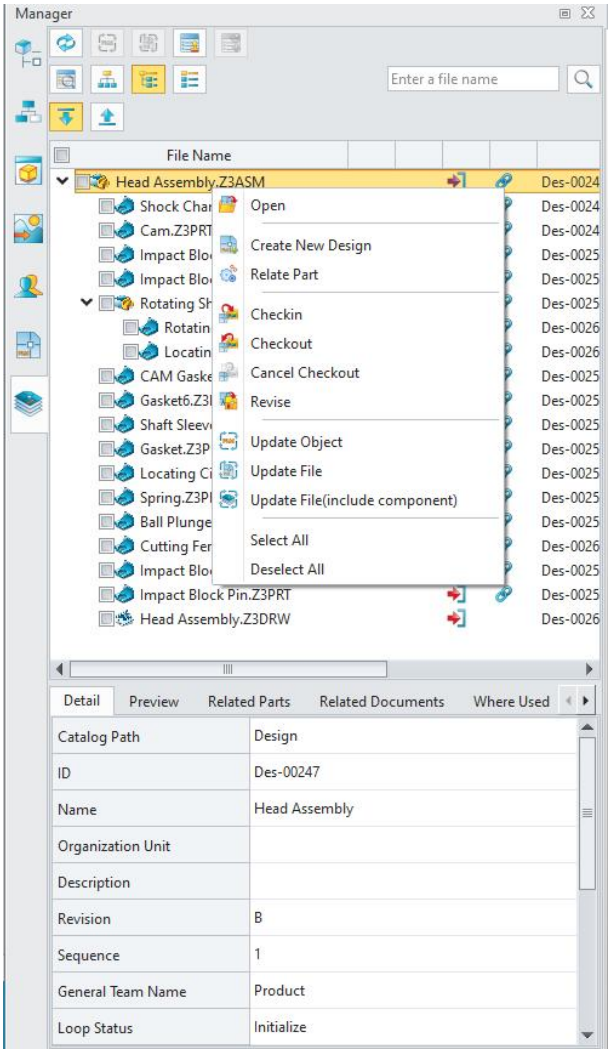

**【Where is it】**

ZW3D >> ZWTeammate >> **Project Collaboration Area**

## 8.9.2 Design Structure Panel

The manager area provides a comprehensive panel—Design Structure—for conveniently viewing

the data and status of current digital model files. In this panel, users can perform operations such as creating design objects, Relate Part, checking in, checking out, canceling check-out, revising, updating objects/files, updating files (including subcomponents), and opening files.




**【What users can do】**

✓ The panel supports creating design objects, Relate Part, checking in, checking out, canceling check-out, and revising, forming a complete version control and collaboration mechanism.

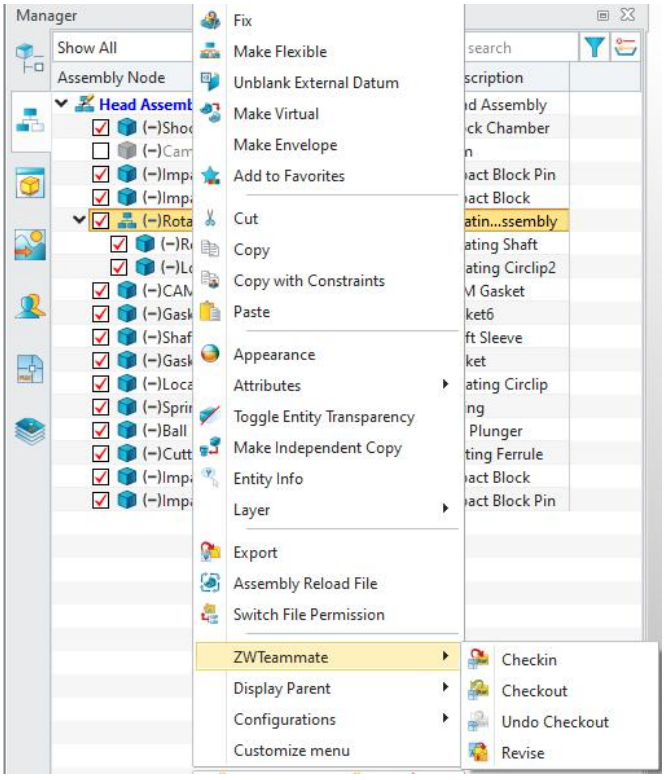
✓ Users can update design objects in the panel, including updating objects, files, and files (with subcomponents), ensuring that locally opened files are synchronized promptly after updates on the server.

【Where is it】

ZW3D >> Manager >> **Design Structure**

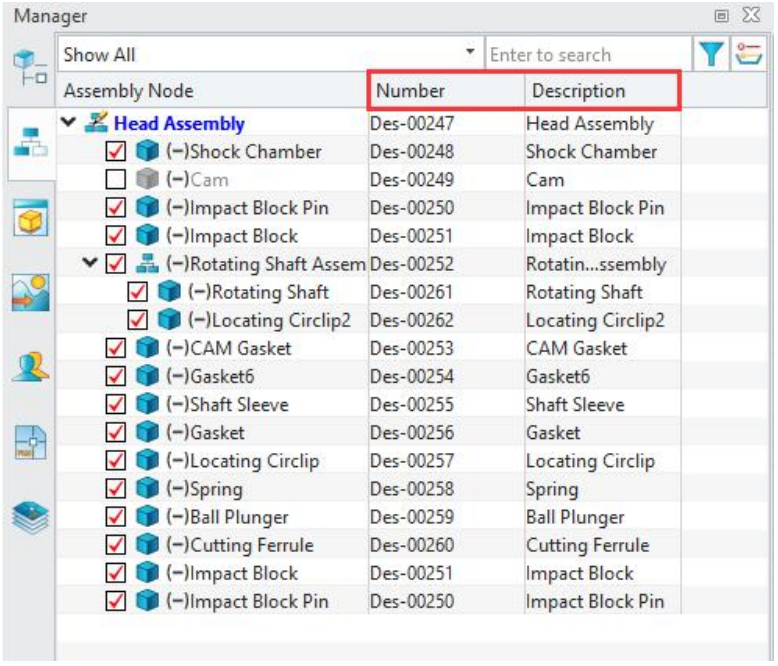
## 8.9.3 PDM Attribute Display and Right-Click Operations in the Assembly Tree

In the ZW3D "Assembly manager" panel, the assembly structure tree supports configuring the display of PDM-related content (e.g., object numbers, names, etc.). Right-click menus can be enabled to perform PDM operations (e.g., Check-in, Checkout).




【What users can do】

- ✓ Supports configuring the display of PDM-related content (e.g., object numbers, names, etc.).



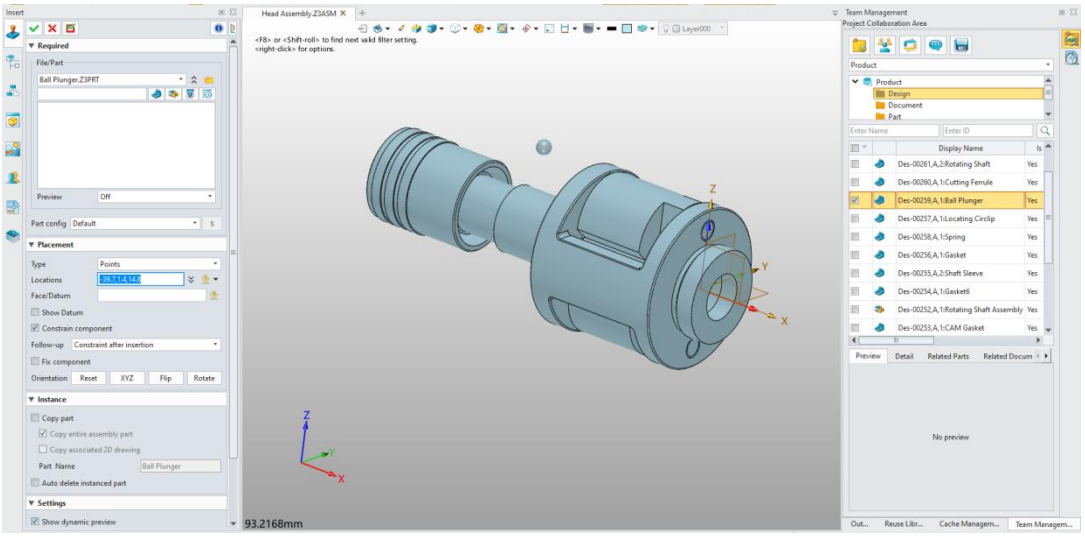

✓ Supports right-click execution of PDM-related operations (expandable).

**【Where is it】**

ZW3D >> Manager >> **Assembly Manager**

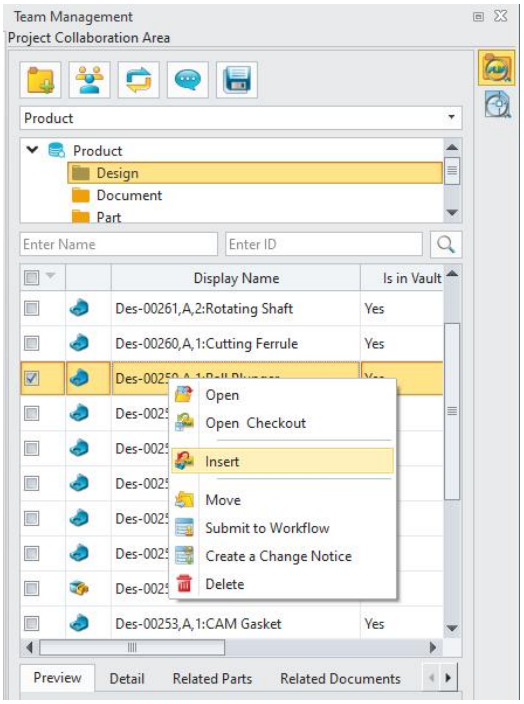
## 8.9.4 Right-Click Insert in Design Query List

Supports inserting models from the query results list into the currently open assembly in the design query interface.



**【What users can do】**

✓ A right-click "Insert" option has been added to the design query results list, allowing users to insert part or assembly models into the current assembly.




**【Notes】**

- Only part or assembly model objects in the query results list can be inserted.
- The target assembly must be in an unchecked-in state.

【Where is it】

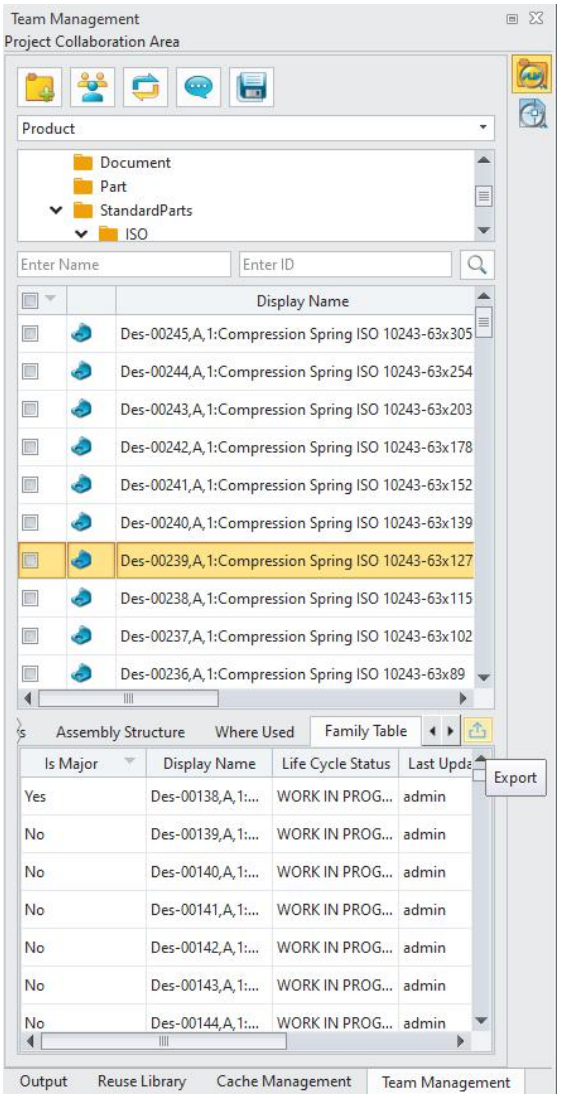
ZW3D >> ZWTeammate >> **Project Collaboration Area/Search for Design >> Right-click "Insert"**

## 8.9.5 Family Table Export in Design Query Interface

The design query interface now supports exporting family tables for models with family table relationships.

**【What users can do】**

✓ An "Export" button has been added to the "Family Table" tab in the design query interface, allowing users to export the entire family table to Excel.

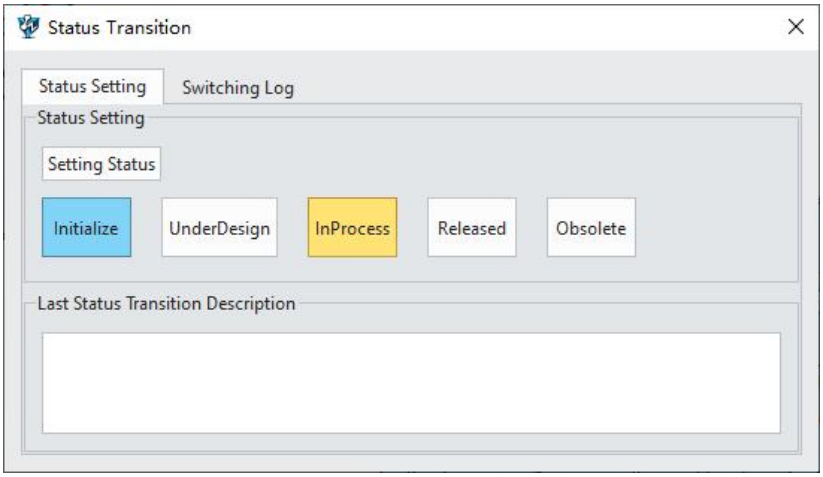


**【Where is it】**

ZW3D >> ZWTeammate >> Project Collaboration Area/Search for Design >> "Family Table" Tab >> **Export**

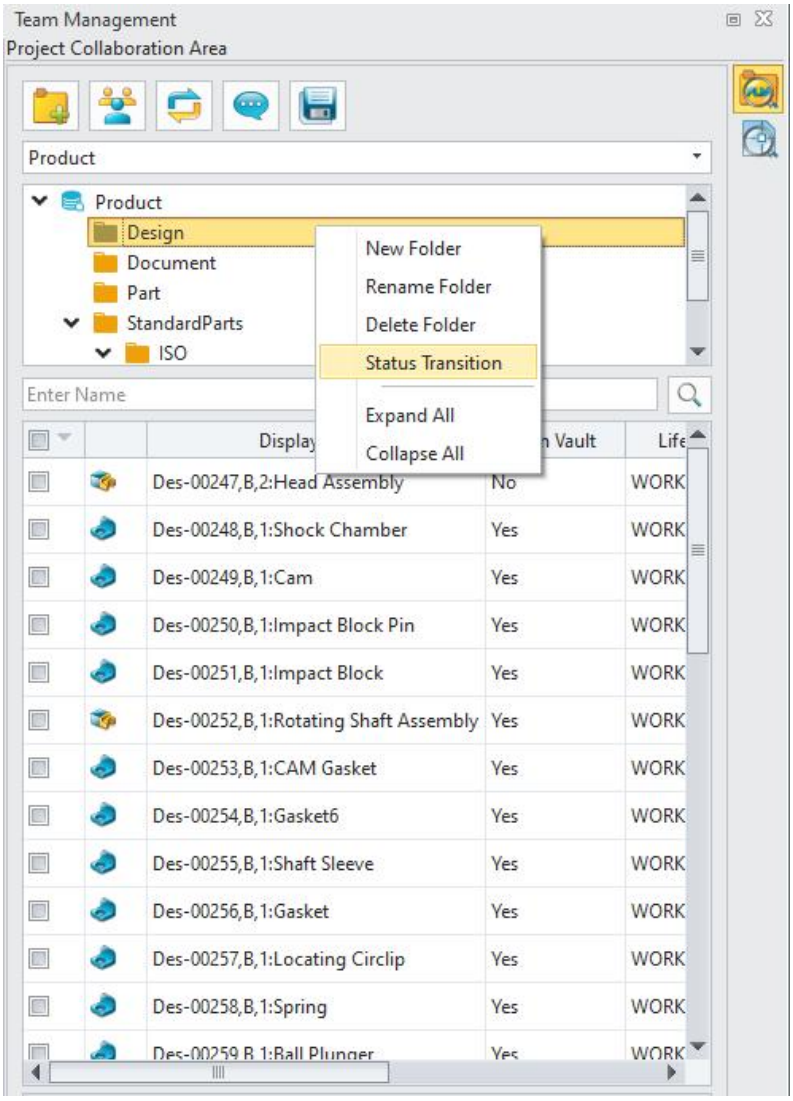
## 8.9.6 Folder-Level Status Transitioning

Team leaders (supervisors) can switch the iteration status of data within the entire workspace or a specific folder at any time, with related comments recorded during the switch.



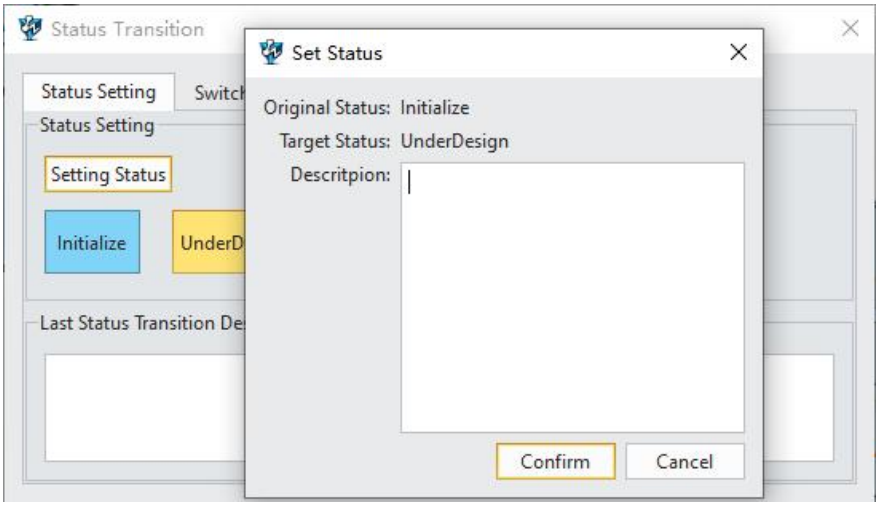

**【What users can do】**

✓ A right-click "Status Transition" option has been added to folders in the project workspace, similar to workspace-level Status Transitioning but specific to the data within the current folder.




**【Notes】**

- The Status Transitioning feature is only applicable to the Project Collaboration Area and does not apply to the main folder.
- The folder-level Status Transitioning in the Project Collaboration Area is simpler than the workspace-level Status Transitioning, supporting only updates to the iteration status of data within the folder.




**【Where is it】**

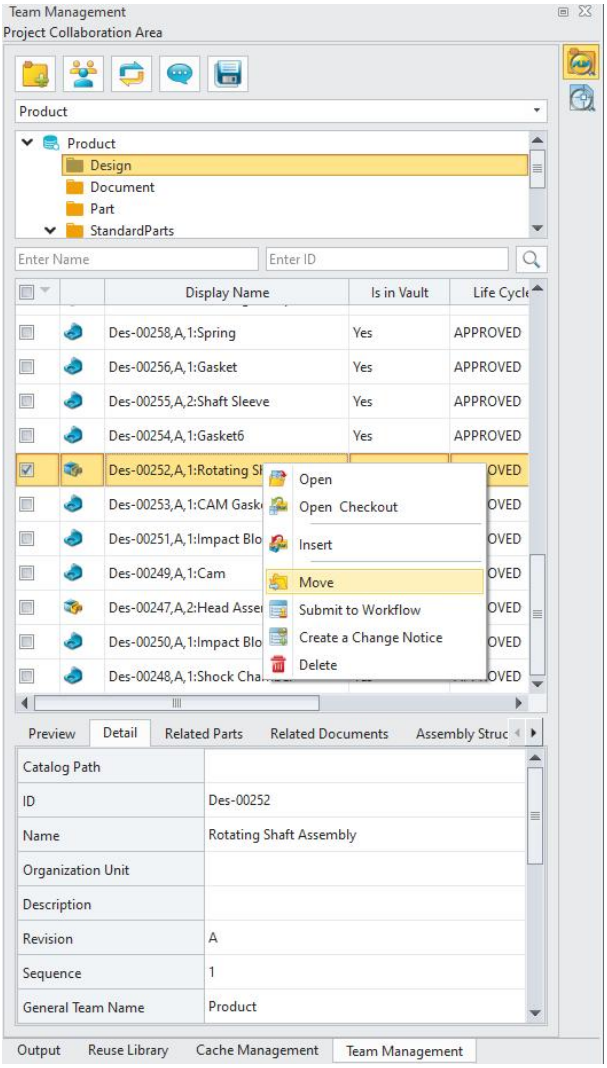
ZW3D >> ZWTeammate >> Project Collaboration Area Panel >> **Right-click "Status Transition" on Project Collaboration Area Folder**

## 8.9.7 Object Folder-Level Move Function

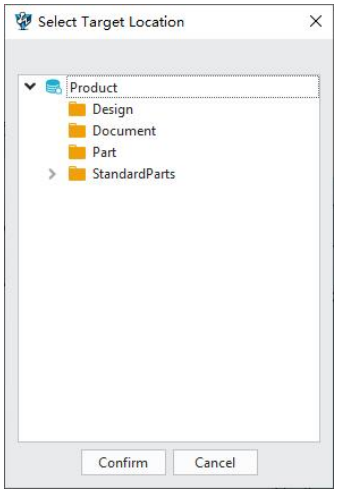
Objects within the same Project Collaboration Area can now be moved between different folders in the workspace.

**【What users can do】**

✓ A "Move" option has been added to the right-click menu for objects in the Project Collaboration Area query results list, enabling movement between folders within the same workspace.




✓ After clicking "Move", users can select the target folder for the move.



**【Where is it】**

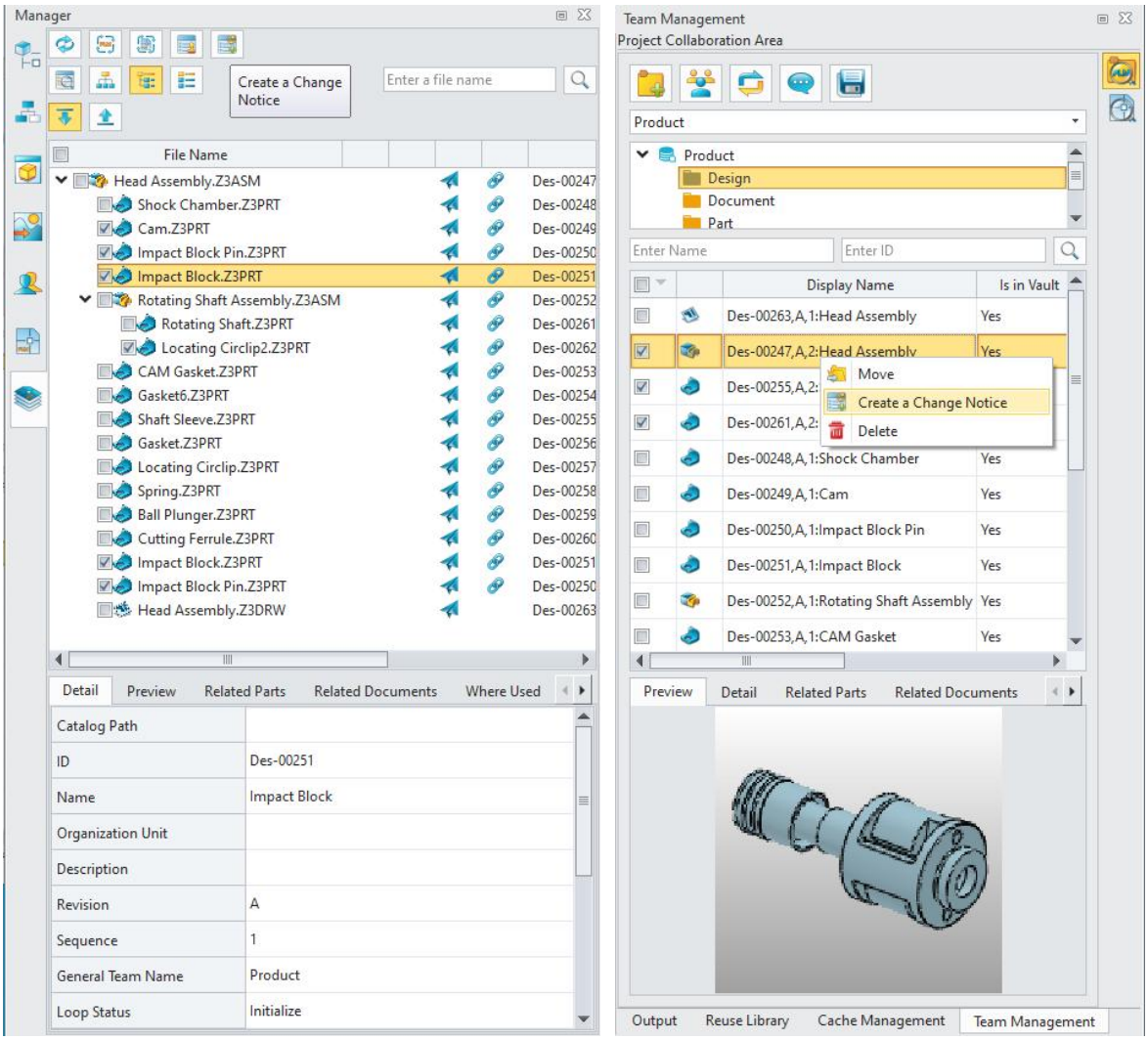
ZW3D >> ZWTeammate >> Project Collaboration Area/Search for Design/Search for Parts/Search for Document Panels >> **Right-click "Move"**

## 8.9.8 Support for Initiating ECN

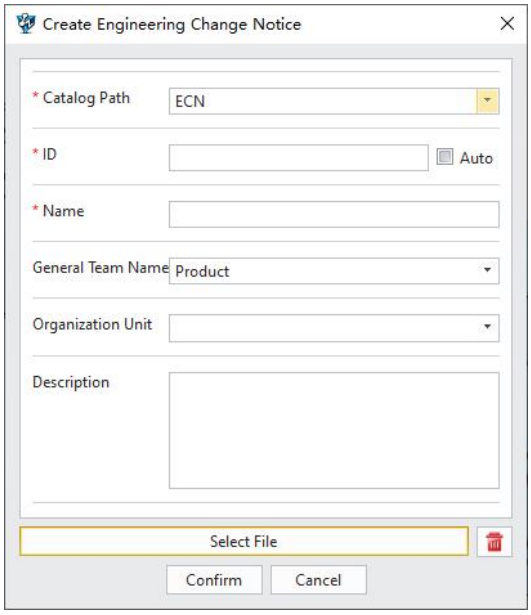
Supports creating and querying change notifications in ZWTeammate (via the embedded browser).

**【What users can do】**

✓ Supports initiating change notifications for design, part, and document objects from the design property, design structure, and various query panels.



✓ Select the objects to be changed, click the "Create a Change Notice" button, maintain the change notification attributes and upload files in the "Create Engineering Change Notice" interface, and click "Confirm" to create the change notification.




**【Notes】**

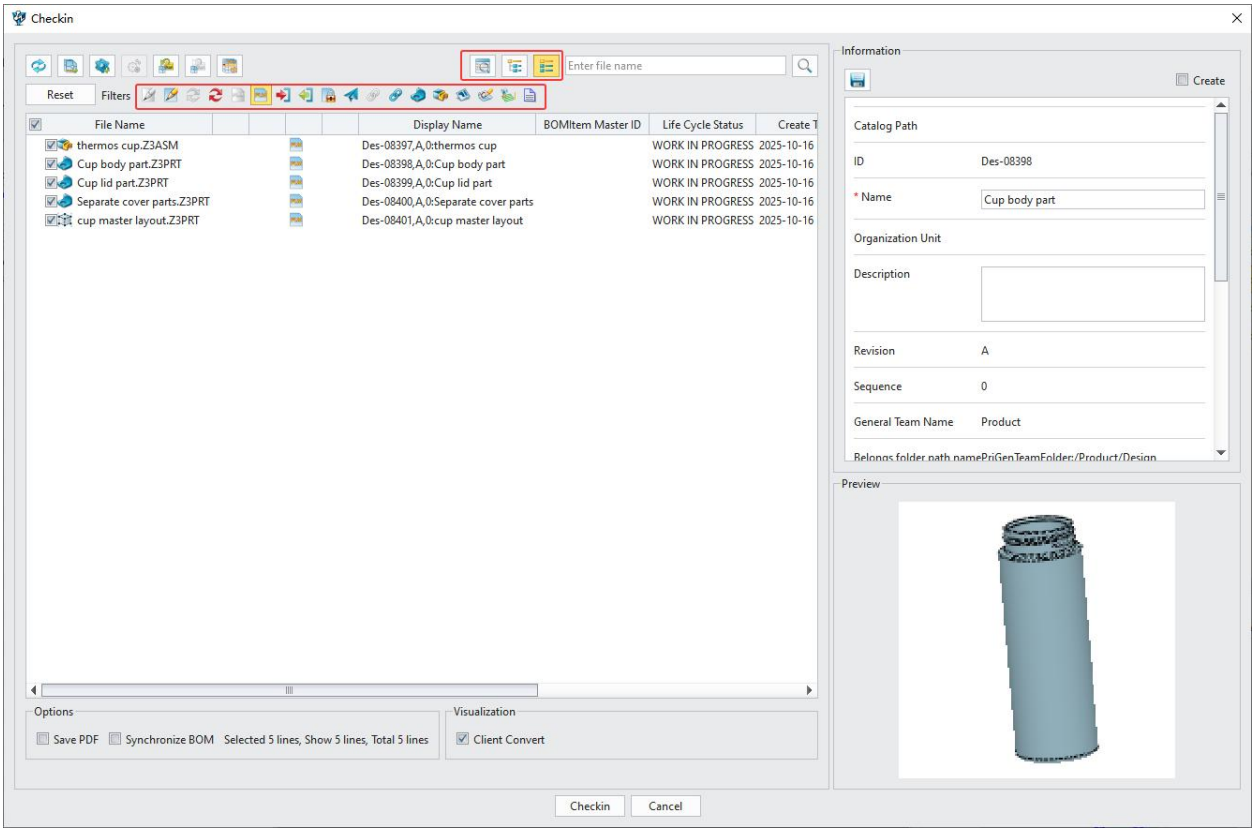
• The selected objects (multiple allowed) will be designated as the "Change Objects" in the change notification.

**【Where is it】**

ZW3D >> ZWTeammate >> **Design Property/Design Structure/Project Collaboration Area/Search for Design/Search for Parts/Search for Document Panels**

## 8.9.9 Default View and Filter Options（ZW3D 2026 SP1）

Support remembering the selected data view and filter items on pages such as Checkin, Checkout, Revise, and Design Structure. After reopening the page, display the data according to the data view and filter items.




**【Where is it】**

ZW3D >> Teammate >> **Checkin**

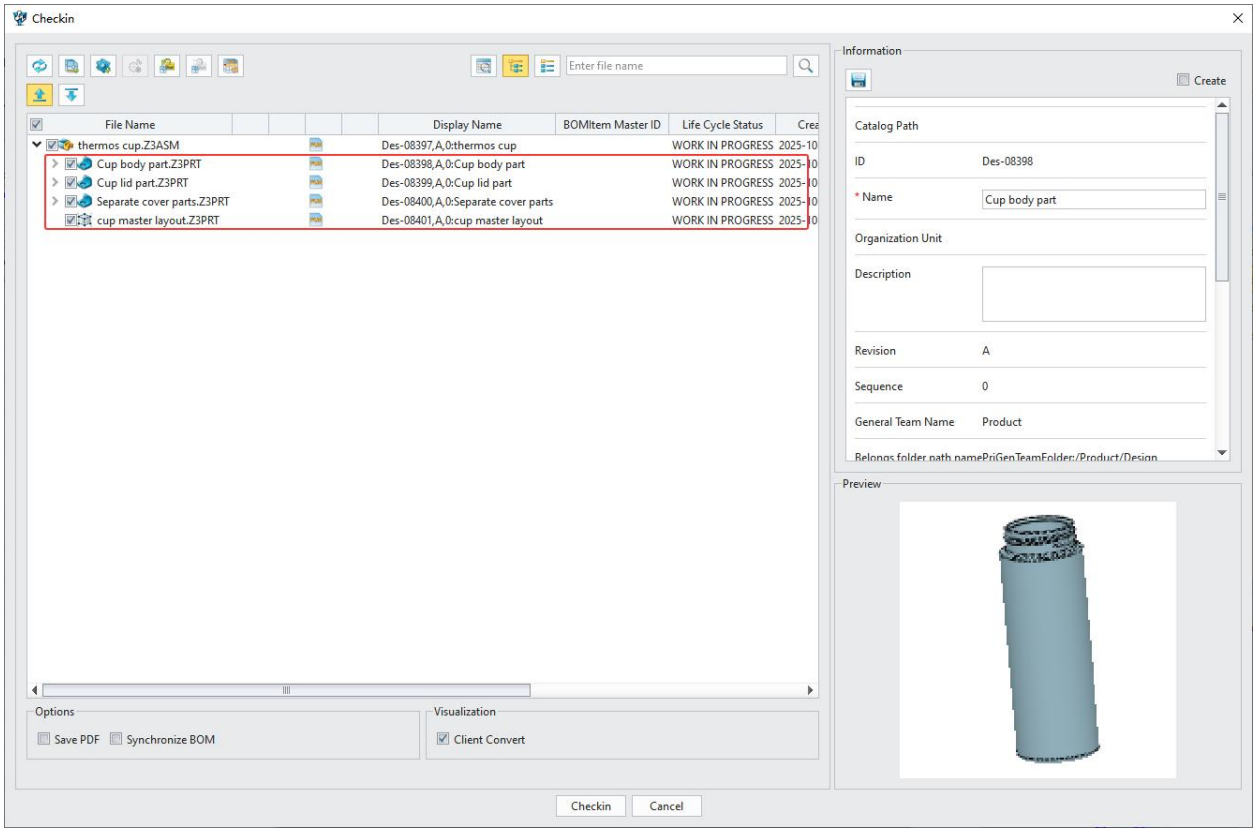
ZW3D >> Teammate >> **Checkout**

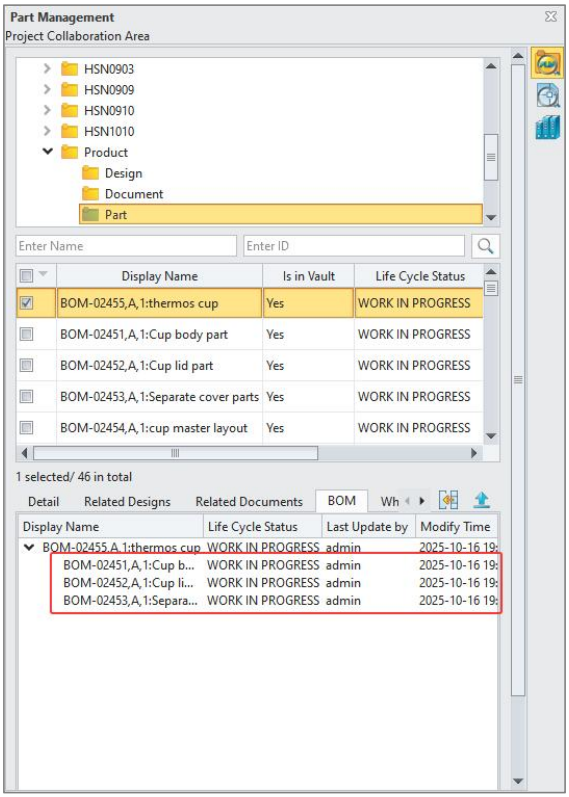
ZW3D >> Teammate >> **Revise**

ZW3D >> Manager >> **Design Structure**

## 8.9.10 Default Unfolding Hierarchy（ZW3D 2026 SP1）

BOM and assembly structure are expanded to the first level by default.


Checkin
Enter file name
File Name
Display Name
BOMItem Master ID
Life Cycle Status
thermos cup.Z3ASM
Des-08397,A,0:thermos cup
WORK IN PROGRESS
Cup body part.Z3PRT
Des-08398,A,0:Cup body part
Cup lid part.Z3PRT
Des-08399,A,0:Cup lid part
Separate cover parts.Z3PRT
Des-08400,A,0:Separate cover parts
cup master layout.Z3PRT
Des-08401,A,0:cup master layout
Information
Create
Catalog Path
ID
Des-08398
* Name
Cup body part
Organization Unit
Description
Revision
A
Sequence
0
General Team Name
Product
Preview
Options
Save PDF
Synchronize BOM
Visualization
Client Convert
Checkin
Cancel


Part Management
Project Collaboration Area
HSN0903
HSN0909
HSN0910
HSN1010
Product
Design
Document
Part
Enter Name
Enter ID
Display Name
Is in Vault
Life Cycle Status
BOM-02455,A,1:thermos cup
Yes
WORK IN PROGRESS
BOM-02451,A,1:Cup body part
BOM-02452,A,1:Cup lid part
BOM-02453,A,1:Separate cover parts
BOM-02454,A,1:cup master layout
1 selected/ 46 in total
Detail
Related Designs
Related Documents
BOM
Last Update by
Modify Time
admin
2025-10-16 19:

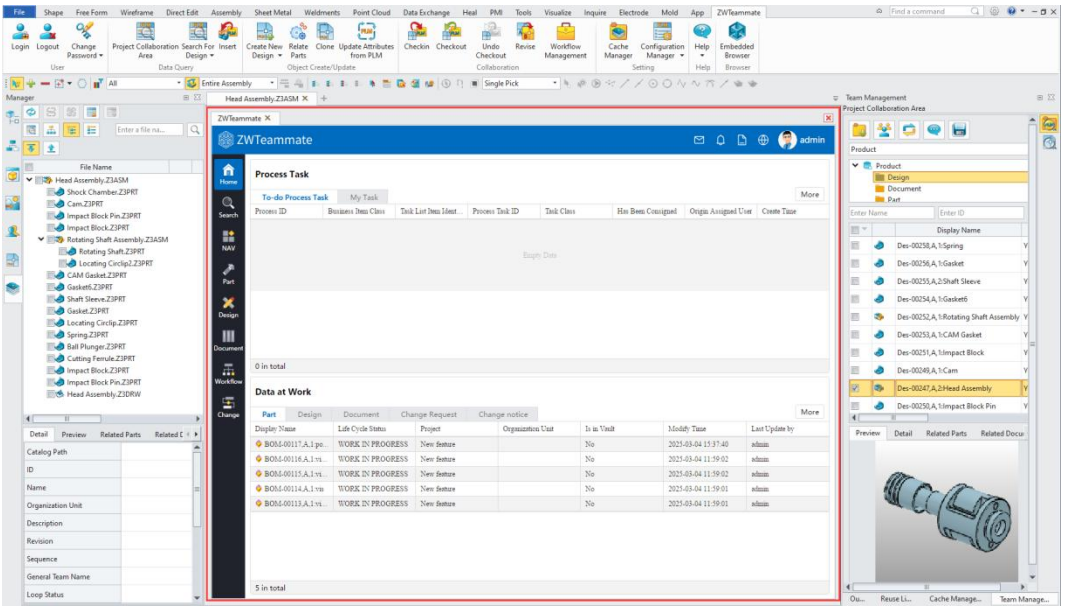
【Where is it】

ZW3D >> ZWTeammate >> **Checkin**

ZW3D >> Search For Parts >> **BOM tab**

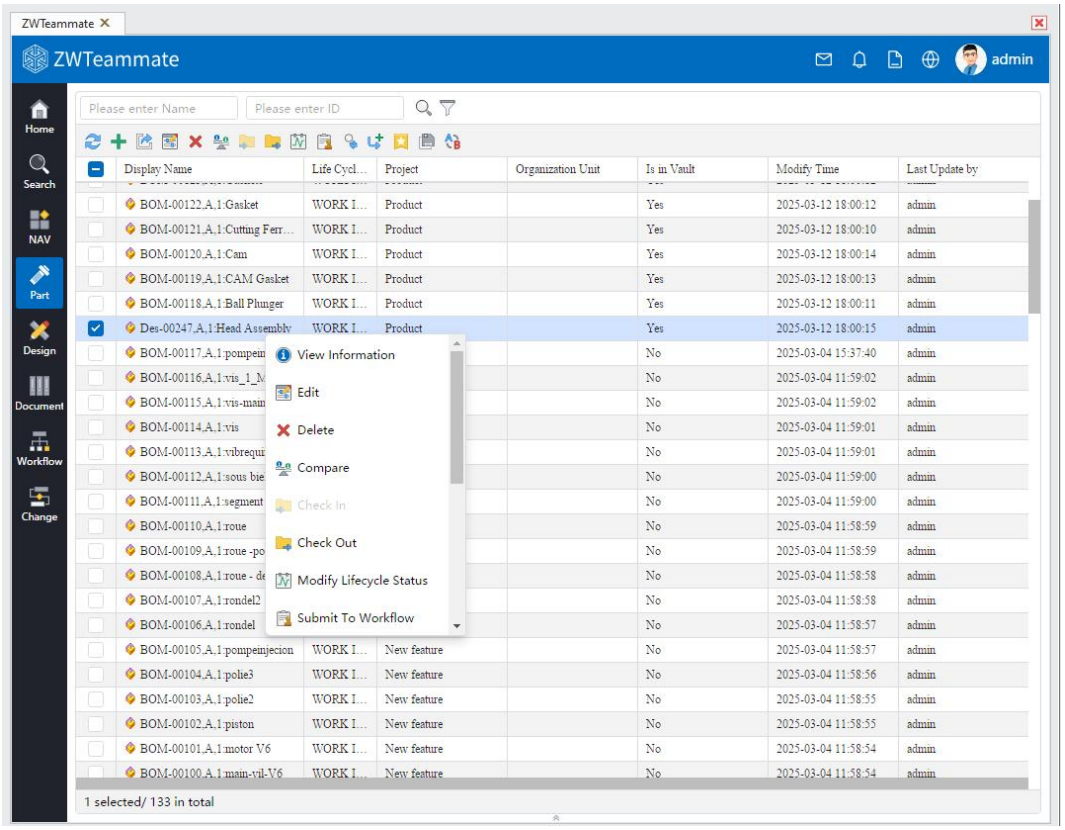
## 8.10 ★Embedded Browser

To enhance user efficiency in managing design processes, the new version integrates the ZWTeammate web client directly into ZW3D. This allows users to conveniently manage parts, design objects, documents, workflows, and engineering change management.



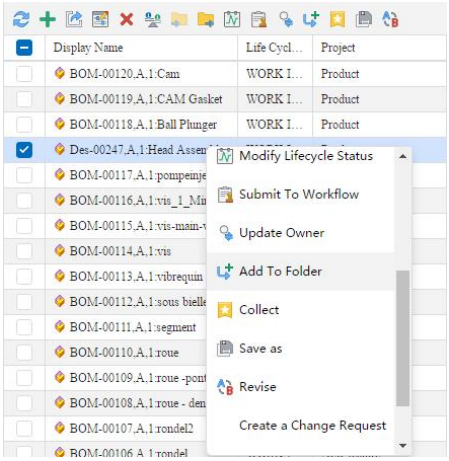

### 8.10.1 Part Management

The part management module supports maintaining parts and related information such as BOM (Bill of materials).

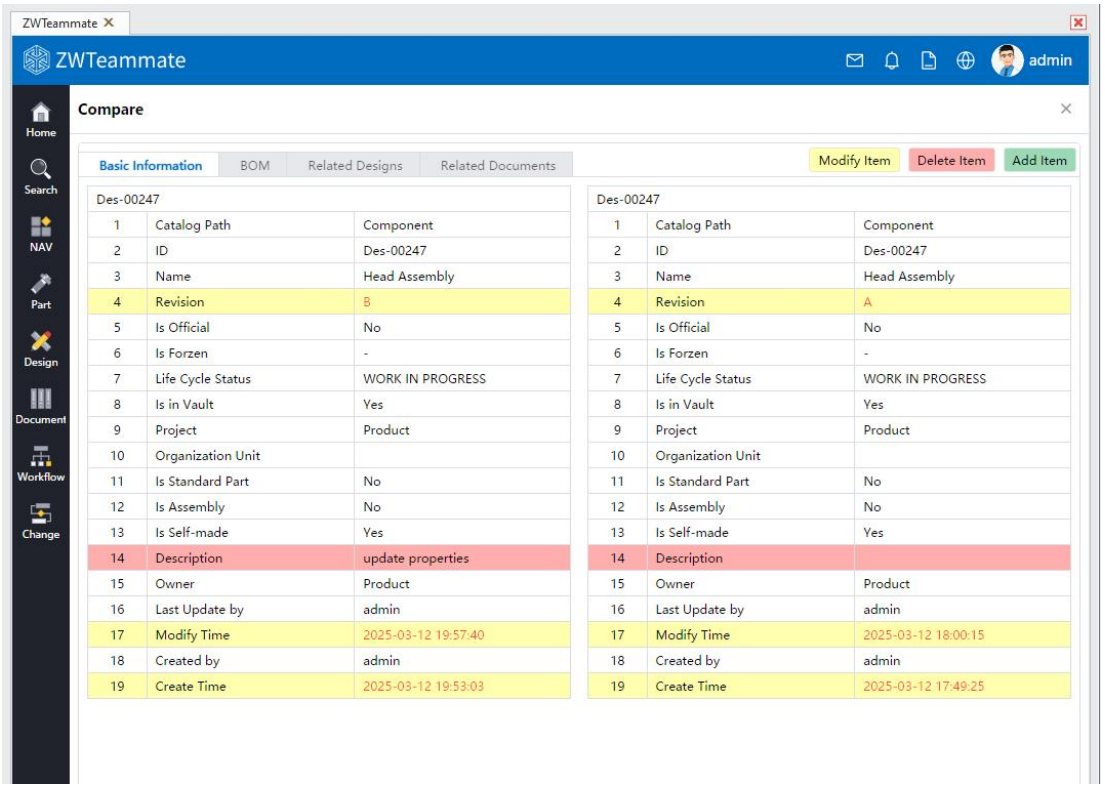



**【What users can do】**

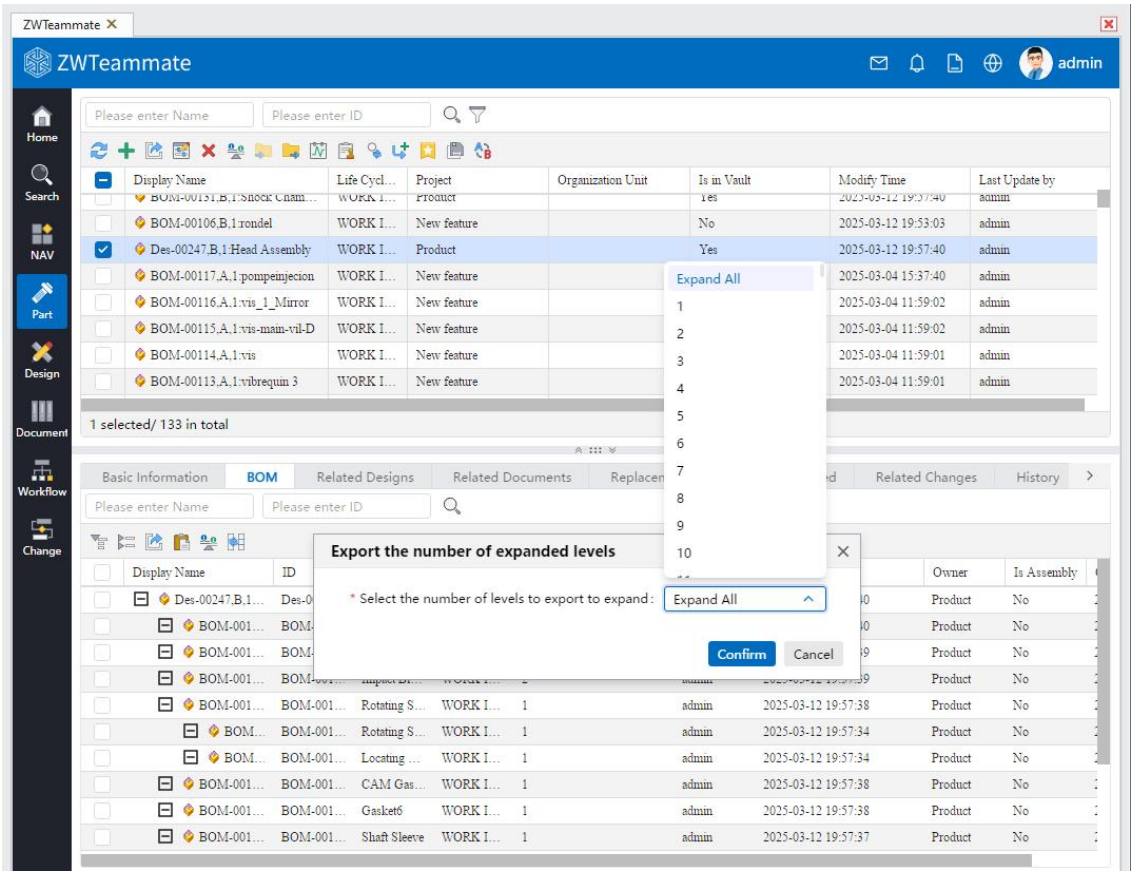
- ✓ Supports creating, editing, and deleting parts in the web interface.
- ✓ Supports modifying lifecycle status, revising, and initiating approvals for part objects.




- ✓ Supports comparing different versions of parts and highlighting differences between versions.




✓ Supports exporting BOM (Bill of materials) to Excel, with customizable expansion levels.



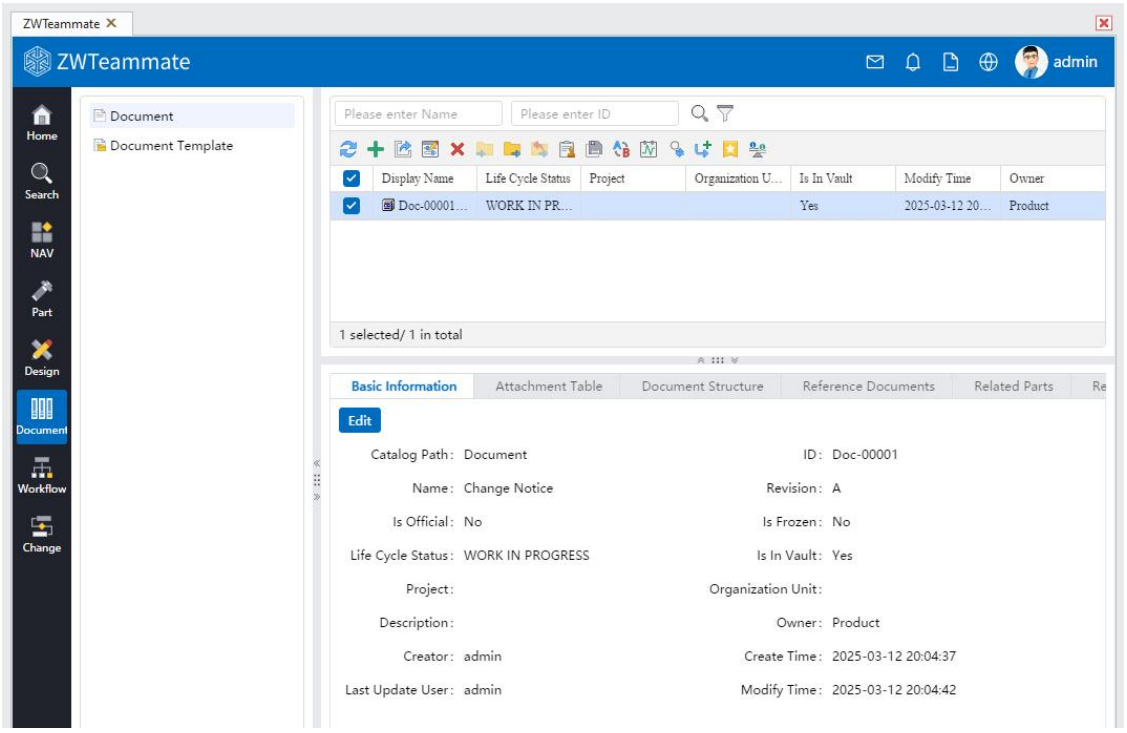

✓ Supports querying related design objects, related documents, substitute parts, linked changes, historical versions, and related workflows for specified part objects.

**【Where is it】**

ZW3D >> ZWTeammate >> Embedded Browser>> **Part**

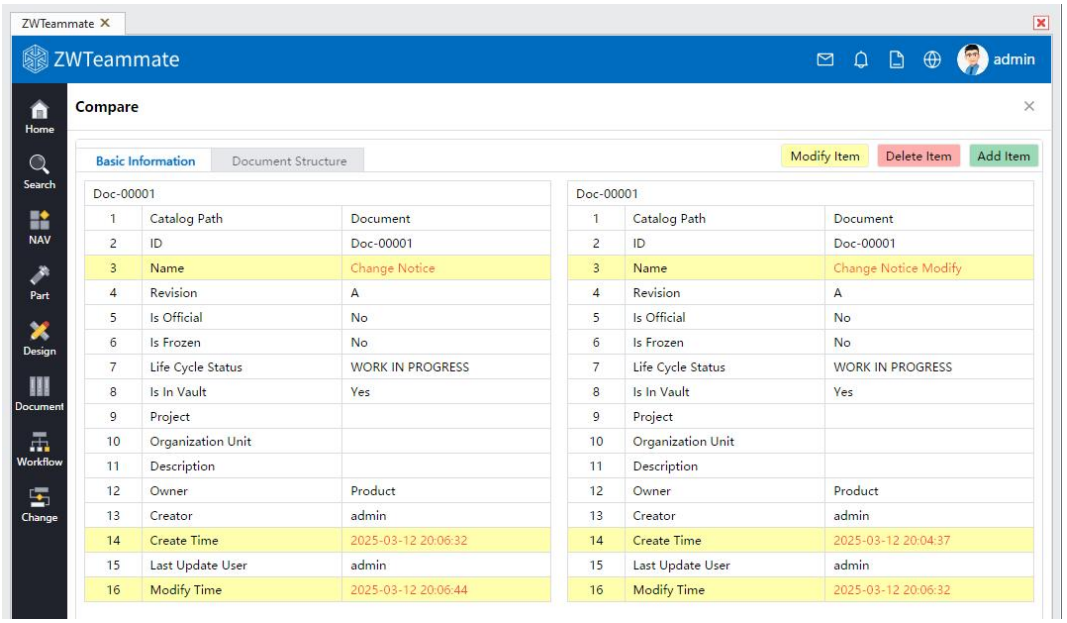
## 8.10.2 Document Management

Supports full lifecycle management of non-CAD design files, covering the entire process from document creation, editing, approval, release, to obsolescence, ensuring document integrity, security, and traceability throughout the product lifecycle.



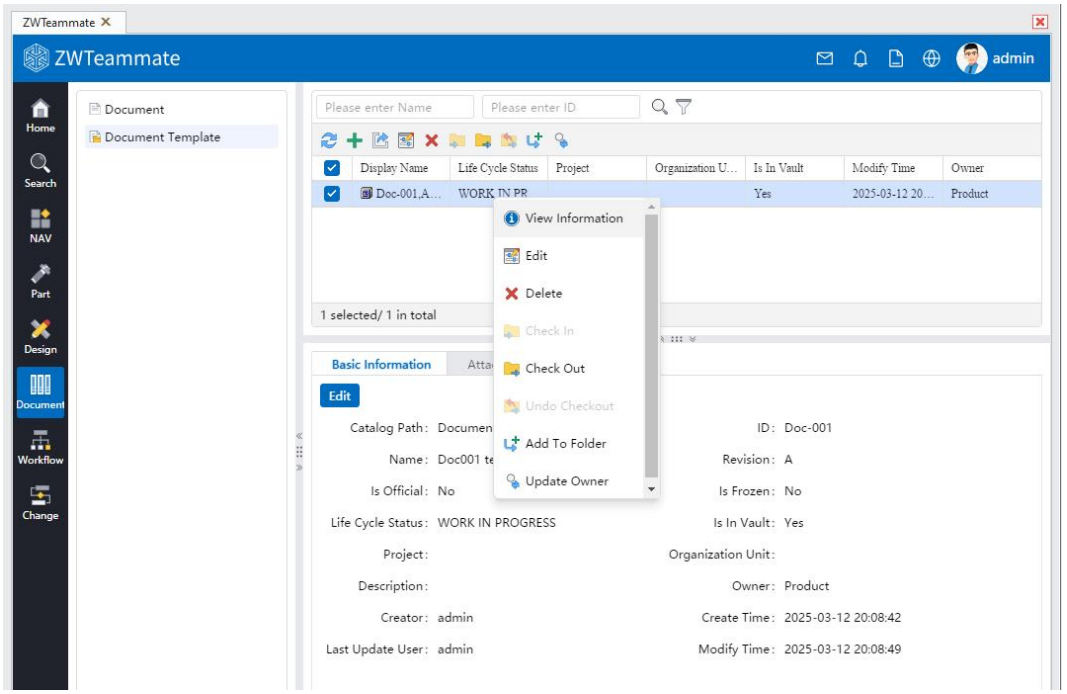

**【What users can do】**

- ✓ Supports creating, editing, and deleting documents in the web interface.
- ✓ Supports modifying life-cycle status, revising, and initiating approvals for documents.
- ✓ Supports comparing document versions and highlighting differences between versions.




✓ Supports querying basic information, attachment lists, document structures, reference documents, related parts, linked designs, related changes, version history, and related workflows for specified document objects.

✓ Supports maintaining document templates, including creating, deleting, checking in, and checking out templates.

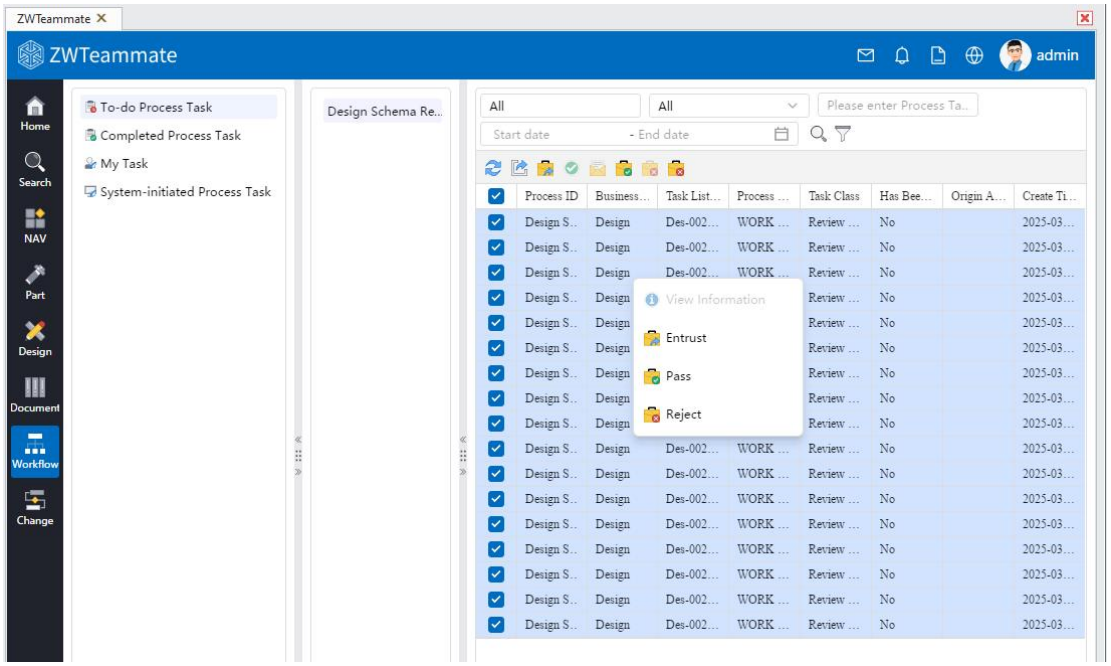


【Where is it】

ZW3D >> ZWTeammate >> Embedded Browser>> **Document**

## 8.10.3 Workflow Management

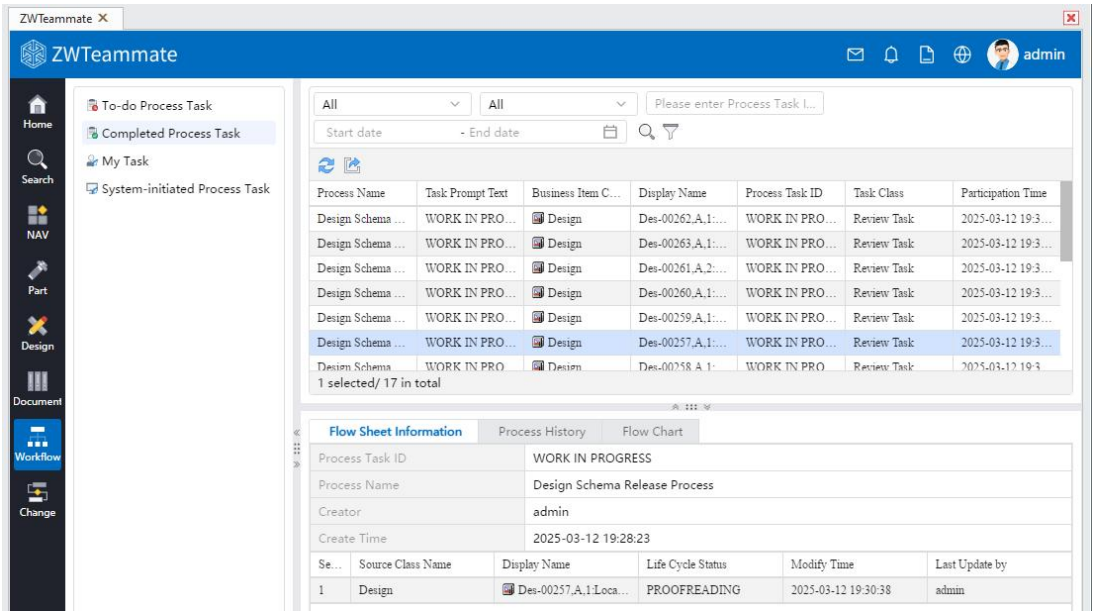
The workflow management module enables unified management of business processes such as design release, part release, document release, and ECN submission. This makes the approval process more transparent and reduces manual handoffs and waiting time. Approvers can batch-process pending workflow tasks, while applicants can query initiated workflows and track approval progress in real time.



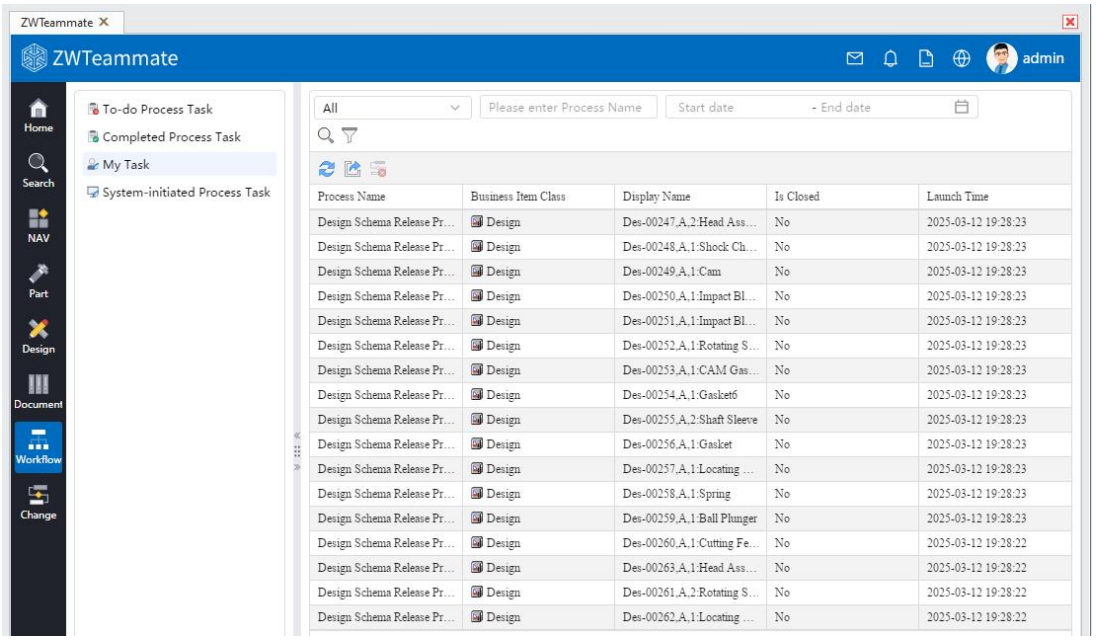

【What users can do】

✓ Supports batch processing of pending workflow tasks, including viewing, delegating, approving, and rejecting workflows.

✓ View completed workflow tasks.




✓ View the status of initiated workflow tasks.



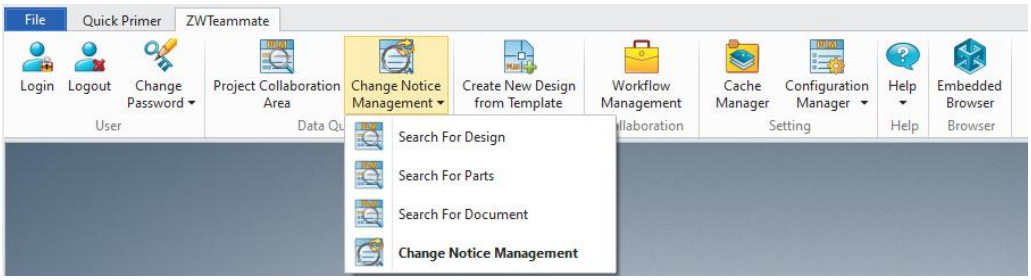

## 【Where is it】

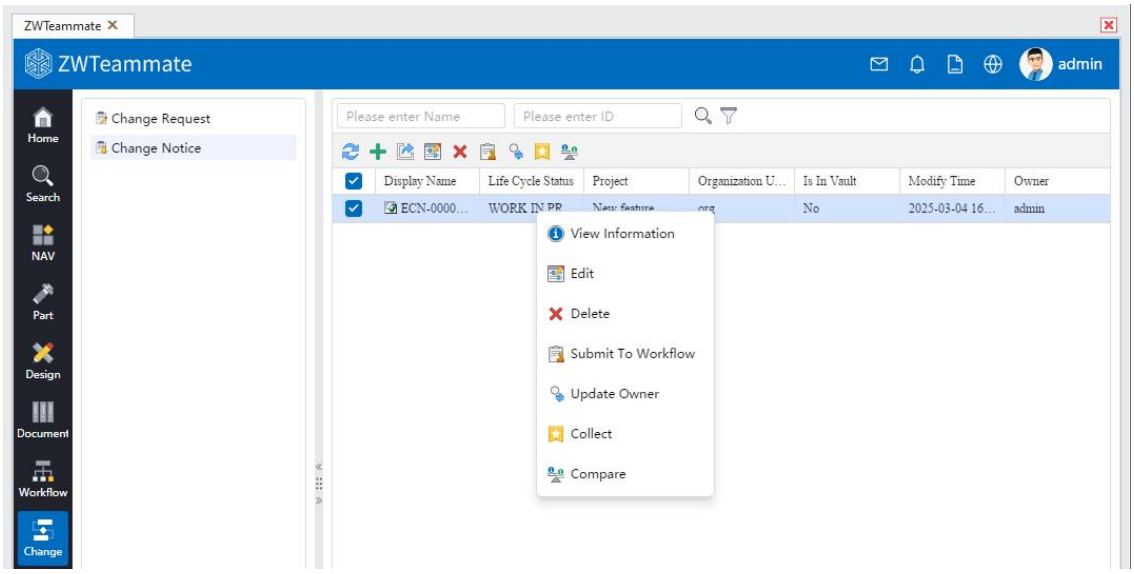
ZW3D >> ZWTeammate >> Embedded Browser >> **Workflow**

ZW3D >> ZWTeammate >> **Workflow Management**

## 8.10.4 Engineering Change Notification (ECN) Query

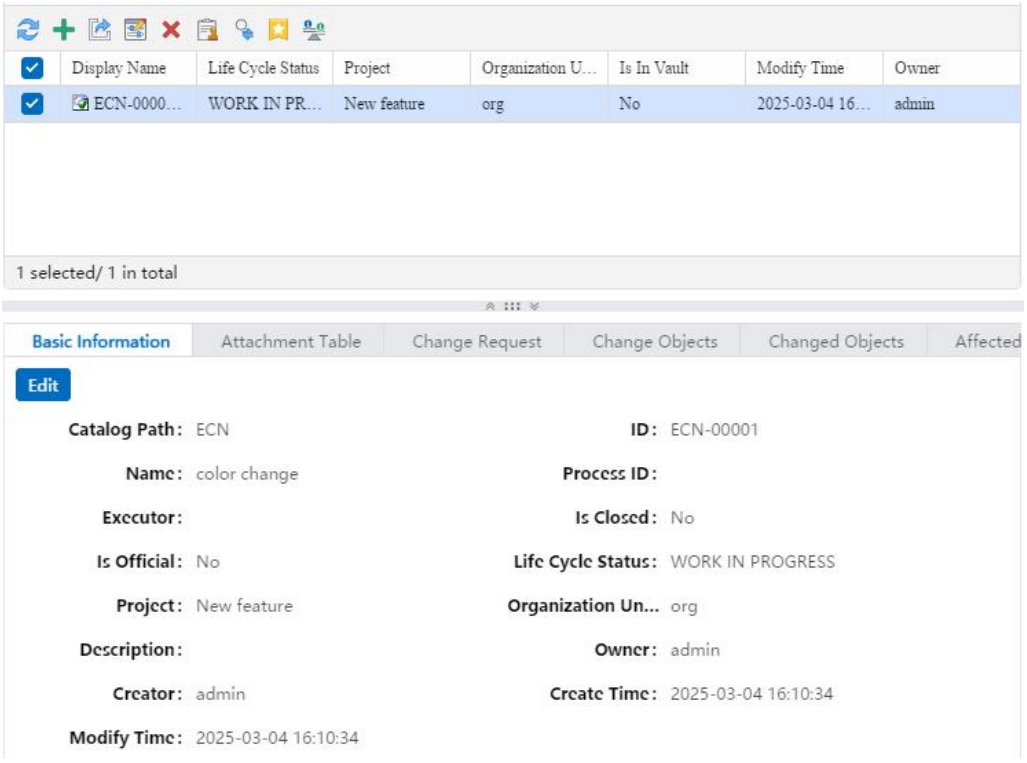
The "Change Notice Management" feature allows users to access the ZWTeammate browser and navigate to the change notification interface, where they can query and manage ECN processes.



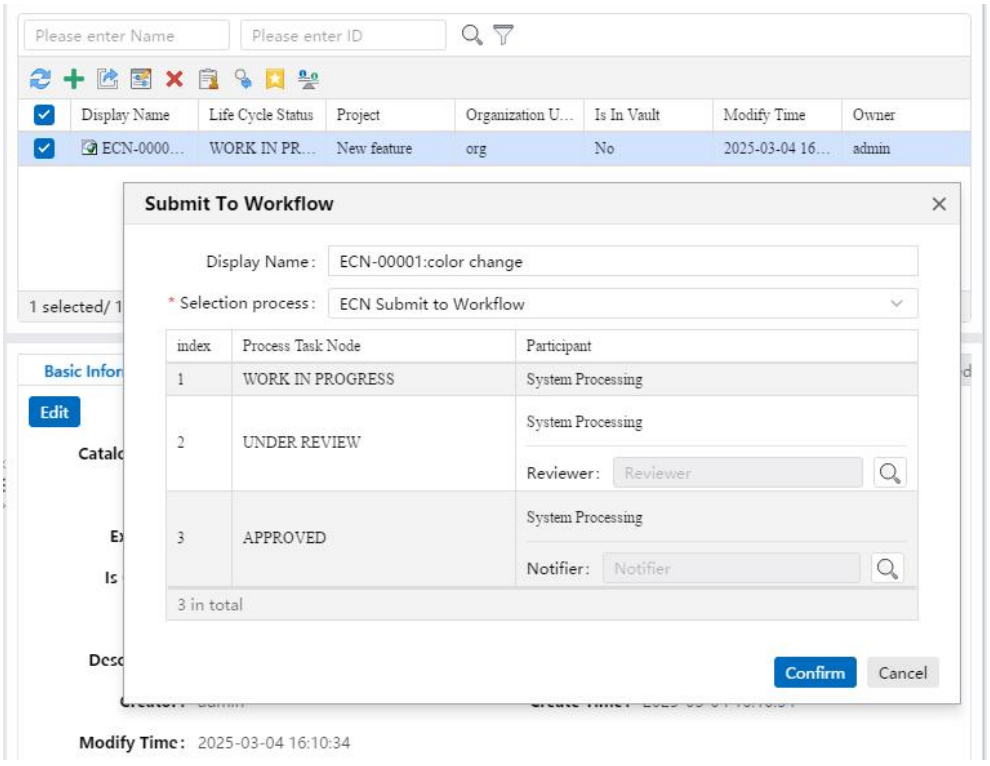




**【What users can do】**

✓ Select the change notification to be processed, supporting operations such as viewing, editing, deleting, initiating workflows, and changing ownership.




✓ Edit: Supports maintaining change notification-related information and modifying objects.




✓ Initiate Workflow: Allows starting a new workflow for the change notification.

【Where is it】

ZW3D >> ZWTeammate >> **Change Notice Management**

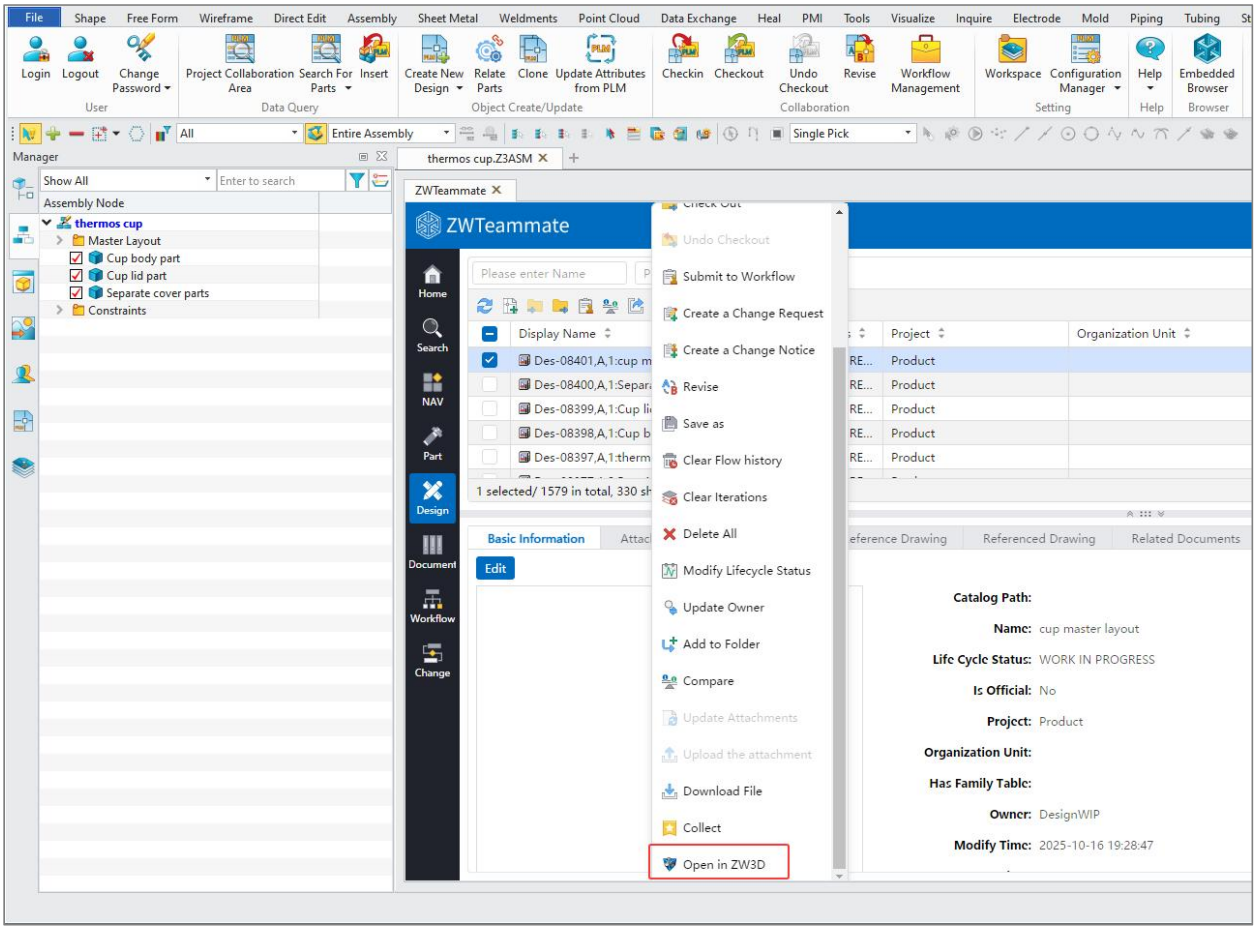
ZW3D >> ZWTeammate >> **Embedded Browser>> Change**

## 8.10.5 Open ZW3D File in Embedded Browser（ZW3D 2026 SP1）

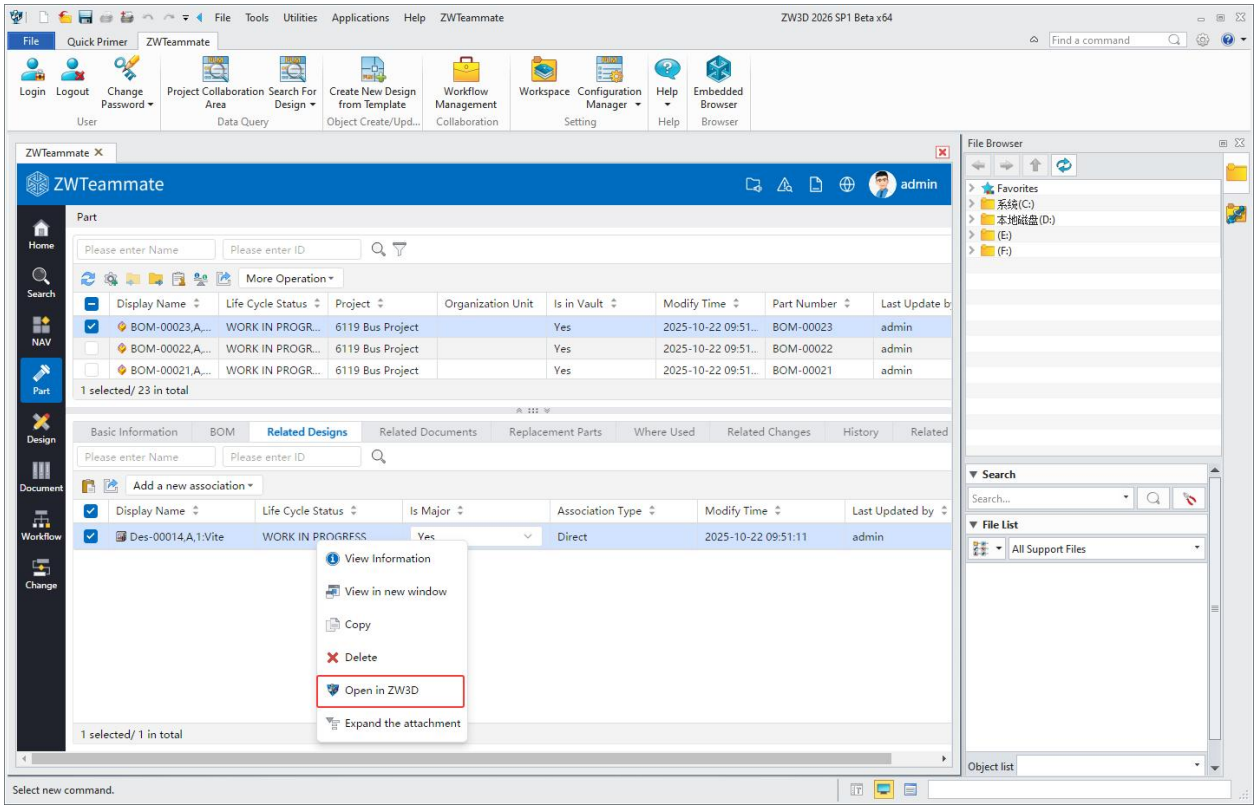
For design objects with attachment types in ZW3D file format, support executing the command to open attachments in an embedded browser.

**【What users can do】**

✓ Directly perform the “Open in ZW3D” operation on the design object.




✓ Perform the “Open in ZW3D” operation on the related design of part.




**【Where is it】**

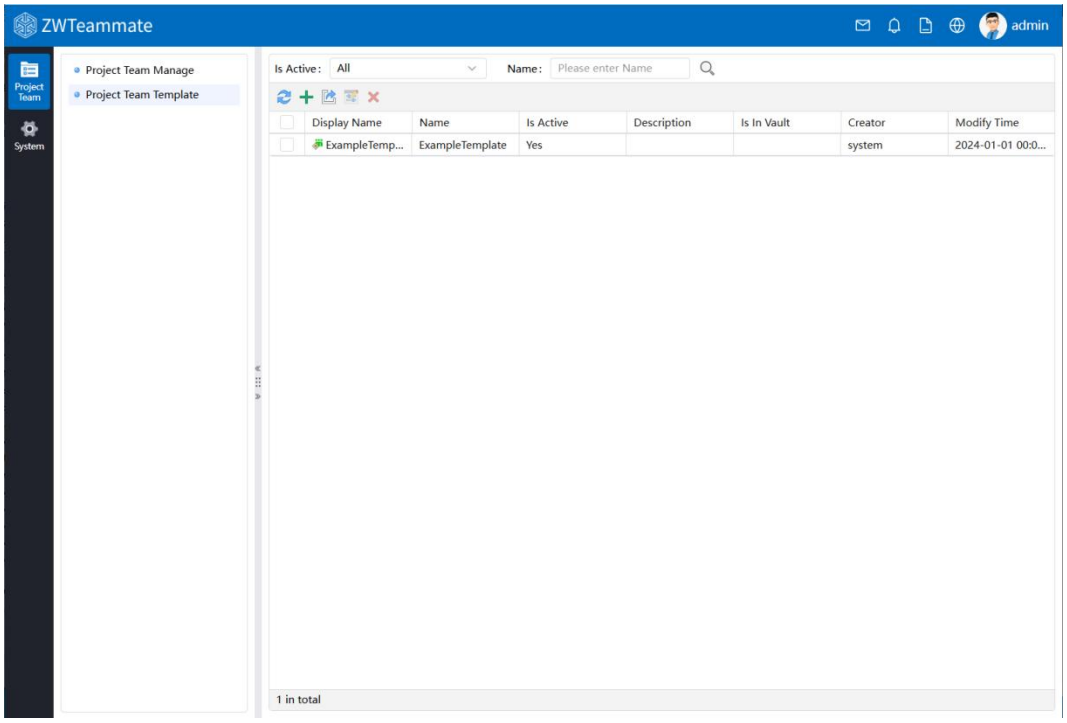
ZW3D >> ZWTeammate >> Embedded Browser>>Design>>**Open in ZW3D**

## 8.11 Standalone Web Management Interface

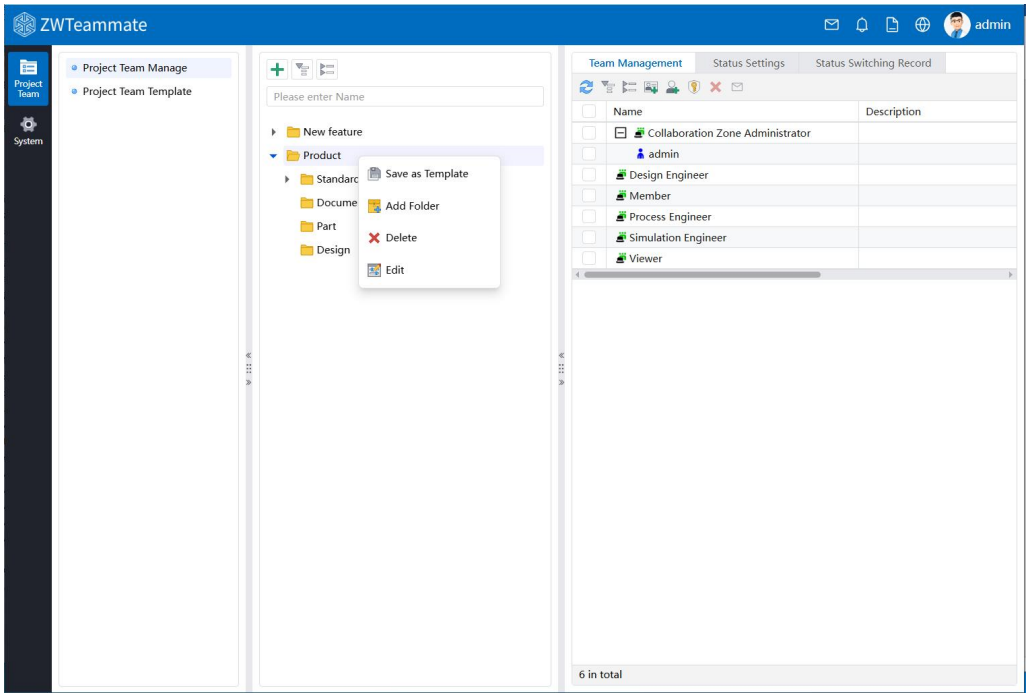
Add a separate management portal (web-based) to allow system administrators to maintain system configuration settings without logging into ZW3D. The standalone web portal includes settings for project collaboration areas, workflows, users and organizations, numbering rules, custom attributes, integration configuration files, and design templates.

**【What users can do】**

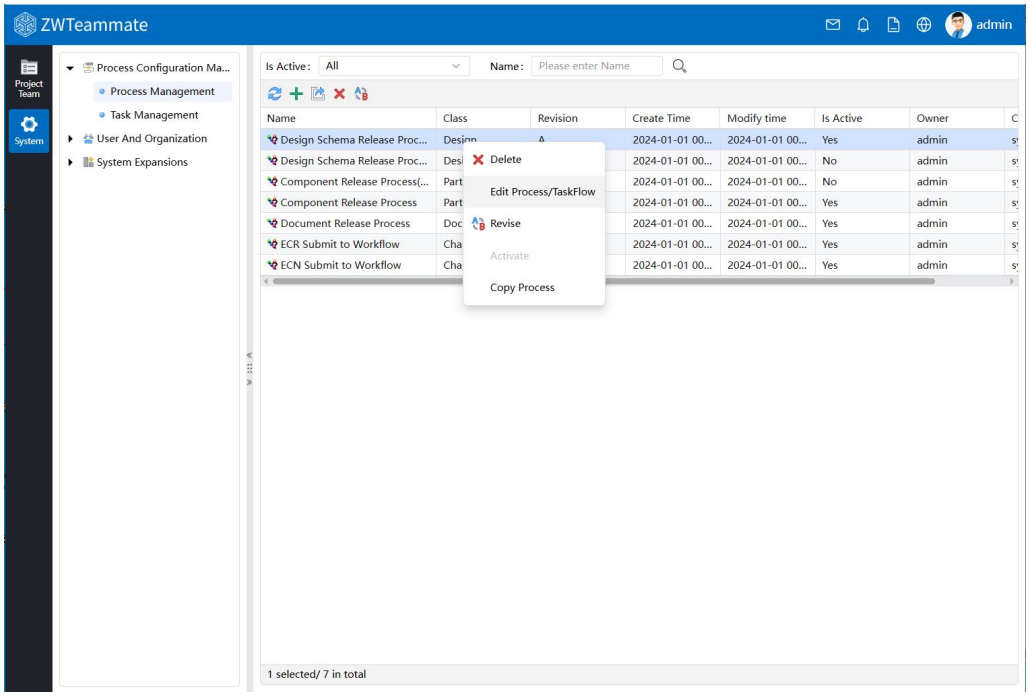
✓ Manage project collaboration area templates under "Project Team Template" in the web management browser.



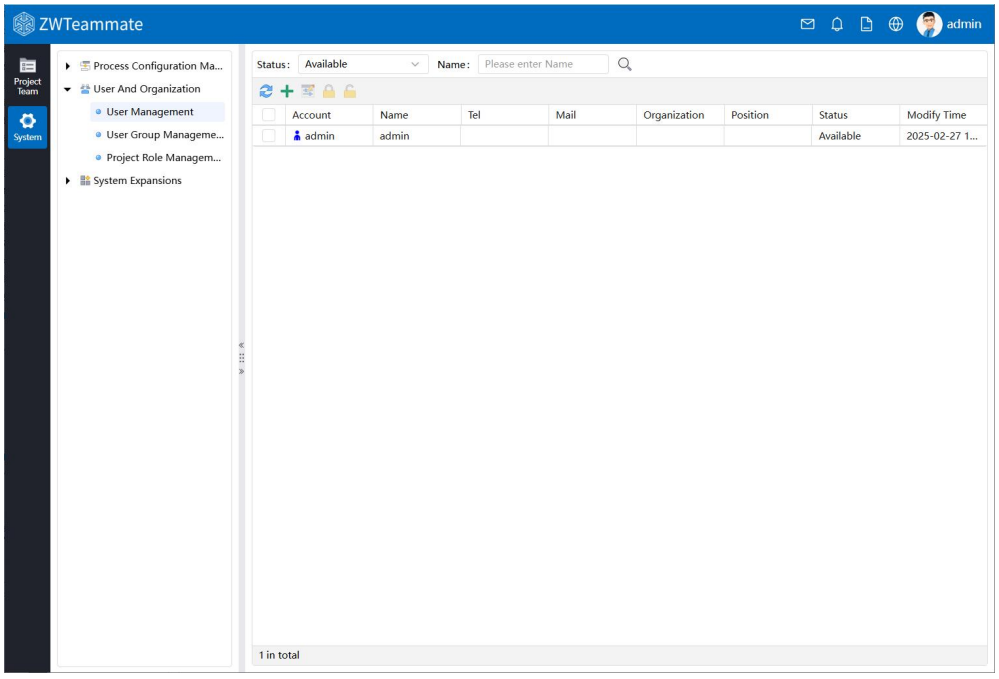

✓ Manage Project Collaboration Areas with functions such as "Add Project Team", "Team Management", "Role Authorization", and "Status Settings". Right-click on a workspace/folder to create a new folder.



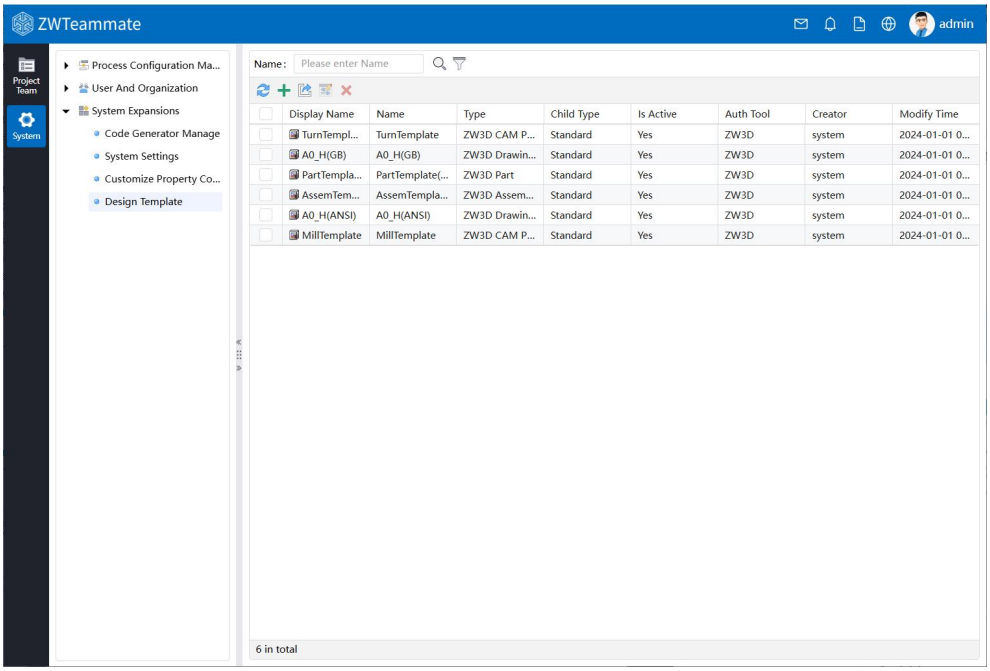
✓ Configure and manage workflows, including creating, deleting, editing, revising, activating, and copying workflows.




✓ Manage users, user groups, and roles under "User And Organization" in the web management browser.



✓ Configure system settings, including numbering rules, custom attributes, integration configuration files, and design templates.




## 【Where is it】

### Standalone Web Management Portal