

ZW3D WHAT'S NEW

V 2019



ZWSOFT CO., LTD.(Guangzhou)

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Room 01-08, 32/F, No.15, Zhujiang West Road, Tianhe District, Guangzhou 510623, China
(8620)38289780

ZW3D™ V2019 What's New

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Highlights of ZW3D 2019

Basics:	Efficiency Improvements Multiple Languages Support
Translator:	Import Drawing Sheets of CATIA/SolidWorks
CAD:	Faster History Rollback New Shape Attribute More Robust Boolean Better "Direct Edit" Expanded Assembly Manager New "Rename Assembly" New Flexible Assembly New 3D BOM
ZWMold:	Renovated Parting Design Reformed "Electrode" Reformed "Standard Library"
Drafting:	Projection View for Picked Shapes New "Auto Dimension" New "Label Coordinate" New "Stack Balloon" Updated Style Manager
CAM:	Volumill 2X and 3X Integration Optimized Link Function in 2X Enhanced Custom CL Command Enhanced Tool Path Verification

Note: Important enhancements in this article are marked with ★

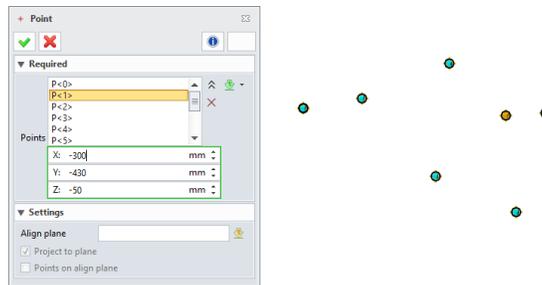
1 Basic

1.1 UI & Work flow Changes

1.1.1 Point list in Point widget

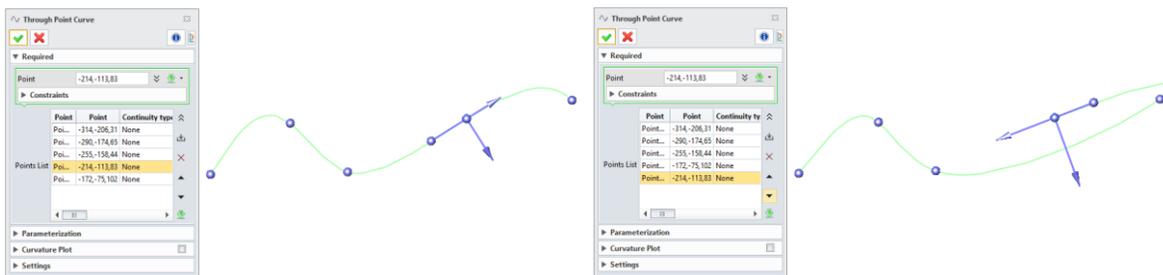
Point list is added into Point widget to facilitate users to edit or delete any picked points. Here are the details.

1. Mutual highlight between picked item within the point list and the corresponding point on the graphic area.
2. Values of X, Y, Z will be shown to allow users to edit when a point item is picked.
3. Direct deletion on any picked item is supported.



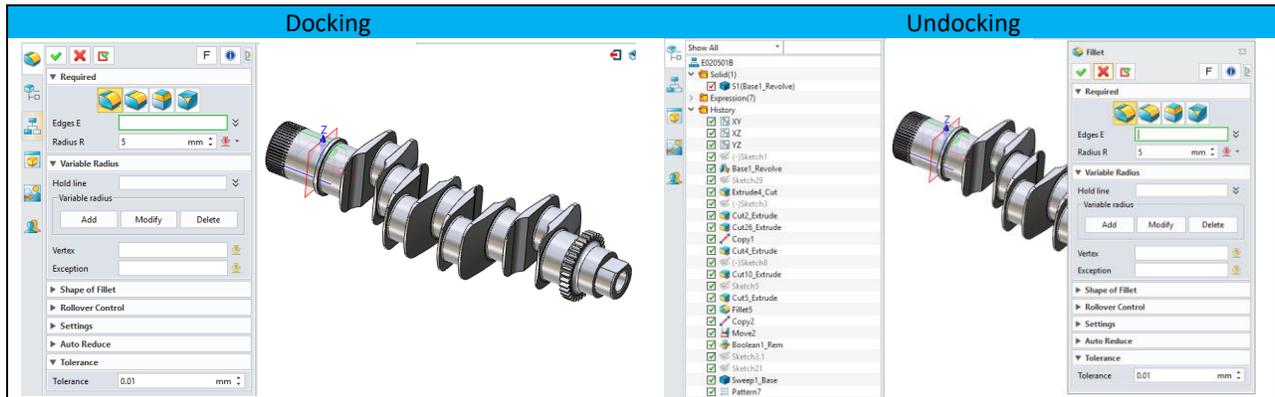
1.1.2 New Columns and Reordering Sequence in Setlist widget

Setlist widget uses the columns to show detail information for each set, so that users can directly to see what each set encloses. The up and down buttons are to reorder the sequence of each item inside the widget, so that users can re-arrange the defined items to have a different result. For example, reordering the picked points inside the Through Point Curve can have a different curve.



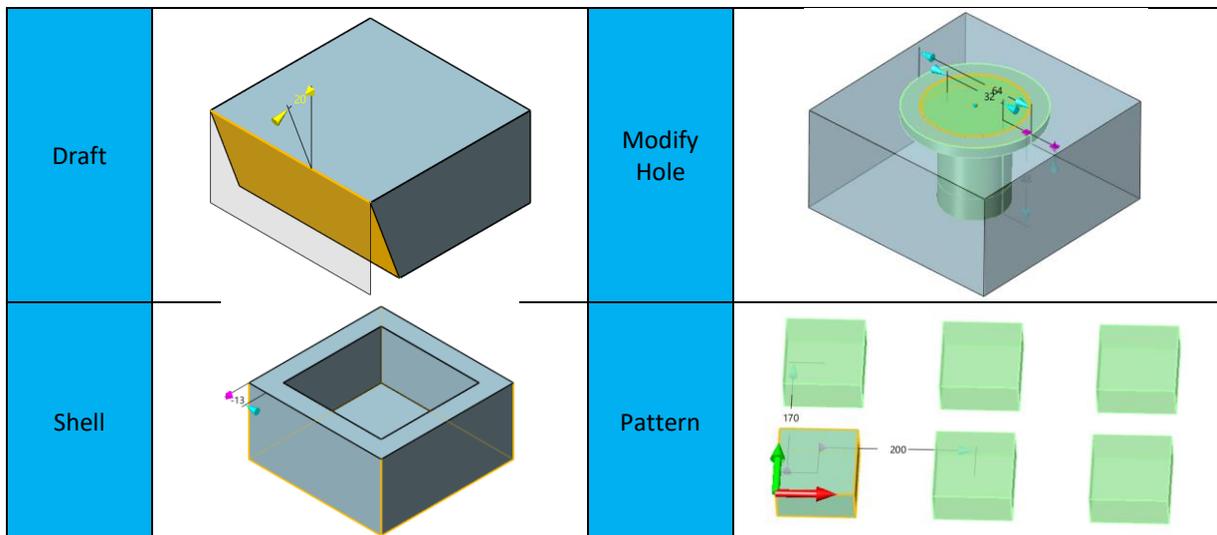
1.1.3 Dock/undock Option form

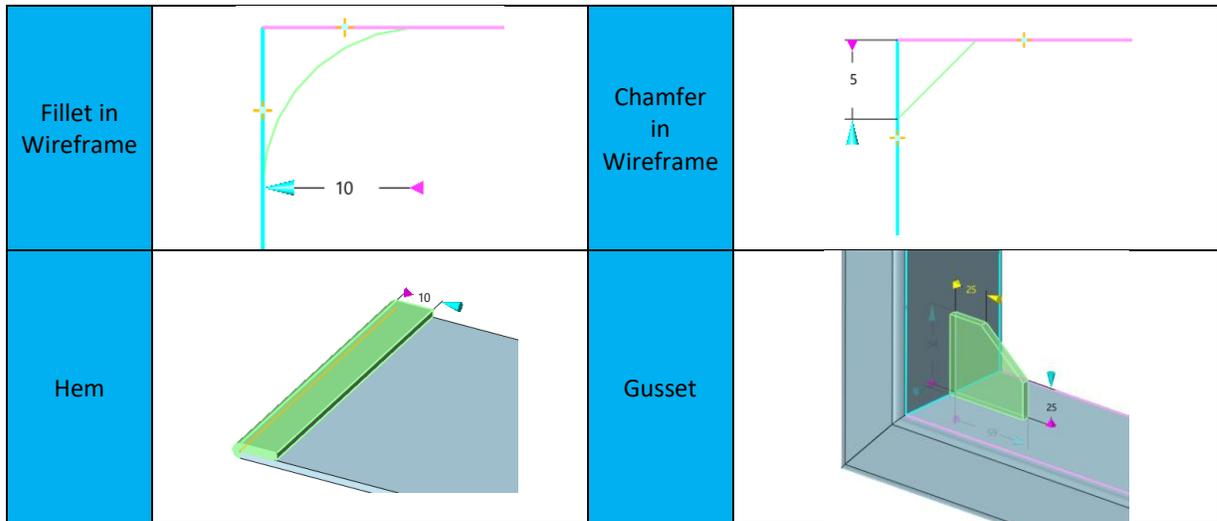
There is a new docking button on each modeling command form which users can control whether the form docks along with the data manager or separates away from. When the form is away from the data manager, users can directly see content inside the manager without the command form blocking.



1.1.4 More DDD Handle Provided

Dynamic Drag Dimension handle is used within the graphic area to allow users to change values of each field of the on-going command. Now more commands are including such handle to provide a better usage experience.





1.2 ★Efficiency Improvement

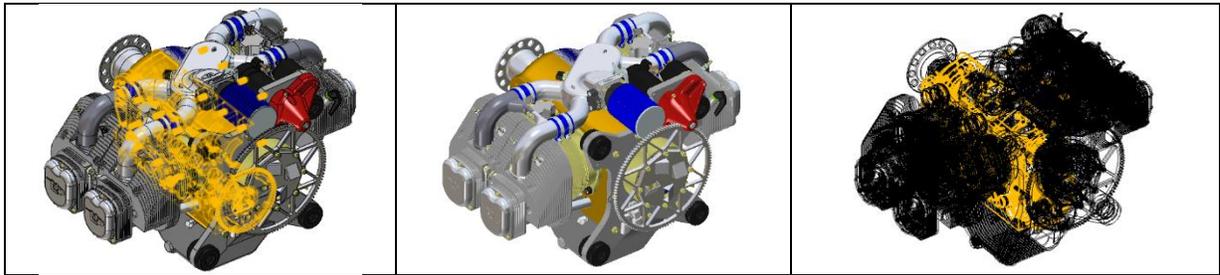
1.2.1 Speed up Display

To speed up display on complex product like big assembly and big drawing sheet, ZW3D 2019 has applied different technologies to optimize display basing on various application scenarios providing quite good efficiency speedup. Here are the details.

1.2.1.1 *New auto display degradation to improve usage experience on big assembly*

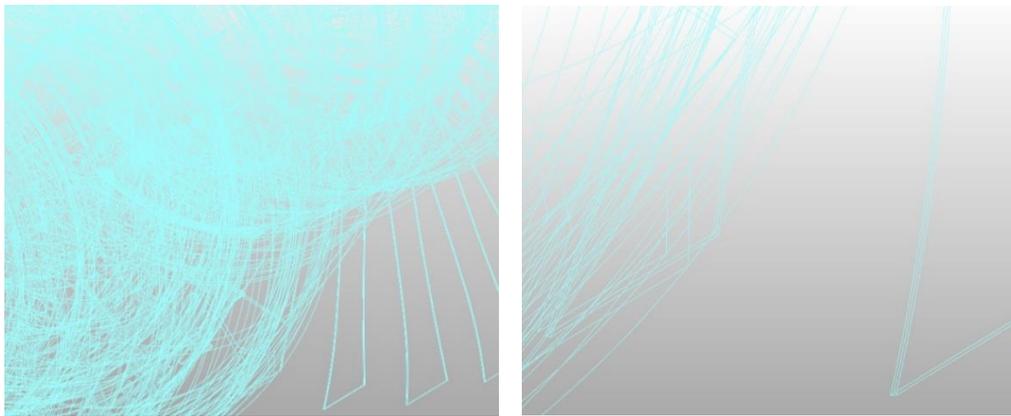
3 display degradation option are provided by “Configuration > Display > Numeric settings > Display degradation”. Users can pick the suitable one to improve the display efficiency to make the work progress forward. There is a time threshold for users to set up how much second the degradation will tack place.

No display degradation	No-edge display degradation	Wireframe display degradation
<p>No display degradation to improve display efficiency, no matter how big the model is.</p>	<p>Using shaded display without edges to speed up the display, and anti-Alias and see-through effect are disabled.</p>	<p>If the display with No-edge display degradation still exceed the time threshold, wireframe display degradation will be applied to further reduce objects to display.</p>



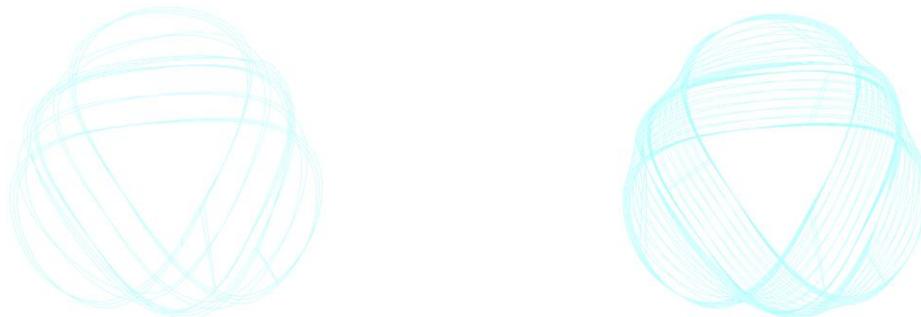
1.2.1.2 *Improve the wireframe display on big product*

This new version has optimized the wireframe display to allow users zoom/pan/rotate the big product which contains thousands of shapes or curves smoothly.



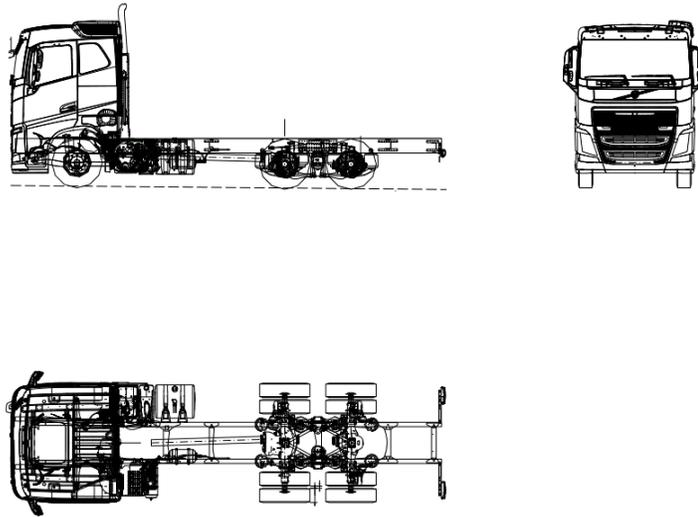
1.2.1.3 *New settings for UV isolines of faces to simplify wireframe display*

“Configuration > Display > Option & defaults > U/V isolines” can set up how much isolines the new face will have. Less isolines can simplify wireframe display to present a better view. Besides using the “Modify isoline Number” command to modify isolines of existing faces, a regeneration will make the faces updated with the U/V isolines setting above.



1.2.1.4 Drafting display speedup on very complex drawing sheet

Drafting has applied a special technology to drastically speed up the display efficiency on complex drawing sheet, especially with hundreds of thousands of entities. Users can pan, zoom in/out or rotate the big drafting smoothly.



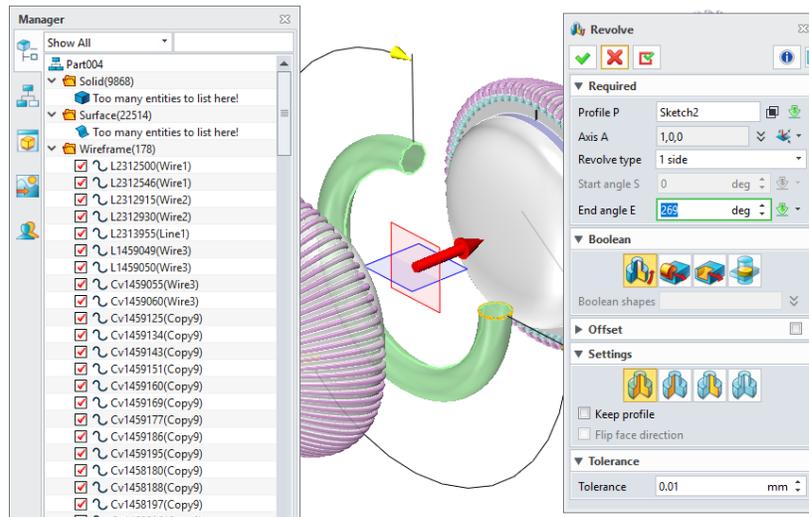
1.2.1.5 Accelerated Blank/Unblank efficiency

Commands related to visibility has been improved to provide users a better efficiency, like Blank, Unblank, Swap Entity Visibility and so on. Users can pick hundreds of thousands of entities like curves and shapes to blank at one time to blank or unblank, which only take seconds.

1.2.2 Operation Efficiency on Command Form

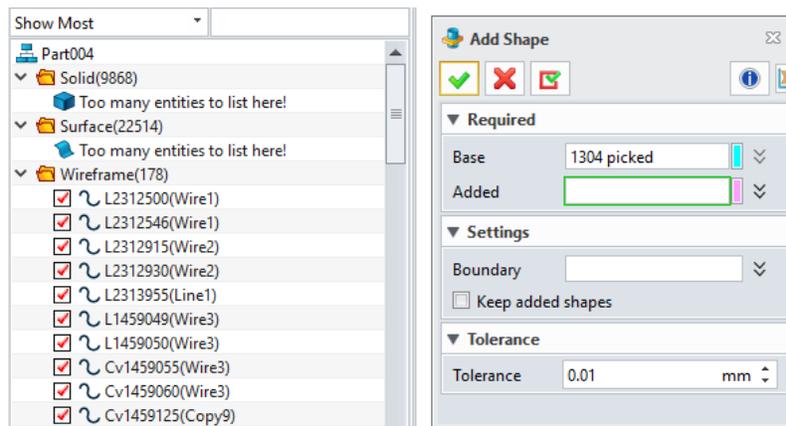
1.2.2.1 Faster preview

ZW3D 2019 has optimized the preview echo to improve its efficiency on complex parts with hundreds of thousands of shapes, so that users can work on those big products smoothly. If the model is not that big and users prefer to use the previous preview effect, users can change “Visual Manager > Echo > Echo type” to “True Echo”.



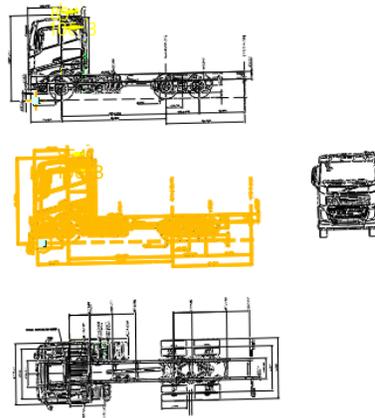
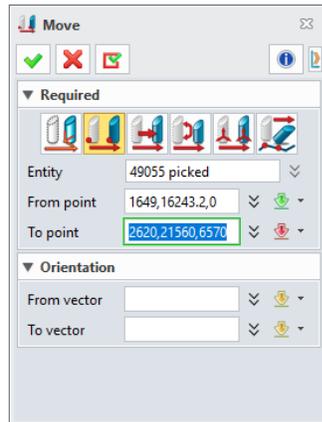
1.2.2.2 Quick reaction for fields with big number input on command form

ZW3D 2019 has optimized the response of fields on the command form when some quite big input is added into to avoid previous slowness, so that users can quickly activate next field to continue their works.



1.2.2.3 Quicker window pick under part context

Window pick is refined to work much faster to pick hundreds of thousands of entities, such as thousands of curves to move.



1.2.3 Modeling Efficiency

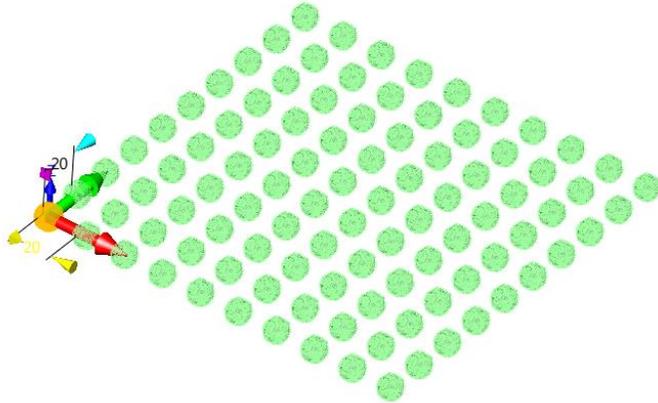
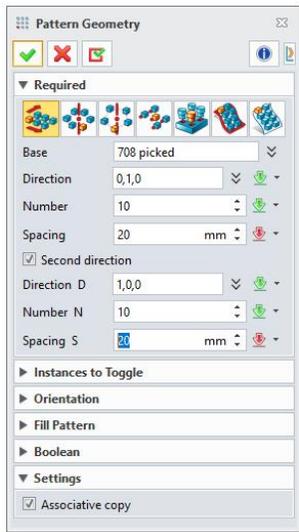
1.2.3.1 New Quick Rollback to Speed up History Redefinition

“Quick Rollback” is a new functionality added to speed up the history rollback during redefinition or rollback. After enabling “Configuration > Part > General > Backup data for history quick rollback” option, there is a cache file to save major outcome data of each modeling feature. When a feature redefinition or rollback takes place, corresponding feature data will be fetched to restore the modeling field as it was created avoiding the regeneration of modeling features before the redefining feature, so that no more waiting time is needed.

1.2.3.2 Better Efficiency on Commands

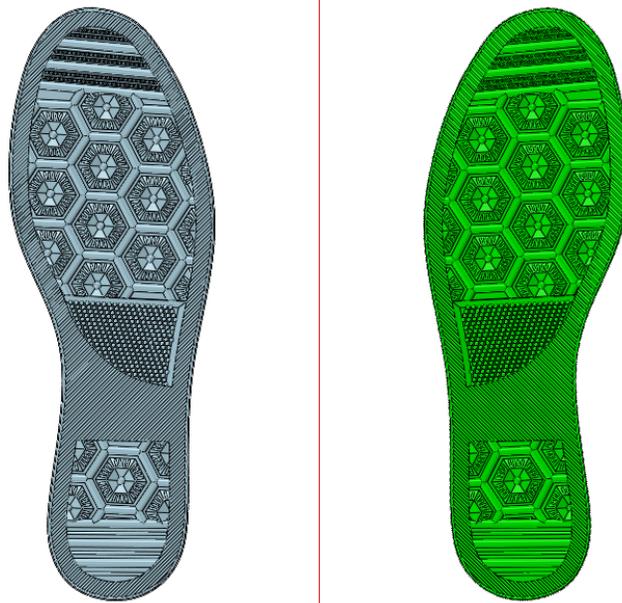
1. Faster “Pattern Geometry”

“Pattern Geometry” has been optimized to better handle tens of thousands of shape instances at one operation, as following 708 shapes patterning 20x20. Memory should be big enough when doing such patterning with huge quantity.



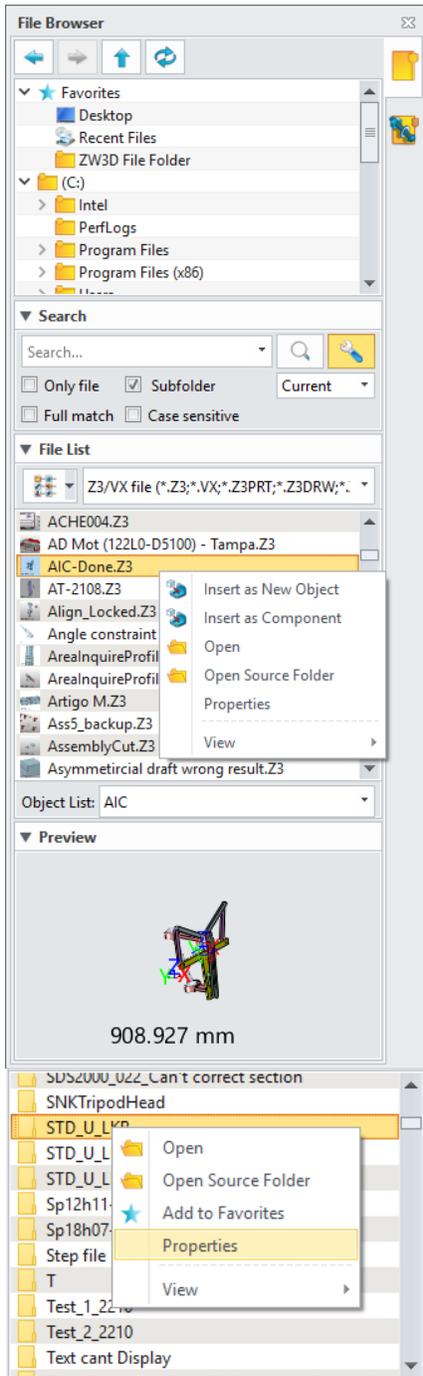
2. Accelerated “Mirror Geometry”

Similar with “Pattern Geometry”, “Mirror Geometry” can better handle single complex shape as following sample which contain a shape with almost 50 thousand of faces.



1.3 Updated File Browser

Since “Reuse Library” is embedded into the File Browser, File Browser has adjusted accordingly. Here are the details.



1. Favorites and Navigator have merged into one

2. More new settings in Search

Users can use different settings to make the search more efficient and accurate.

- Only file
- Subfolder
- Full match
- Case sensitive

3. Right-click menu is supported on the file list

The right-click menu on the file:

- **Insert as New Object** – a new instance will be generated basing on the picked file and inserted into current assembly as component.
- **Insert as Component** – insert picked file into current assembly as component.
- **Open** – directly open the pick file
- **Open Source Folder** – open the directory of picked file
- **Properties** – Open the property form of the picked file

The right-click menu on the folder:

- Open
- Open Source Folder
- Add to Favorites
- Properties

4. New “Object List” to show all objects enclosed in the picked file

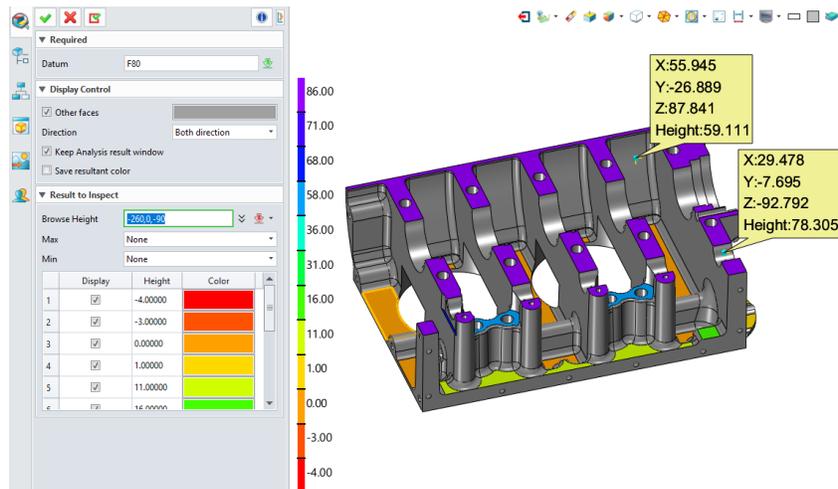
5. Preview is supported

1.4 Inquiry

1.4.1 New “Height Analysis”

“Height Analysis” can find all height values of all planar faces inside current model basing on given datum. Here are the details.

1. Automatically identify all planar faces
2. Any plane or planar face can be the start datum.
3. “Max” and “Min” options can narrow down the result to locate specific range of the result.
4. “Browser height” can check out multiple heights of any picked points right away.
5. Non-planar faces can be hidden or specified special color.
6. The resultant color can save into the face attribute of corresponding faces.
7. Faces of each height can be specified color inside the result list.



The analysis result is only available when the “Height Analysis” command is on. If you like to check the result after the command finishes, you can turn on the “Keep analysis result window” option on, then you can use “Height analysis result” command to bring up the result again.

➔ Where it is

Part/Assembly context > Inquiry Ribbon > Inspect Entities > Height Analysis

1.4.2 New “Undercut Analysis”

“Undercut Analysis” projects light in given direction to current model, then colors the faces which are illuminated and not. Users can use this function to find out which part of the model is blocking the molding, or which part can’t be machined in given direction.

1. Color is classified into 4 types to distinguish each other

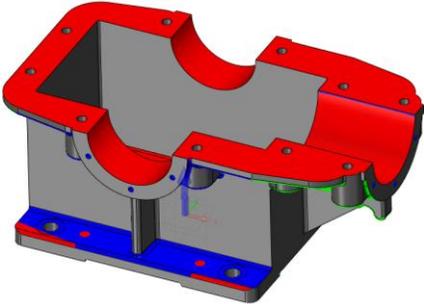
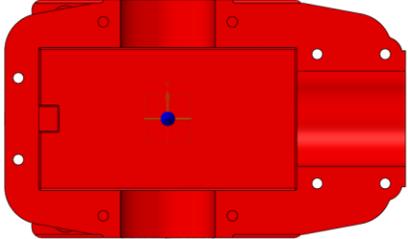
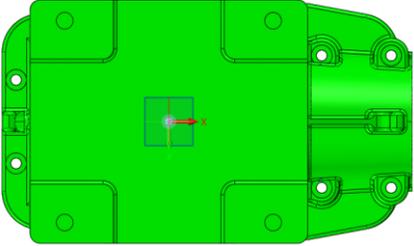
Undercut - faces that can’t be illuminated in both direction

Vertical - faces that are vertical to the given direction

Top - faces that can be illuminated in positive direction

Bottom - faces that can be illuminated in negative direction

2. “Browser Angle” field can check any angle info of picked points

<p>Isometric view</p>	
<p>Top view</p>	
<p>Bottom view</p>	

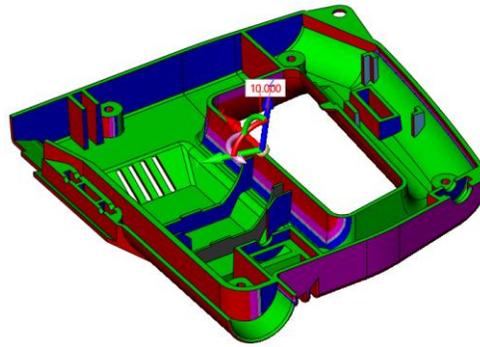
➔ Where it is

Part/Assembly context > Inquiry Ribbon > Inspect Entities > Undercut Analysis

1.4.3 Others

1. New handle for changing direction on-place

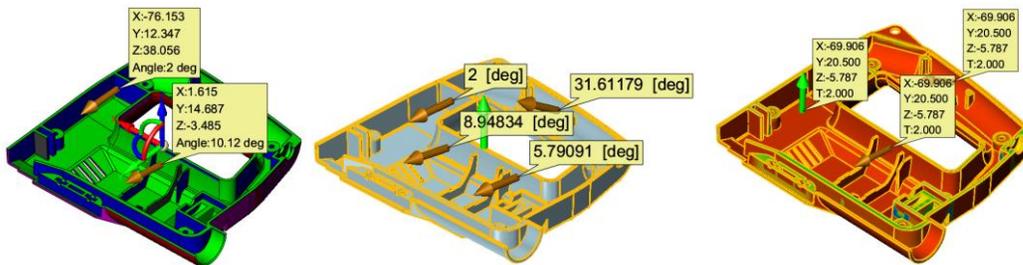
After “Direction” field is defined, the handle will show on the model. Users can adjust the hand to change the direction to check different result.



2. Multiple points to on-place check is added for various inquiry commands

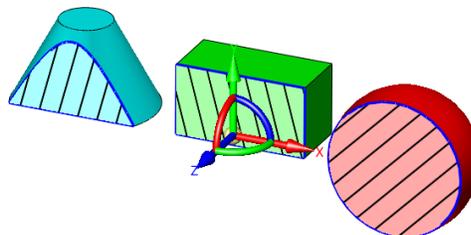
Here is the list of commands whose “Browser” field is supported multiple picked points.

Draft in Face Analysis, Check Draft, Thickness Analysis.



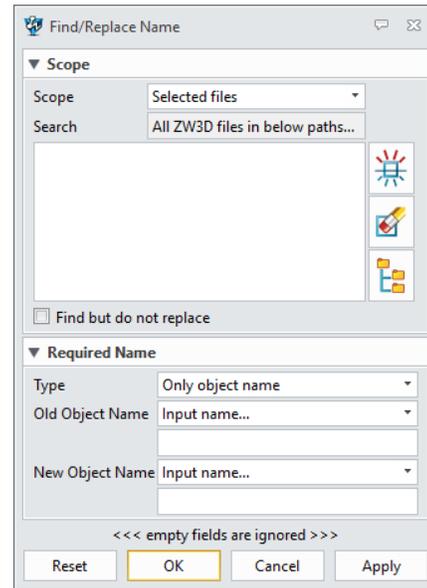
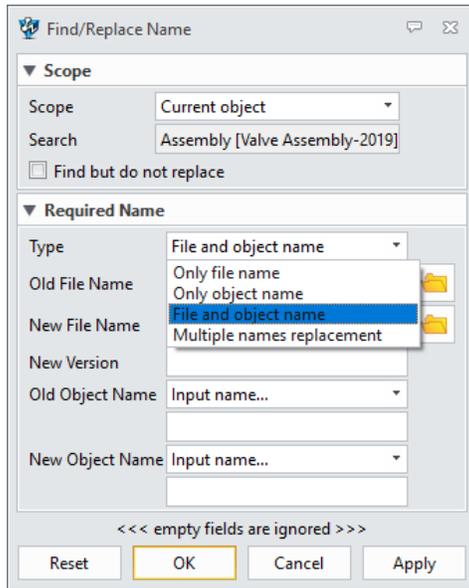
3. “Dynamic Section View” supports to use shape color

Since the shape color can be different, users can distinguish each section of different shapes.



1.5 “Find/Replace Name” Adjustment

“Find/Replace Name” have adjusted the workflow on its form and given more restriction to avoid unwanted replacement. Here are the details:



1. Specify search scope

The search scope is clearly defined with different options, and when doing multiple Find/Replace, users must select which directories are involved.

2. Specify what name should be replaced

New type is offered to allow users define which name should be find and replace:

- **Only file name**

If the input file name is found in the link info inside current search object, it will be replaced with new file name without any further consideration.

- **Only object name**

If the input object name is found in the link info inside current search object, it will be replaced with new object name without any further consideration.

- **File and object name**

Only when the file name and object name are both found in the link info of current object, then the replacement takes place, otherwise no action happens.

- **Multiple names replacement**

External Excel file is used as the name input for Find/Replace. the excel can be defined as following:

- ✧ Data should start on first row and column.
- ✧ Name order on each row is: old file name, new file name, old object name, new object name.
- ✧ One row represents one find/replace.
- ✧ Empty cell means no find/replace.

OldSampleFileName.Z3	NewSampleFileName.Z3	OldPartName	NewPartName
		OldPartName1	NewPartName1
OldSampleFileName2.Z3	NewSampleFileName2.Z3		
OldSampleFileName3.Z3	NewSampleFileName3.Z3	OldPartName3	NewPartName3

- **Specify file name or object name by picking a file**

When specifying a file name to search, users can use the file folder button to open the desired one, then all object names inside the file will be read out to allow users to pick as the object name to make sure the object exists.

After opening a file with the folder button, “Reset” button can be used to clear all the fields.

➔ **Where it is**

Root context > Root Ribbon > Utilities > Find/Replace Name

Root context > Utilities Menu > Find/Replace Name

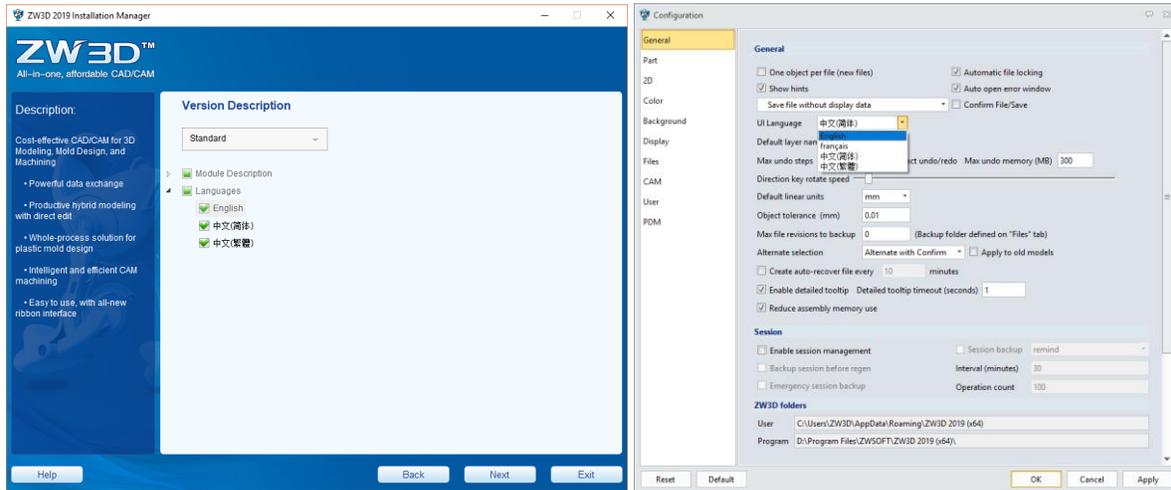
Part/Assembly context > Tools Ribbon > Utilities > Find/Replace Name

Part/Assembly context > Utilities Menu > Find/Replace Name

Drafting context > Tools Ribbon > Utilities > Find/Replace Name

1.6 ★Multiple Languages Support

ZW3D 2019 installation package includes different language resources, so that users can install a ZW3D with different UI languages. After installation, users can toggle the UI language through “Configuration > General > General > UI Language”.



1.7 System requirement change

XP is no longer supported from ZW3D 2019. Here is the supported OS for ZW3D:

- Win 7 SP1 and above
- Win 8.1
- Win 10

For ZW3D Network License Manager, here are the supported OS:

- Win7 sp1, Win 8.1, Win10
- Microsoft Windows Server 2008 R2 SP1
- Microsoft Windows Server 2012
- Windows Server 2016

2 Translator

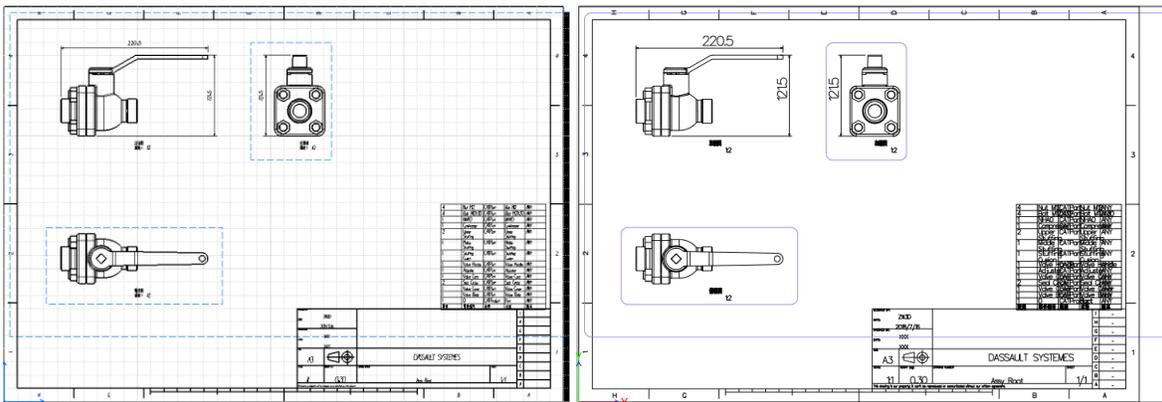
2.1 ★New Support to Import Drawing Sheet of CATIA and SolidWorks

ZW3D 2019 can directly import the drawing sheet file of CATIA and SolidWorks without DWG/DXF as the media. Here are some details:

- The drawing sheet file can import into part and drafting context.
- Multiple sheets can be imported

When importing into part context, entities of each sheet will be put into different layers.

When importing into drafting context, multiple sheet will be created.



2.2 Import Format Update

The updated formats are marked in red.

Format	Suffix	Supported Version
Catia V4	.model, .exp, .session	4.1.9 – 4.2.4
Catia V5/V6	.CATPart, .CATProduct, .CGR, .3DXML	V5R8---V5/V6 R2018
NX(UG)	.prt	11– NX 12
Creo(Pro/E)	.prt, .prt*, .asm, .asm.*	16 – Creo 5.0
SolidWorks	.sldprt, .sldasm	98– 2018(Only 64 支持)
SolidEdge	.par, .asm, .psm	V18 – ST10

Inventor	.ipt, .iam	Up to V2019
ACIS	.sat, .sab, .asat, .asab	R1 – 2018 1.0
DWG	.dwg	R11 - 2013
DXF	.dxf	R11 - 2013
IGES	.ige, .iges	
STEP	.stp, .step	203, 214
Parasolid	.x_t, .x_b, .xmt_txt, .xmt_bin	Up to 30.0
VDA	.vda	
Image File	*.bmp, *.gif, *.jpg, *.jpeg, *.tif, *.tiff	
Neutral File	*.z3n, *.v3n	
PartSolutions	*.ps2, *.ps3	
STL	*.stl	
3DXML	.3dxml	V4.3
XCGM	.x cgm	R2012-2016 1.1
JT	.jt	JT 8.x/9.x, 10~10.2
OBJ	.obj	

2.3 Other Improvements

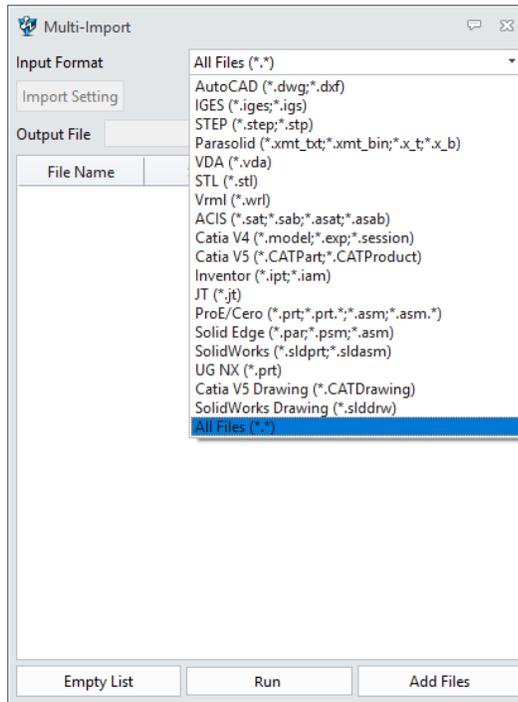
Basing on the feedback, here are some changes done for translator:

1. New “Read empty layer” option for IGES, DXF format import

If there is no entity inside a layer, the layer won’t be imported by default. If users prefer to keep that layer, please check on this option.

2. More formats supported for “Multi-Import”

More common data exchange formats are supported in Multi-Import as following shown.



3. Faster import efficiency for files containing numerous shells

Sewing shell during import has been optimized, so that file with numerous shells can be imported much faster, like 50%+.

4. New “Export folder” option on “Configuration > File”

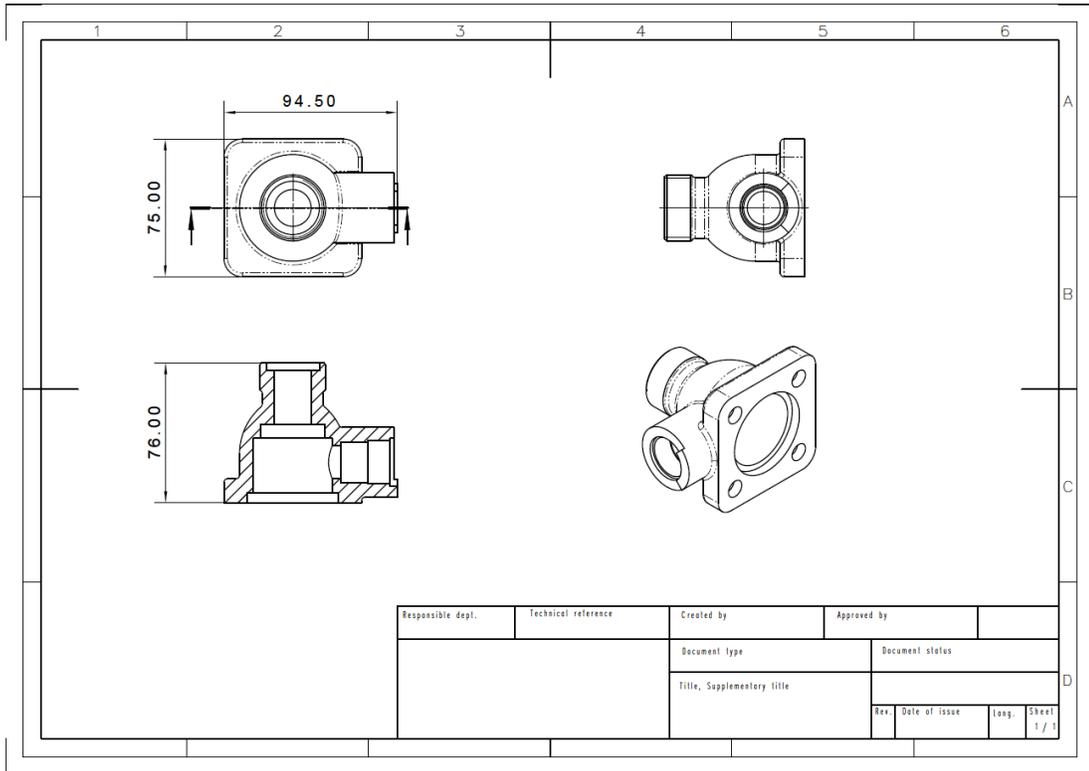
The new option can set up what directory the exporting file is saved into. If the option is blank, ZW3D file folder will be used.

5. Parasolid format updated to 30

Besides importing Parasolid file with version 30, the export also can write out the file with version 30.

6. New “Line Width Scale” in PDF export

PDF export can maintain the line width of the curves. If the lines of the exported PDF are too thick, users can scale it down.

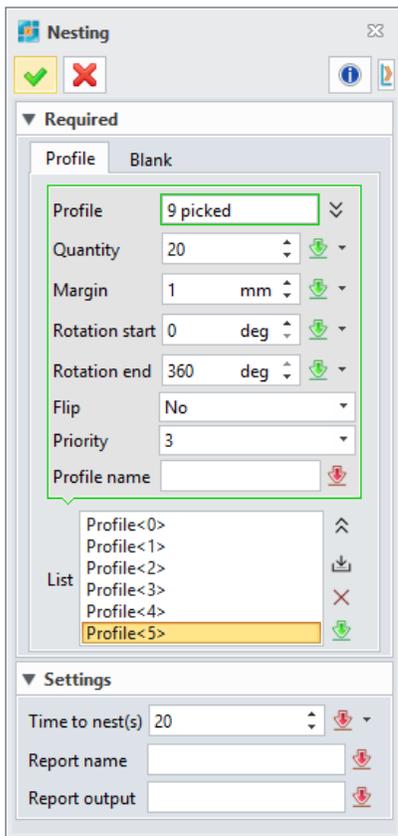


3 CAD

3.1 New “Nesting” in Sketch

Nesting can directly pick any geometries to form close loops as Nesting Profiles and Blank. Nesting needs a specified dongle key inserted into user’s PC to activate before starting.

1. Nesting profile definition



- “Profile” field allows users to select any sketch entities to define a profile following some rules:

- ✧ Only one outer and closed loop
- ✧ Inner loops can’t intersect or nest with each other

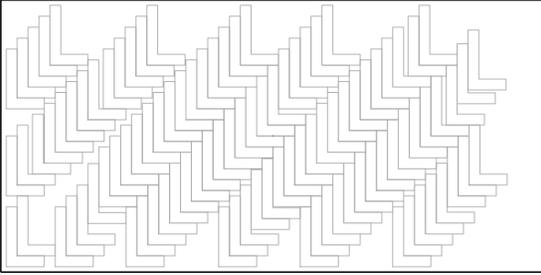
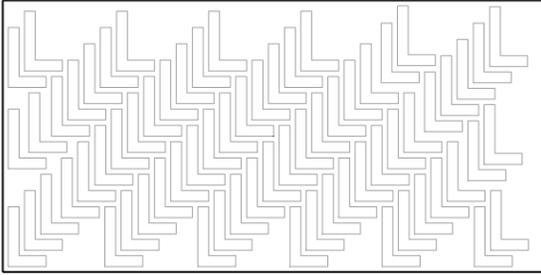
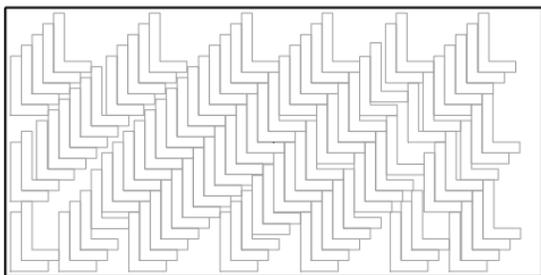
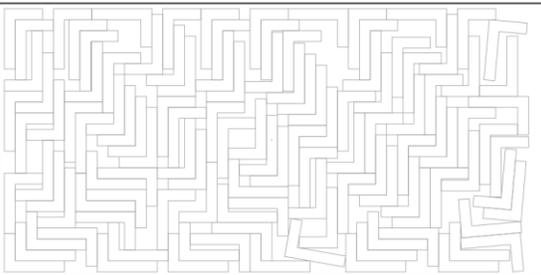
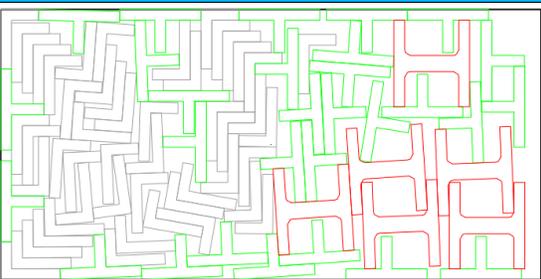
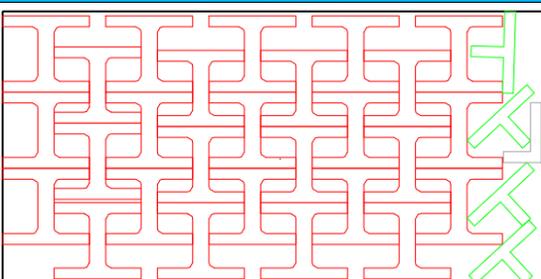
- “Quantity” field is to define how much the profile defined above is planned to nest. If the blank is big enough, the profile will be nested as the plan quantity, but if not, the actual nesting quantity will be less than the plan.

- “Margin” field is to define the gap around the profile.

- “Rotation start” and “Rotation end” are the angle range allowing the profile rotating during nesting if needed to accommodate the profile as much as possible.

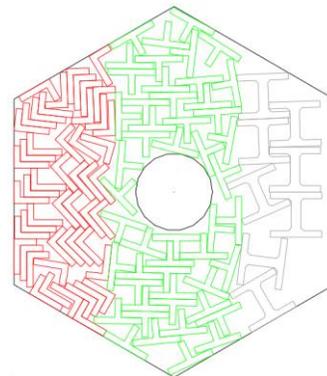
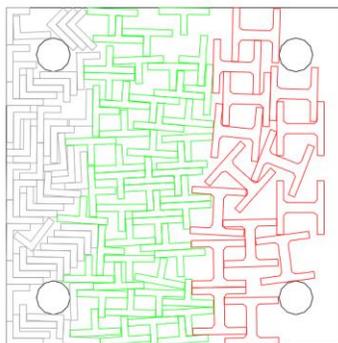
- “Flip” is to define whether the profile can be flip around during nest when necessary.

- “Priority” is to define the nest sequence when multiple profiles are about to nest.
- “Profile name” is to give each profile a name to distinguish each other.

<p>0mm Margin, 100 Planned, 100 Nested</p> 	<p>5mm Margin, 200 Planned, 70 Nested</p> 
<p>0mm Margin, No rotation, No flip</p> 	<p>0mm Margin, 360 rotation, Flip</p> 
<p>Different priority: L 1st, H the last</p> 	<p>Different priority: H 1st, L the last</p> 

2. Nesting blank definition

Blank can be defined with any sketch geometries with only one outer close loop and no intersection and nesting inner loops.



“Blank Margin” offers 2 ways to define: even and uneven margin in 4 directions, users can decide with the “Way” option.

3. Nesting time

Nesting result can be different basing on how long the nesting time is using. Normally the longer the time is, the better the result is. Users can input a suitable time balancing the result and time cost.

4. Nesting result

Nesting result will generate as a big sketch block with each profiles and blanks residing within as sub-block under the same sketch context. Users can activate the resultant sketch block to export as dwg to further process.

User can specify what “Report name” is and where the “Report output” is saved into, and the report provides following info:

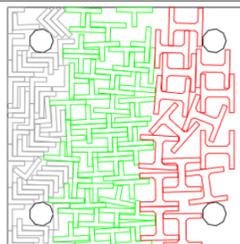
Nesting report

Profile information

name	Scheduled quantity	Area	Amount completed	image	Total area
Profile_6	50	800	50		40000
Profile_7	50	1050	50		52500
Profile_8	50	2371.46	15		35571.9

Blank information

name	Area	Nesting quantity	image	Utilization rate
Blank_2	242146	115		0.528904



→ Where it is

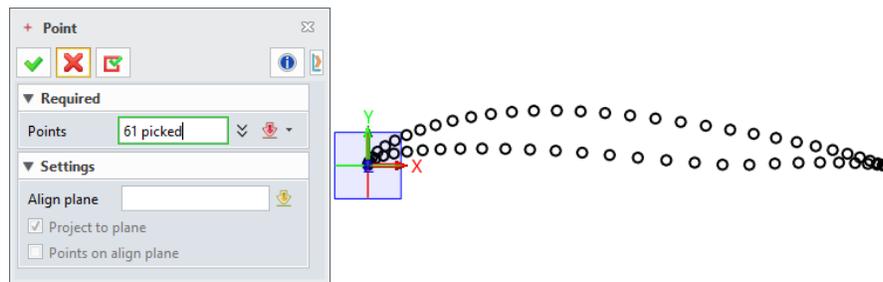
Sketch context > App Ribbon > Nesting

3.2 Wireframe Design

According to feedbacks, a few wireframe enhancements have been done as following:

1. New “From file” option in the point menu to import points from file

“From file” option can read multiple points from an external file one by one and generate corresponding points. This option can be found on the drop-down menu or right-click menu of points field.



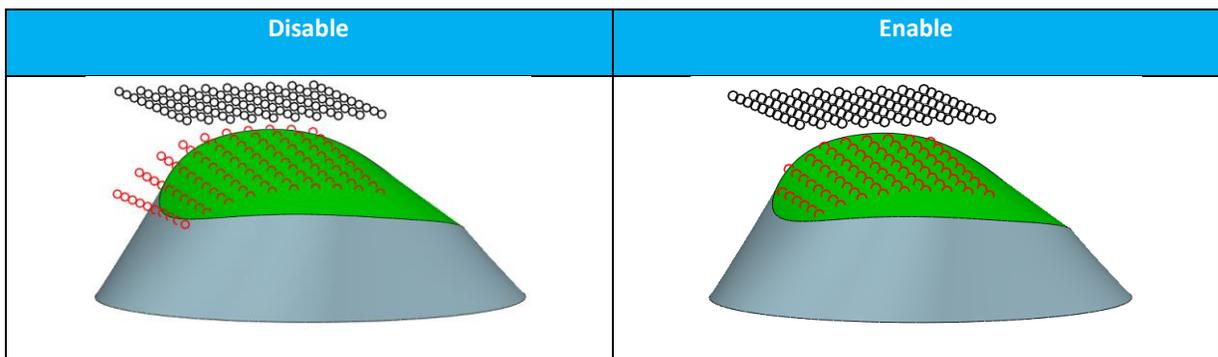
2. New “From file” in “Point Cloud Curve”

“From file” in “Point Cloud Curve” can read out all points from an external file and generate a curve through all those points directly with no manual operation involved.



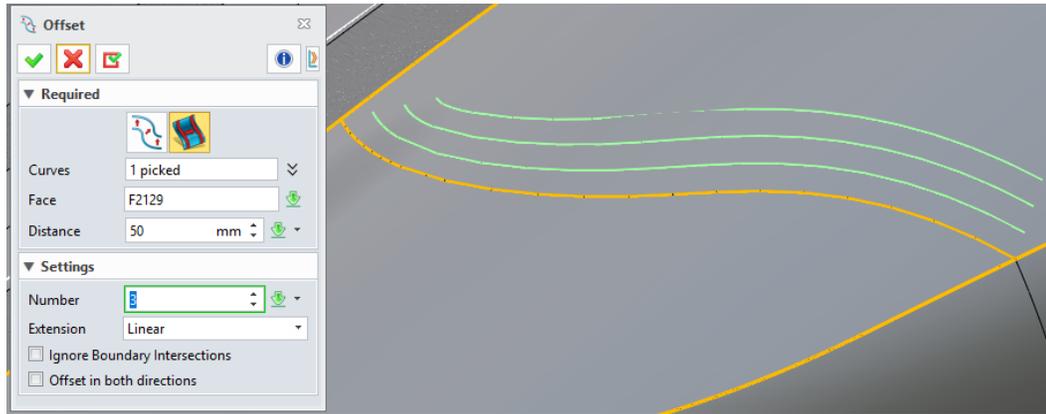
3. New “Trim to face boundaries” option in “Project Points”

This new option can erase all projection point outside of the target face.



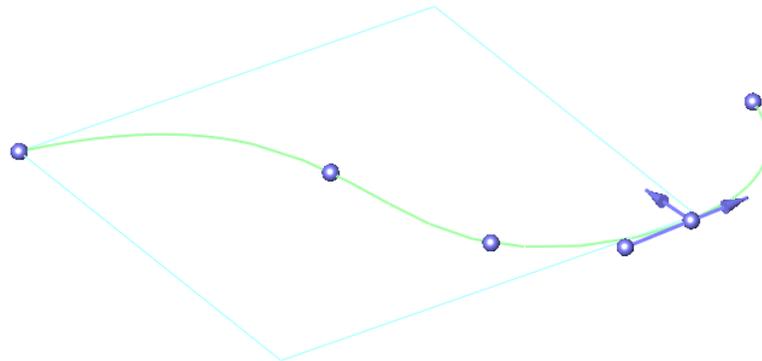
4. New “Number” in “Offset Curves”

This “Number” option can generate multiple curves during one offset operation to improve operation efficiency.



5. New “Auto” in “Align Plane”

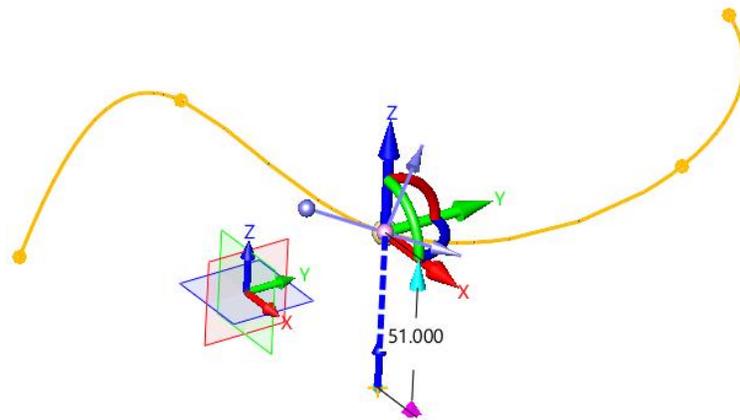
“Through Point Curve” has added an “Auto” in “Align type” to generate a temporary plane basing on existing input points to constrain point location. The auto option is enabled, new input point or editing point will constrain on the temporary plane.



6. More new option in “Modify Curve”

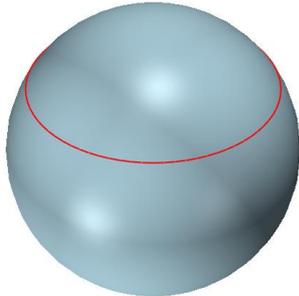
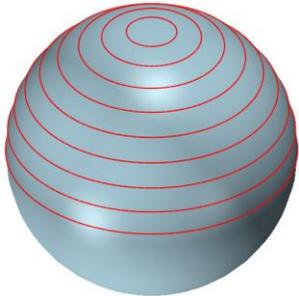
“Modify Curve” also added “Align plane” option with “Auto” as above to constrain point movement within the auto or picked plane.

New “Use orientation tool” option can add a movement handle on the picked point on curve to guide point relocation.



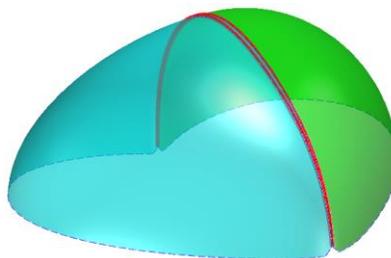
7. New “Isocline Curve”

“Isocline Curve” can generate curves on the given face, whose face normal at points of those curves are at the given angle to the pointed direction. Users can generate curves at one single angle, also can have curves at multiple angles which is evenly divided at given angle range.

Single Angle	Angle Range
	

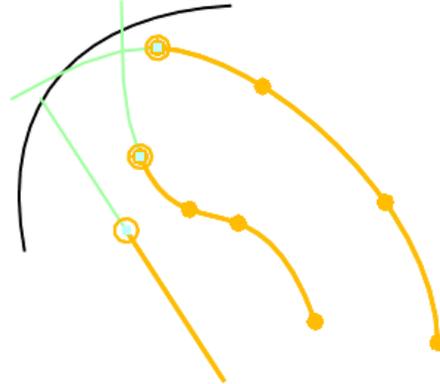
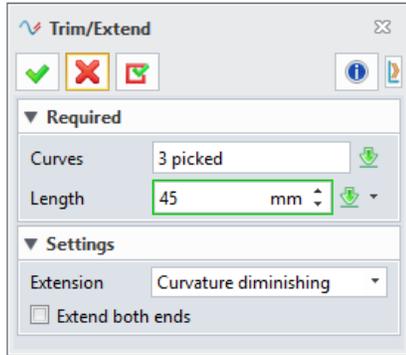
8. Better “Intersection curve” on tangent faces

“Intersection curve” now have more curves on tangent faces since Boolean algorithm is greatly enhanced on tangent faces intersection.



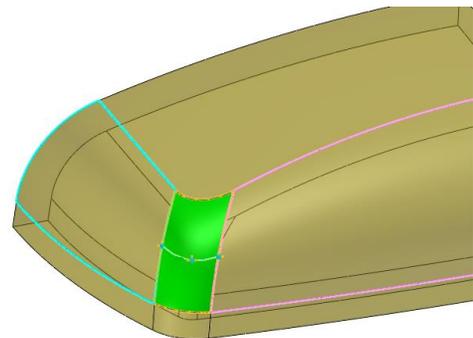
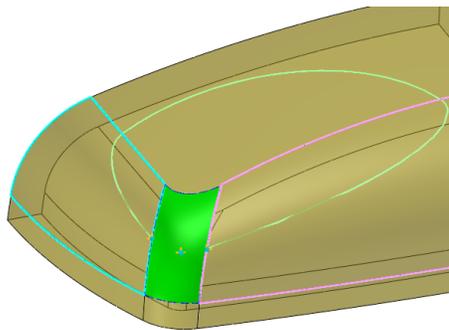
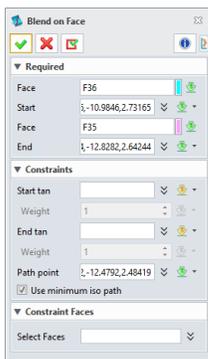
9. “Trim/Extend” supports multiple curves

“Trim/Extend” can work on multiple curves at one operation.



10. New “Constraint Faces” on “Blend on Face”

“Constraint Faces” can guide how the blend curve will pass through, especially when the curve needs to cross faces that are from different shapes. For example, following green face is individual shape, and the blend curve is expected to pass through.



3.3 Part Design

3.3.1 ★Modeling History

3.3.1.1 *New “Quick Rollback” to Accelerate Redefinition*

“Quick Rollback” is a new functionality added to speed up the history rollback during redefinition or rollback. After enabling “Configuration > Part > General > Backup data for history quick rollback” option, there is a cache file to save major outcome data of each modeling feature. When a feature redefinition or rollback takes place, corresponding feature data will be fetched to restore the modeling field as it was created avoiding the regeneration of modeling features before the redefining feature, so that no more waiting time is needed.

- 1. Quick rollback relies on cache data, so only new modeling history or regenerated history after this function is enabled can have such acceleration.**
- 2. Features related to component can't support quick rollback, so their redefinition will work as before.**

Component features like Insert, movement, constrain of the old assembly file, or assembly hole, assembly cut, etc.

When component features are included in the feature rollback list, since there are no rollback data for those features, a full regeneration will take place as previous version to restore the modeling field.

- 3. Rollback data file is saved on the specified folder, independent with the model Z3 file.**

Since the rollback data is not saved into the Z3 file, if the model Z3 file is moved to different PC, rollback data needs to generate again through a regeneration.

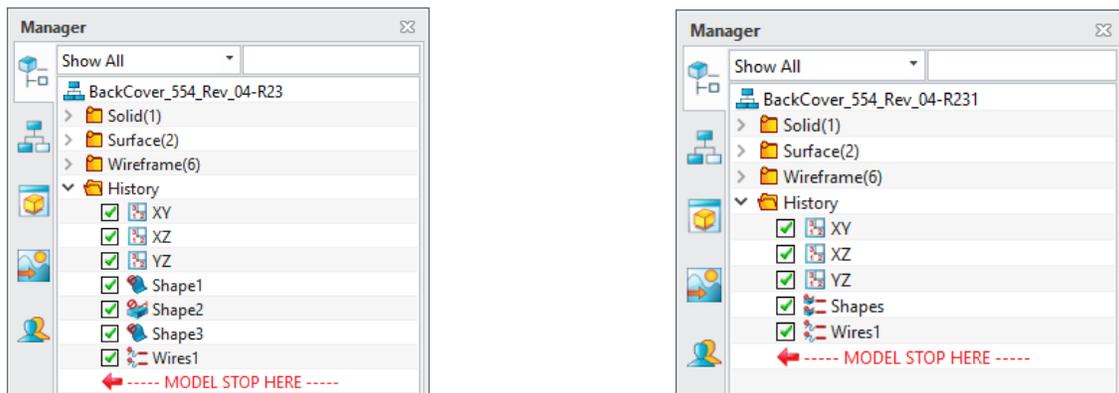
Users can specify which folder the rollback data should be into through “Configuration > File > Quick rollback cache file”. And this file has a max size limit which users can set up your own size through “Configuration > Part > General > Backup data for history quick rollback > Max folder size”.

When the folder size is exceeded, the oldest cache file will be deleted to free some space for the new cache file.

3.3.1.2 New Explicit Shape/Wireframe Feature Involved in History Feature

After import, there are corresponding feature nodes on the history tree for each shape and wireframe entities, so that users can manipulate those import geometries as normal feature, such as, deleting shape feature to erase explicit shape permanently to reduce the file size.

Only one explicit wireframe feature for all imported wireframe entities, and the imported shapes not only can have one feature for each, but also can use one feature for all with “Configuration > Part > General > Enable multi-shell explicit shape features” option.

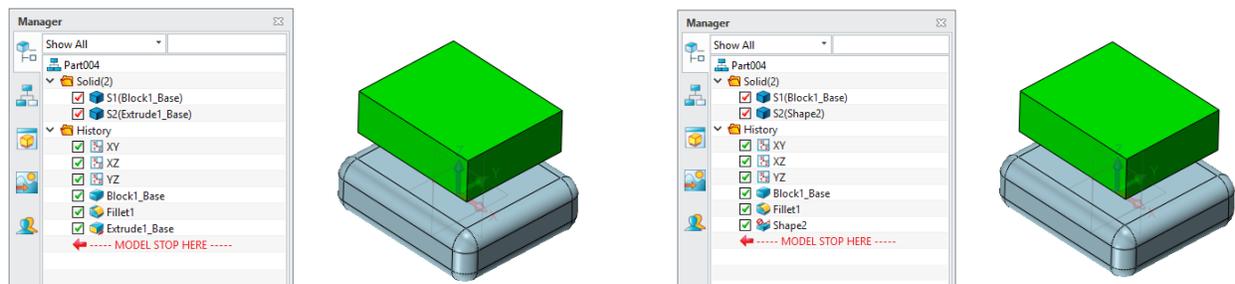


Same explicit features will be generated when “Encapsulate” history. During encapsulation, following features will be ignored and remains:

Datum, sketch, curve list, assembly feature

Same explicit features also will be generated after Defeature operation, and thanks to that, file size will be downsized 50% comparing with previous version.

Defeature has improved to unlink more dependent features. For example, after Extrude a face of a block generates a shape, to defeature this new shape will not affect the block feature.



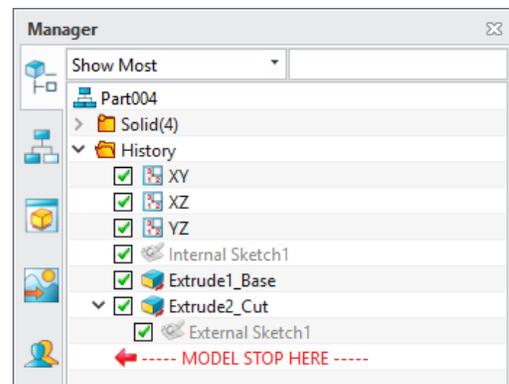
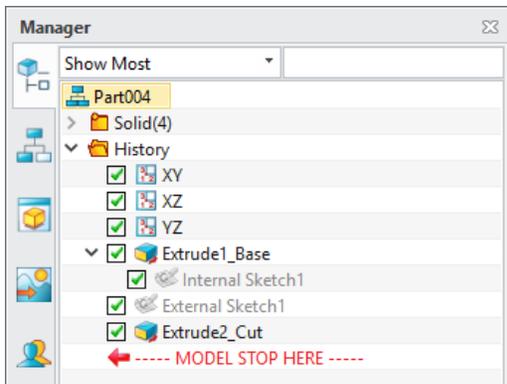
3.3.1.3 New "Internal Sketch"

Internal sketch is the one created on-place during another modeling feature like Extrude, and it will be registered under the modeling feature showing the ownership. Internal sketch only can be used by its parent feature.

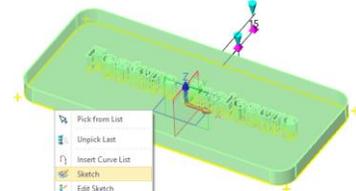
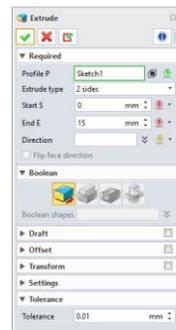
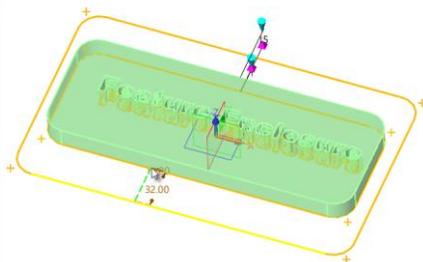
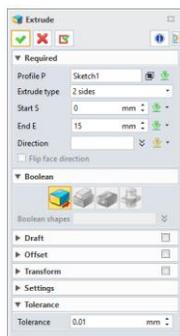
Internal sketch can turn into external through "Make External" from its right-click menu.

If an external sketch is only used by one feature, it can turn into internal with the "Make Internal" from its right-click menu.

When a modeling feature is deleted, its internal sketch will be deleted too.



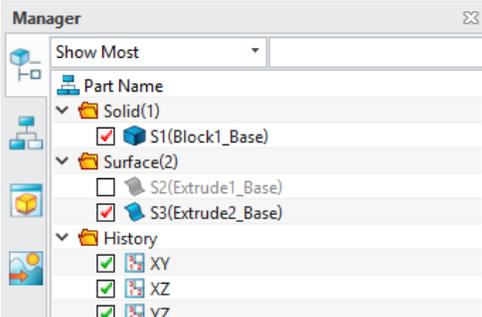
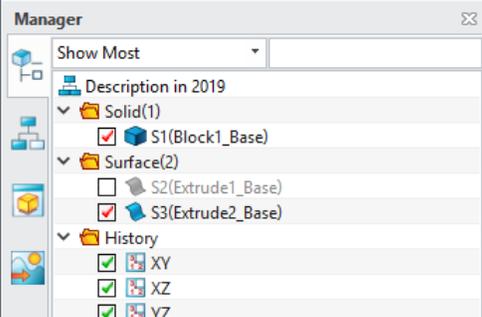
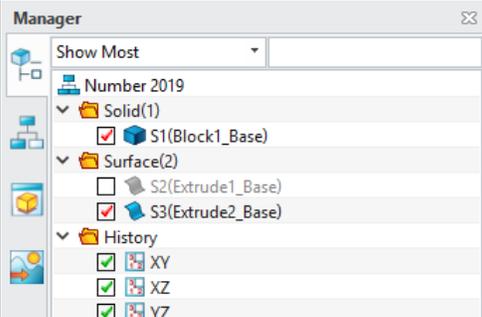
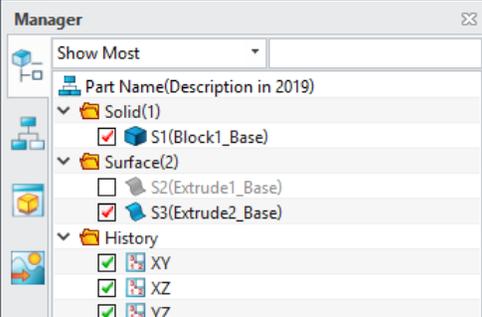
When redefining a modeling feature, its internal sketch can edit on-place, such as moving its entities with dragging, even can go into sketch context to furthermore edit with the right-click menu without interrupting the modeling feature.

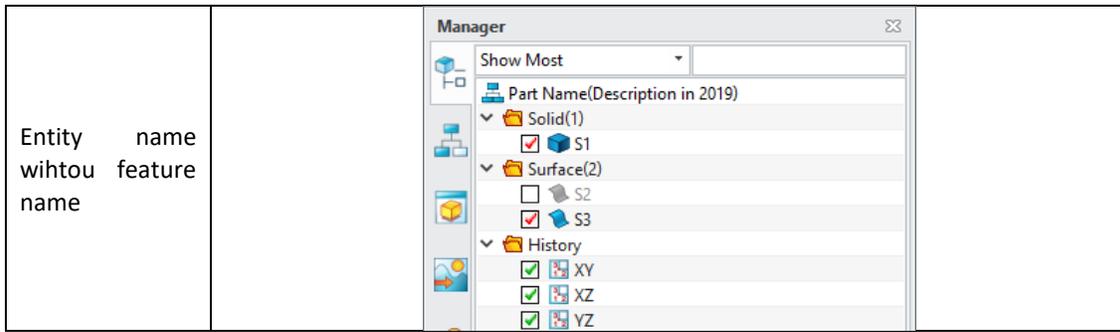


3.3.1.4 Other Improvement in History

1. New “Configuration > Part > Folder display settings > Part name & Entity name” settings

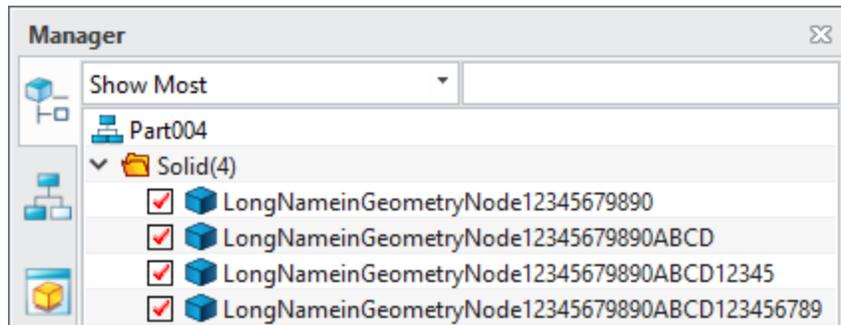
These 2 settings are to control what name of the root node and geometry node will be used to display.

<p>Display name</p> <p>part</p>		 <p>Manager</p> <p>Show Most</p> <p>Part Name</p> <ul style="list-style-type: none"> Solid(1) <ul style="list-style-type: none"> <input checked="" type="checkbox"/> S1(Block1_Base) Surface(2) <ul style="list-style-type: none"> <input type="checkbox"/> S2(Extrude1_Base) <input checked="" type="checkbox"/> S3(Extrude2_Base) History <ul style="list-style-type: none"> <input checked="" type="checkbox"/> XY <input checked="" type="checkbox"/> XZ <input checked="" type="checkbox"/> YZ 	
<p>Display description</p>		 <p>Manager</p> <p>Show Most</p> <p>Description in 2019</p> <ul style="list-style-type: none"> Solid(1) <ul style="list-style-type: none"> <input checked="" type="checkbox"/> S1(Block1_Base) Surface(2) <ul style="list-style-type: none"> <input type="checkbox"/> S2(Extrude1_Base) <input checked="" type="checkbox"/> S3(Extrude2_Base) History <ul style="list-style-type: none"> <input checked="" type="checkbox"/> XY <input checked="" type="checkbox"/> XZ <input checked="" type="checkbox"/> YZ 	
<p>Display name</p>		 <p>Manager</p> <p>Show Most</p> <p>Number 2019</p> <ul style="list-style-type: none"> Solid(1) <ul style="list-style-type: none"> <input checked="" type="checkbox"/> S1(Block1_Base) Surface(2) <ul style="list-style-type: none"> <input type="checkbox"/> S2(Extrude1_Base) <input checked="" type="checkbox"/> S3(Extrude2_Base) History <ul style="list-style-type: none"> <input checked="" type="checkbox"/> XY <input checked="" type="checkbox"/> XZ <input checked="" type="checkbox"/> YZ 	
<p>Display name & description</p>		 <p>Manager</p> <p>Show Most</p> <p>Part Name(Description in 2019)</p> <ul style="list-style-type: none"> Solid(1) <ul style="list-style-type: none"> <input checked="" type="checkbox"/> S1(Block1_Base) Surface(2) <ul style="list-style-type: none"> <input type="checkbox"/> S2(Extrude1_Base) <input checked="" type="checkbox"/> S3(Extrude2_Base) History <ul style="list-style-type: none"> <input checked="" type="checkbox"/> XY <input checked="" type="checkbox"/> XZ <input checked="" type="checkbox"/> YZ 	



2. Long name support in geometry node

Geometry node name now can be up to 256 characters to cover different language usage.



3. New “Toggle Entity Transparency” on the right-click menu of the shape node

This command is to change picked shape to predefined transparency which is set by “Configuration > Display > Numeric settings > Transparency used in On/Off”.

4. Faster Feature Dependencies analysis

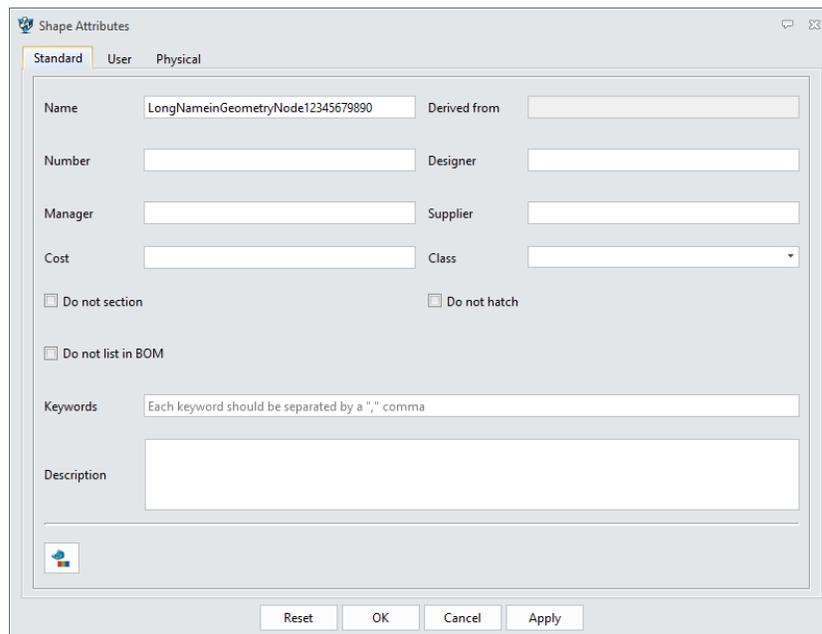
Feature dependencies analysis is optimized to respond instantly which is much faster than previous version.

3.3.2 ★New Shape Attribute

Shape now can have its own attributes which are very much the same as the part attributes. When a multiple-shape part is used to simulate assembly design, it's very necessary to set up shape own attribute, since one shape works as an individual component.

1. Almost the same shape attributes as part

Similar with part attributes, shape can have standard, custom and physical attributes.



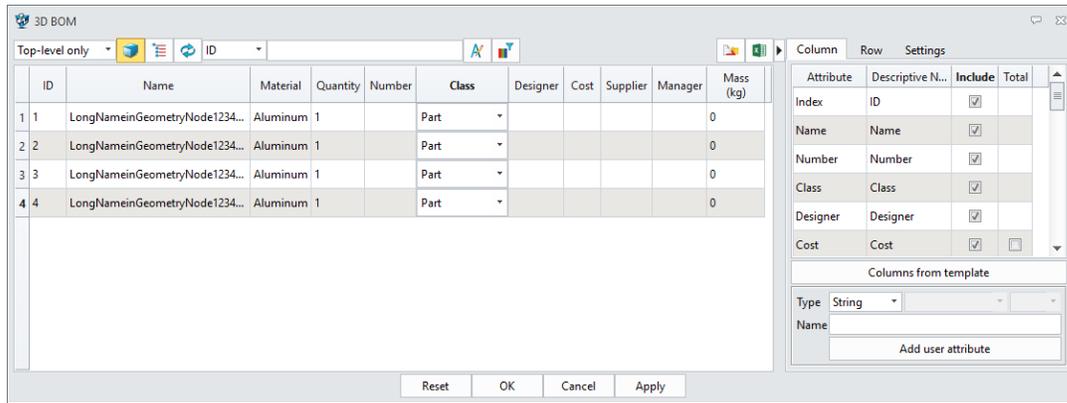
2. Attributes inheriting from part

Shape standard and custom attributes can directly inherit from corresponding part attributes to avoid duplicate work.

There is a “Inherit Attributes” command on the shape’s right-click menu to copy attribute from picked shape or component.

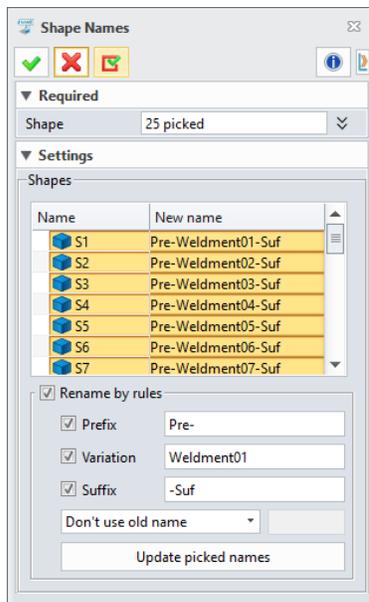
3. Attribute batch edit

The “3D BOM” on Assembly Ribbon can list out all shapes and their attributes, and allow users to edit shapes’ attributes on the spread sheet.



4. Batch renames

There is a “Shape Name” command on the right-click menu when multiple shapes are picked.



- **Pick shapes to rename**
- **Shape list**
- **Rename by rules**

All picked shapes will be listed and can be renamed when double-click the New name cell. Ctrl/Shift pick is supported.

Renaming by rules only work picked shapes above. Users can add prefix or suffix into the old name to form a new name, or just discard the old name. if the names follow some pattern, the Variation field is offered to do such thing, such as increasing the last character, A01, A02...

➔ **Where it is**

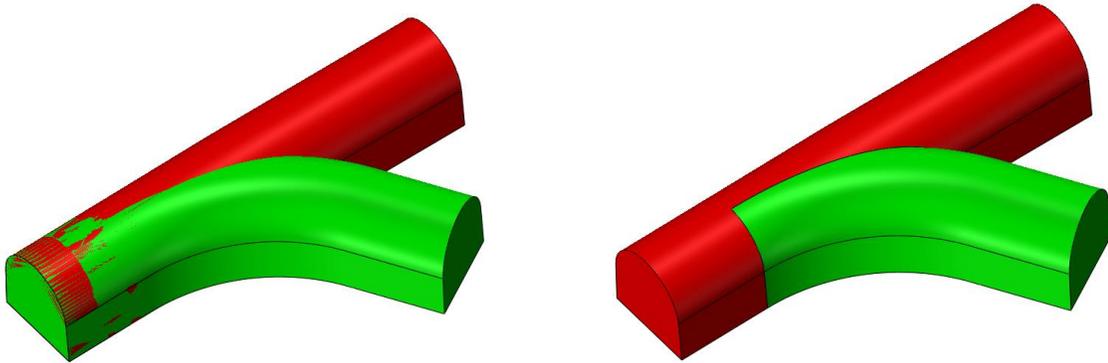
Part/Assembly context > right-click on shape node on history tree > Shape Attribute

Part/Assembly context > right-click on shape nodes on history tree > Shape Names

3.3.3 ★Boolean Improvements

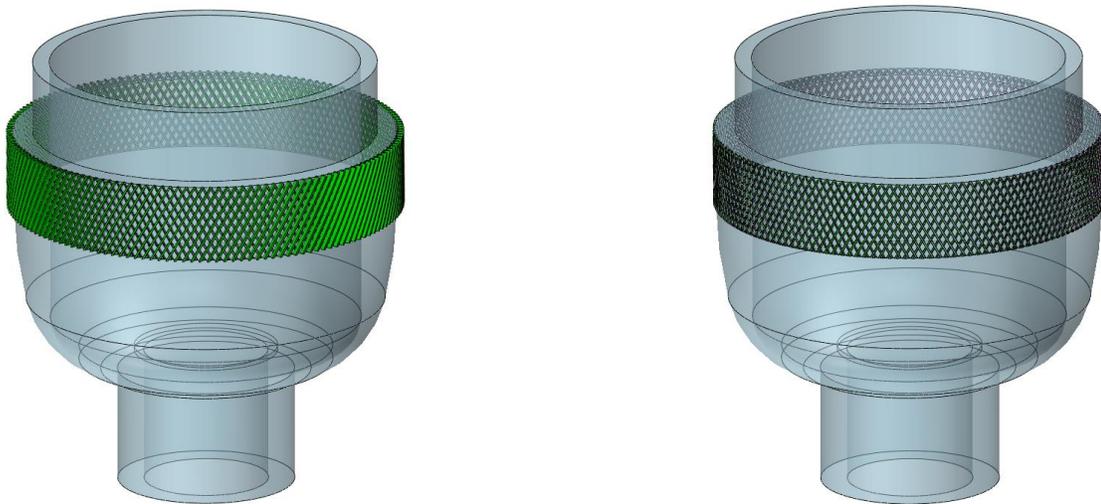
Boolean has upgraded its algorithm to better handle the tangent face intersection and separated into 3 individual commands basing on feedback.

1. Better result on tangent surface/surface intersection



2. More robust Boolean

Benefiting from the surface/surface intersection improvement, general Boolean operation is more robust with expected outcome.



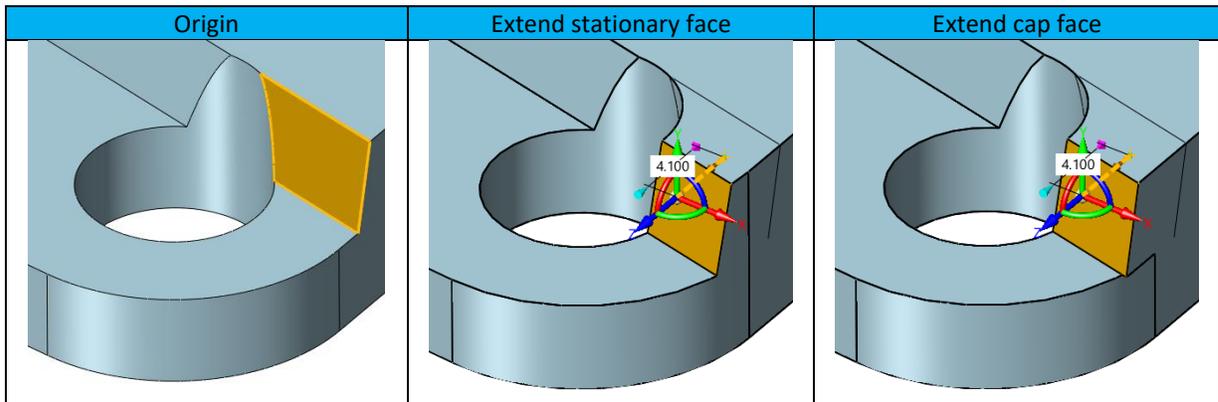
3. 3 individual Boolean commands

Basing on feedback, the old Combine command has separated into 3 individual commands: Add, Subtract and Intersect, so that users can define shortcut for each Boolean command.

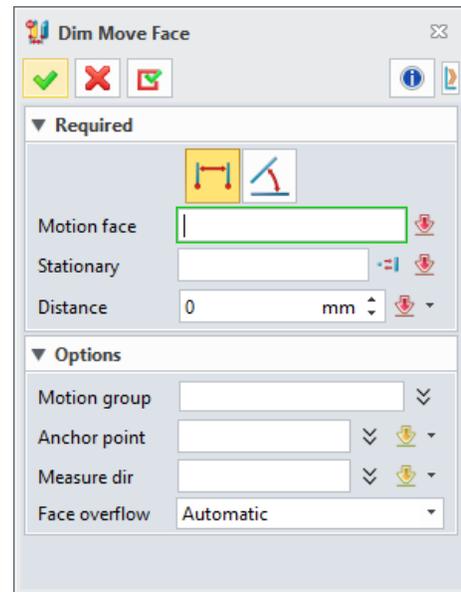
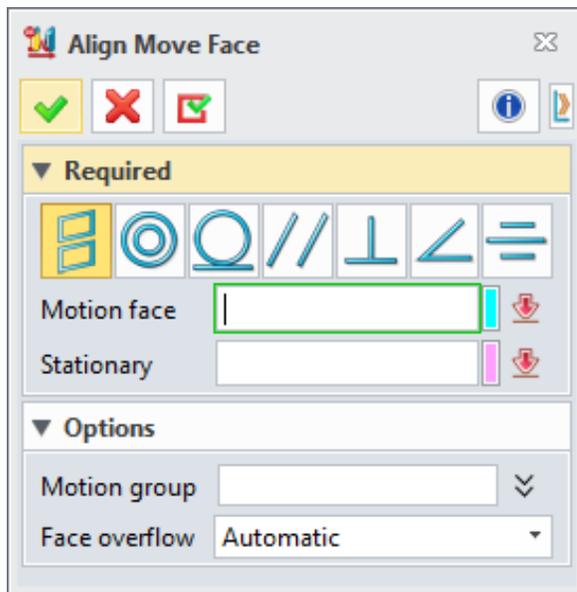
3.3.4 ★Direct Edit Enhancements

Direct Edit focuses on improving its correctness and robustness and expands the overflow behavior to more commands. Here are the details:

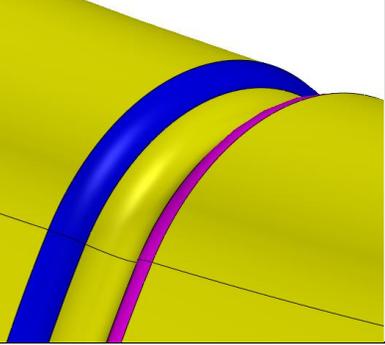
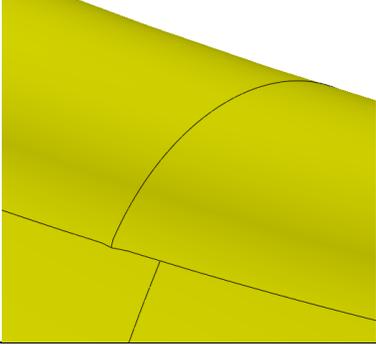
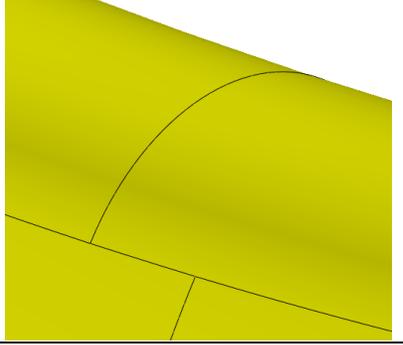
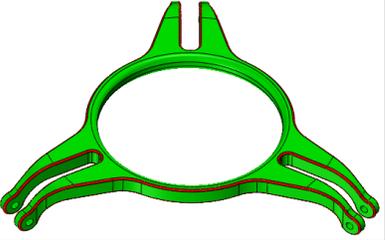
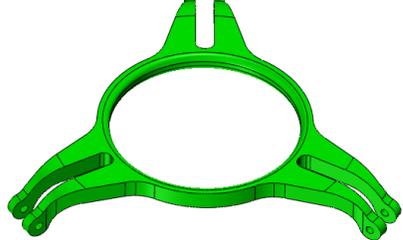
1. Modified overflow behavior to improve its face region recognition for better closure



2. Overflow is expanded to “Align Move Face” and “Dim Move Face” commands

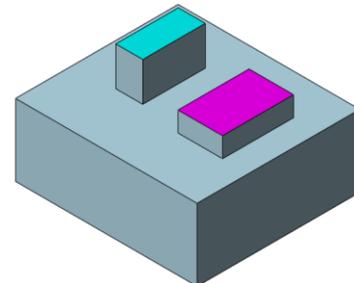
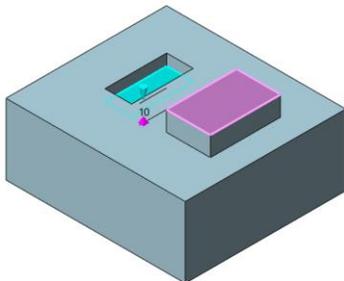


3. Upgraded “Simply” to optimize its face extension and better handle fillet faces

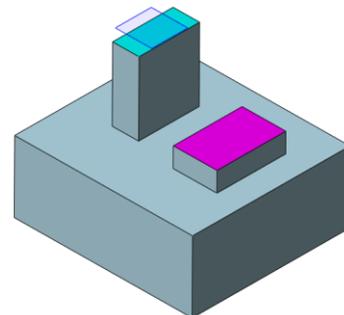
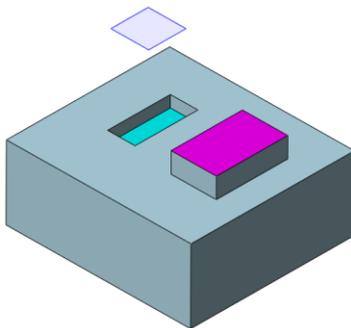
Origin	Previous	2019
		
	<p data-bbox="800 726 865 751">Failed</p>	

4. Better “Replace”

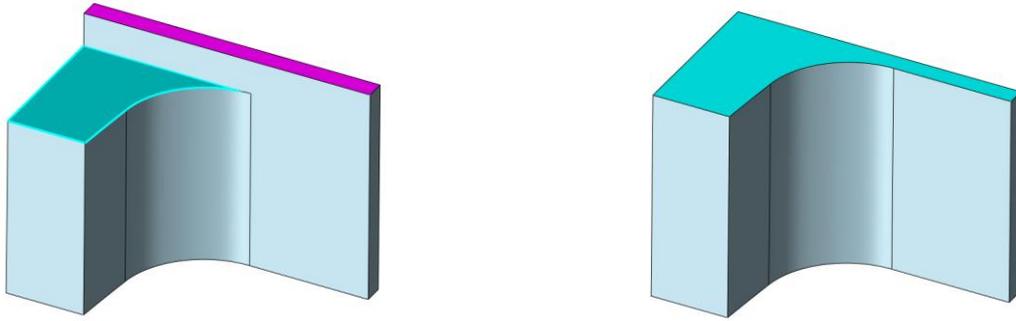
- New “Offset” field to offset the tool face at given distance first, then finish the replacement.



- Datum plane can be used as the tool face

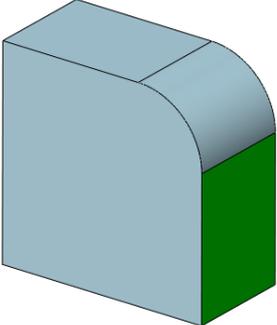
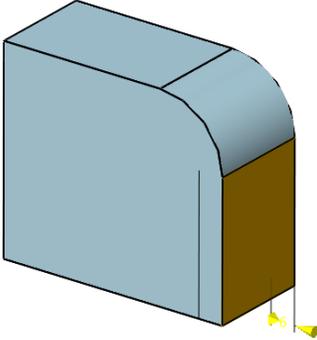
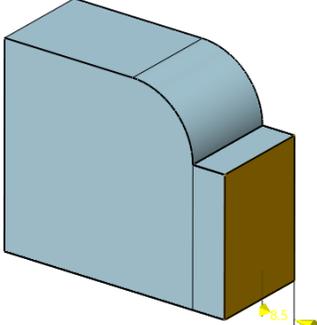


- More robust Replace on faces with tolerant edges

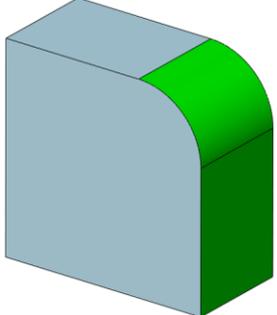
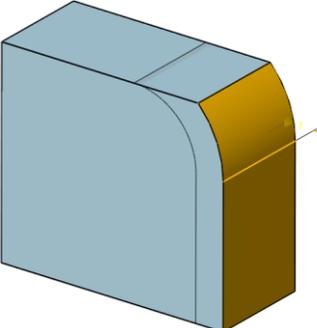
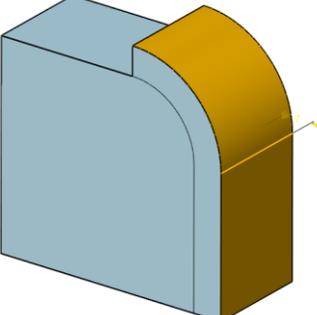


5. New “Side faces” on “DE Face Offset” to generate new faces to close

This new “Side faces” can give a different result on faces who are tangent to adjacent faces, and meanwhile empower “DE Face Offset” to cover more cases.

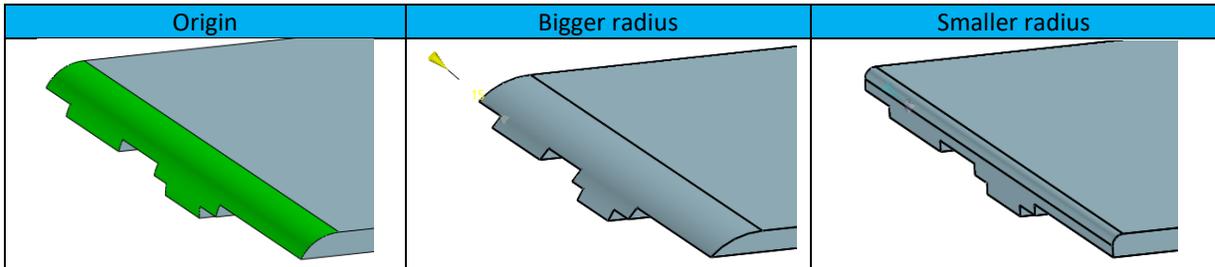
Original	No side faces	With side faces
		

Different results when fillet faces are offset together.

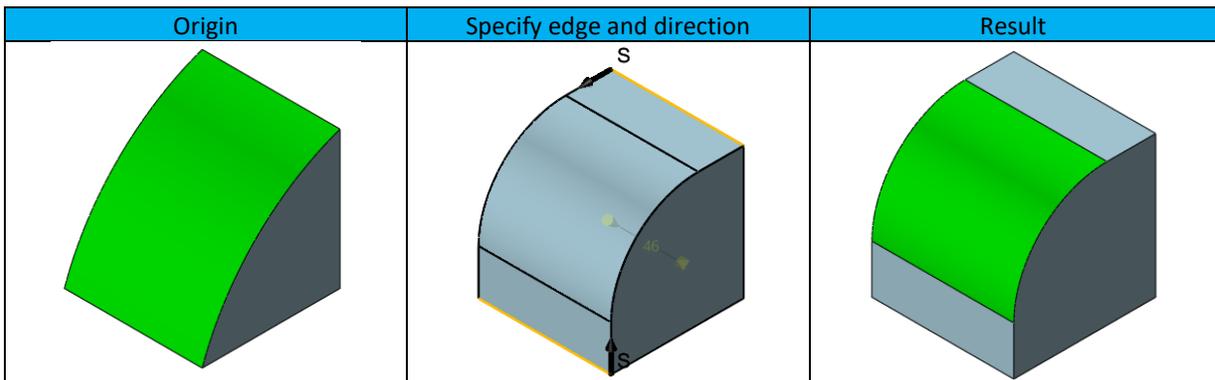
Original	No side faces	With side faces
		

6. “Modify Fillet” works better on fillets which are not tangent to their support faces

The new “Hold fillet to edge” option works on fillet face which is not tangent to its support faces when the fillet radius is modified to be bigger.

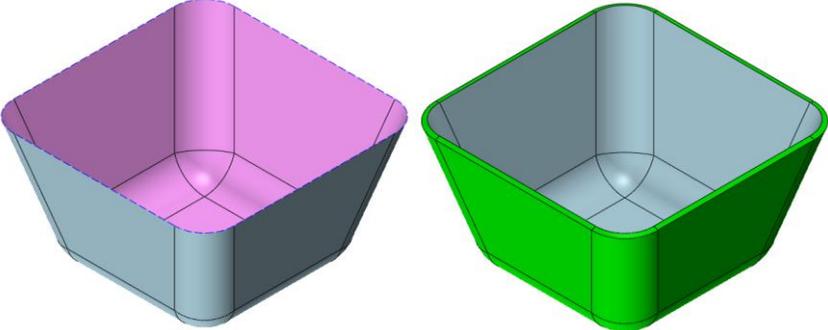
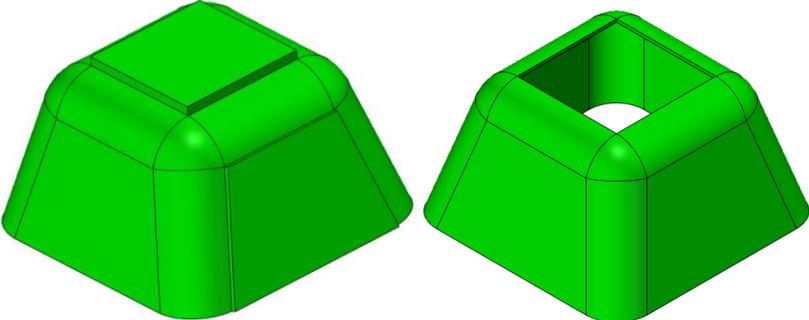
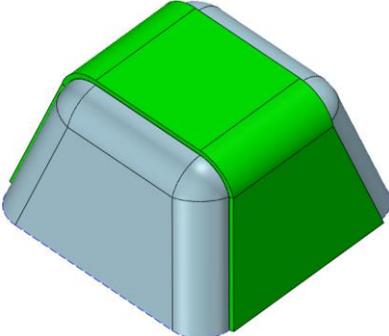
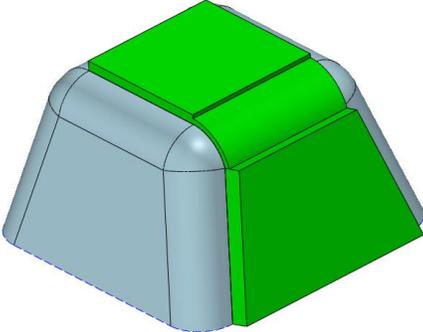


When un-tangent fillet face becomes smaller enough to need both support side faces, the new “Close strategy” can guide how those support side faces are constructed. If “Selected” is picked, users need to pick which edge will generate the support side face and what direction the side face will be in.



3.3.5 New “Thickness”

“Thickness” can shell an open shape or picked connected faces into a solid shape.

<p>Evenly thicken on open shape</p>	
<p>Un-even thickness, even with an open face</p>	
<p>Thicken connected faces evenly</p>	
<p>Thicken connected faces un-evenly</p>	

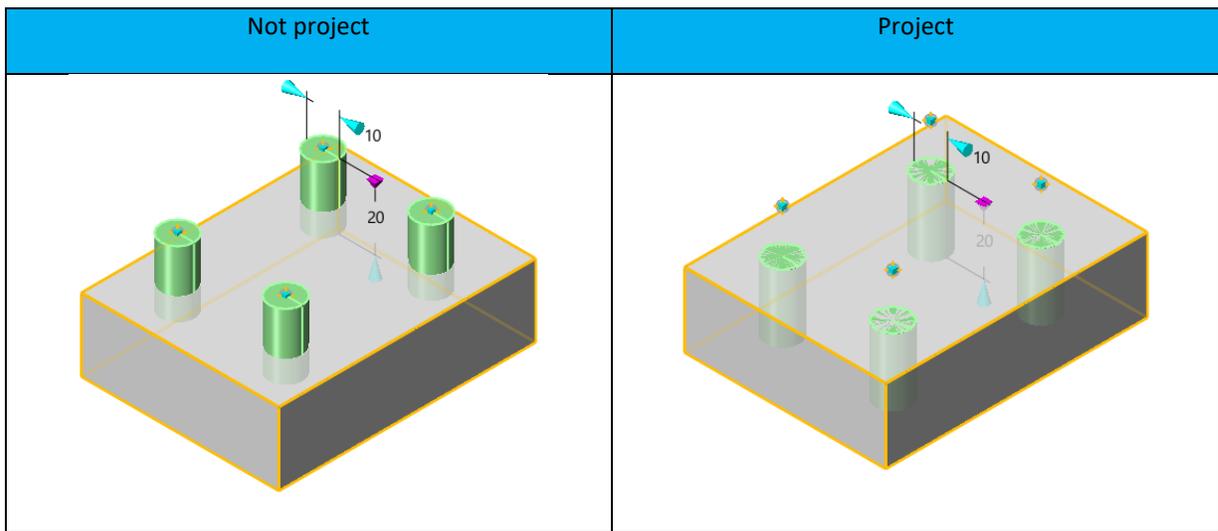
→ Where it is

Part/Assembly context > Shape Ribbon > Edit Shape > Thicken

3.3.6 “Hole” Adjustments

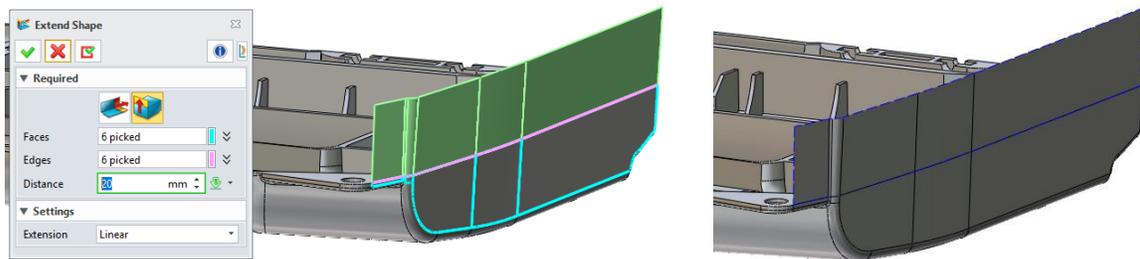
Hole’s location can create an internal sketch on-place through Location field’s drop-down menu to define where the holes will be, and modify this internal sketch any time during hole feature redefinition, such as entering sketch context to add more constraints or dimensions. The external sketch can still be selected from the history tree.

New “Project location point” option can control whether the hole location should be projected onto the location plane.



3.3.7 “Extend Shape” Improvement

Since “Extend Open Shape” has added support to work on the solid shape, its name has changed into “Extend Shape”. The new “Extend General Shape” can work on multiple edges of one face or different edges from various faces on one operation. Faces extent from solid shapes will be individual shapes to avoid breaking the solid shape.

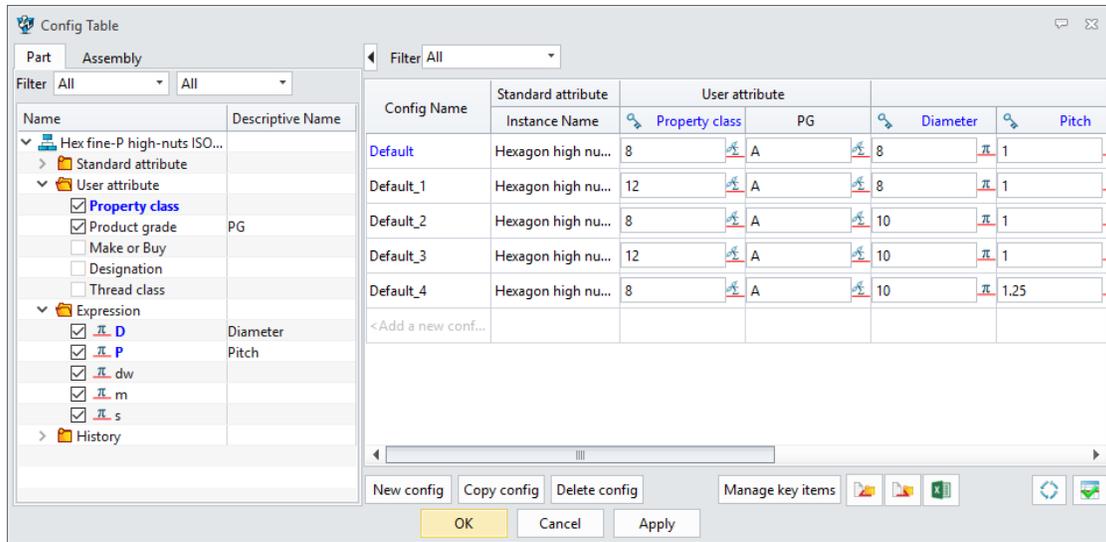


3.3.8 Changes in “Config Table”

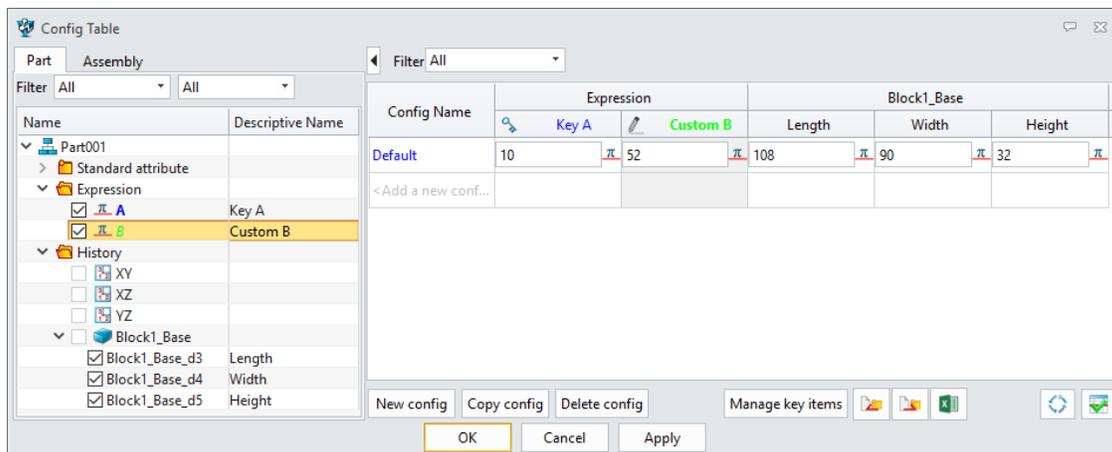
“Config Table” has added a few new functionalities and tweaked some workflow to facilitate users as following:

1. Two rows for the table header

There is another extra row in the header to show which category each configured item belongs to.



2. Descriptive name can work on every configured item



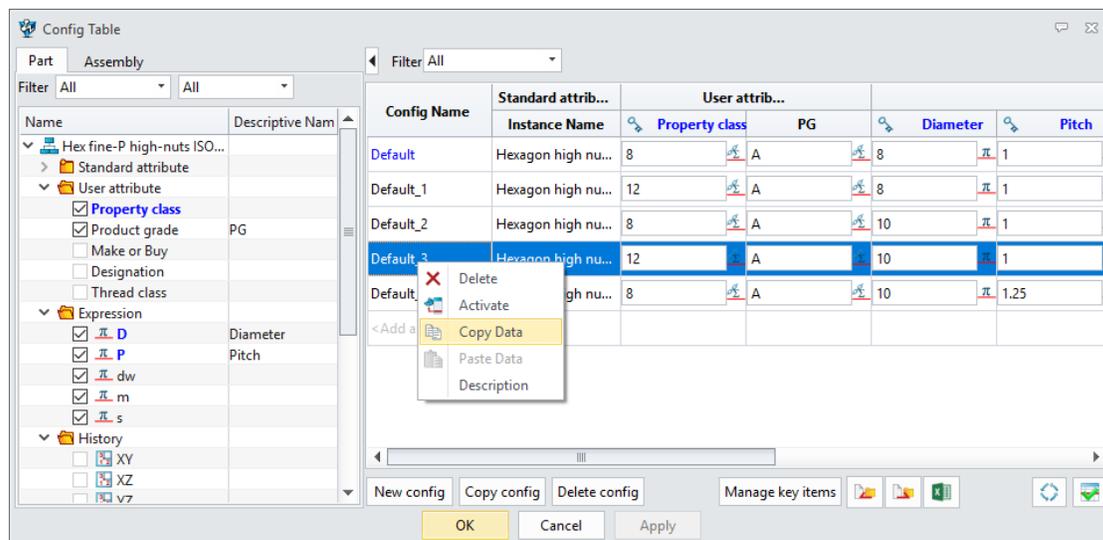
3. Optimized sequence of each configured items

The sequence of each configured items will be same as the left tree so that they can be much clearer to find.

4. New “Add a new config” row

Users can input a new config name into the last row “Add a new config” of the sheet to create a new config directly.

5. New Copy/Paste Data



➔ **Where it is**

Part/Assembly context > Tools Ribbon > Insert > Config Table

Part/Assembly context > Right-click on blank area of the history tree > Config Table

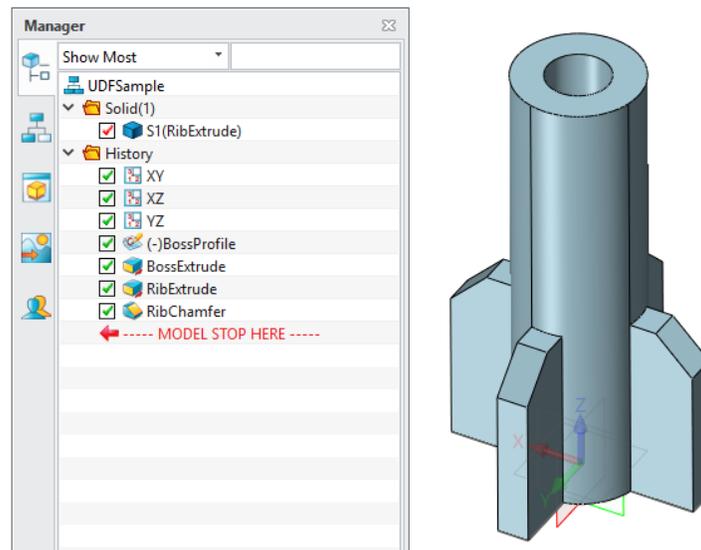
Part/Assembly context > Right-click on blank area of the graphic area > Config Table

3.3.9 ★New “User Define Feature”

“User Define Feature” can gather several features of a common model structure and make them into one UDF feature, then users can re-use this UDF feature on different models to create same structure without re-doing every feature to save more time. Here is a simple run-through.

1. Get the modeling features about to re-use ready

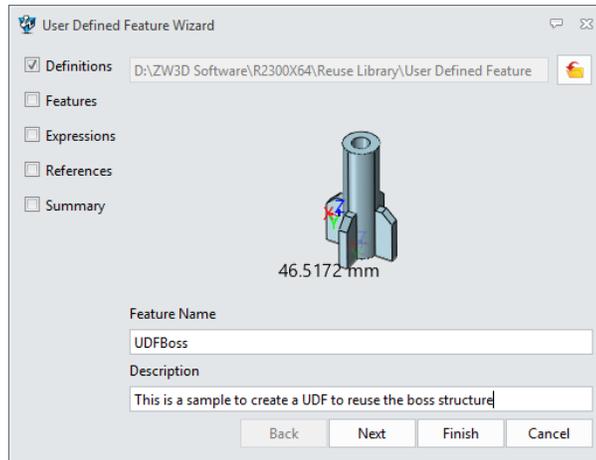
The following structure is an assembly boss consisting of one Sketch, 2 Extrusion and a Chamfer.



2. Start “Insert menu > Features > User Defined Feature > User Defined Feature Wizard” to define the UDF name and where it is about to save

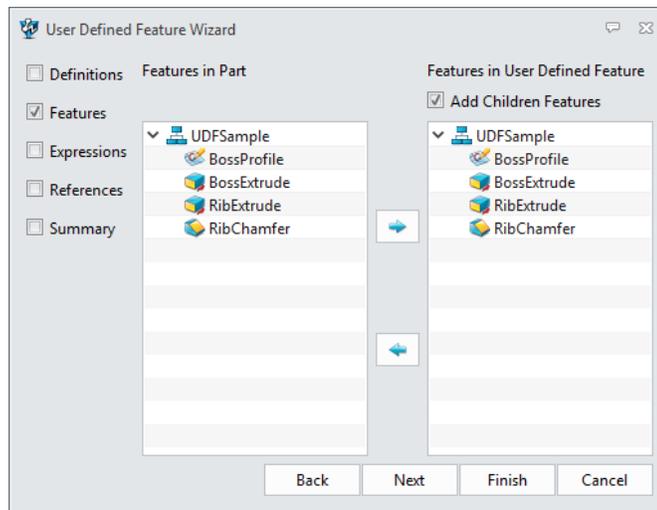
By default, the UDF is saved into UDF folder under the installation directory. Users can specify another location.

New UDF should have unique name. If existing UDF is expected to overwrite, users need to delete it manually under the UDF folder.



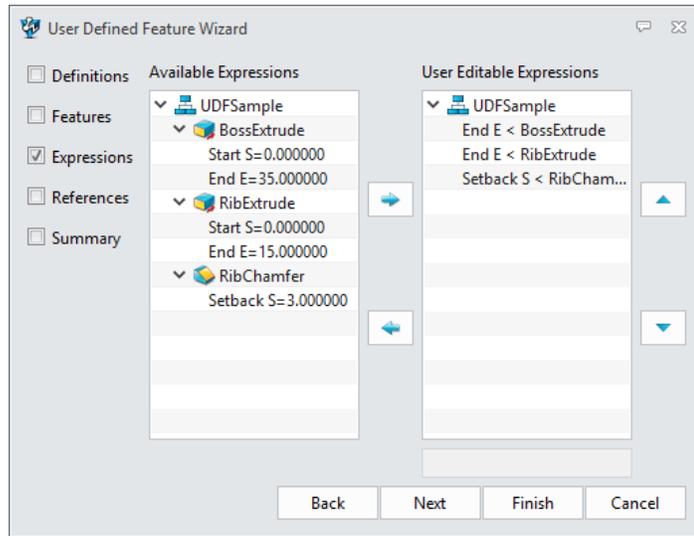
3. Define which features will be included in UDF

All modeling features will be listed under “Features in Part”. If the features are greyed out, these features can’t be picked into make UDF.



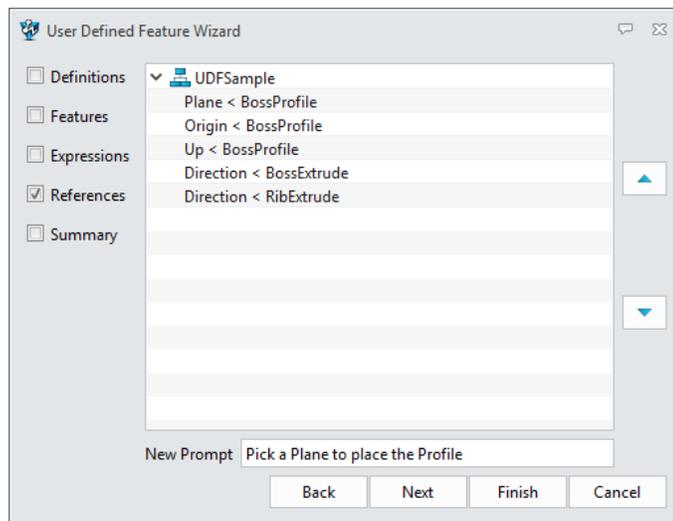
4. Define which feature parameters need to re-input new value during this UDF insertion

“Available Expression” list extracts all the parameters from the picked features on previous step to allow users to pick and put into “User Editable Expression” list, which lists all parameters that need to re-input during this UDF insertion. The feature sequence inside the “User Editable Expression” is the same sequence users input new value during UDF insertion. Users can re-order them through the up and down buttons.

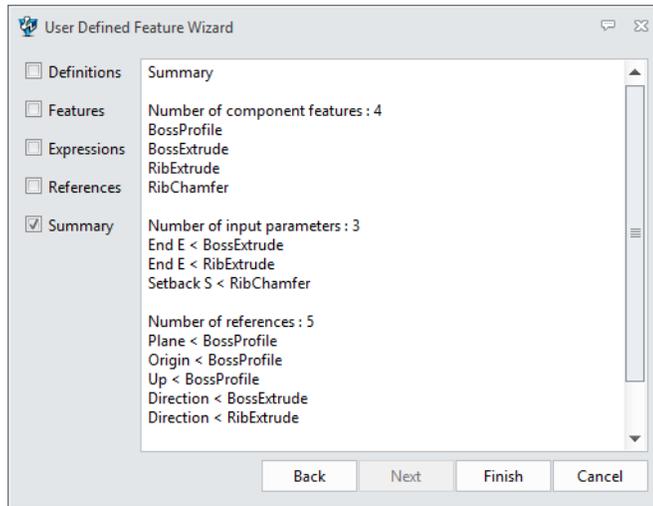


5. Define all references the UDF features are using

Inside the “References” tab, all reference the UDF features picked on previous step will be listed. Users can re-order their sequences and prompt so that it will be easier and more instructive for other users to understand and work during the UDF insertion.

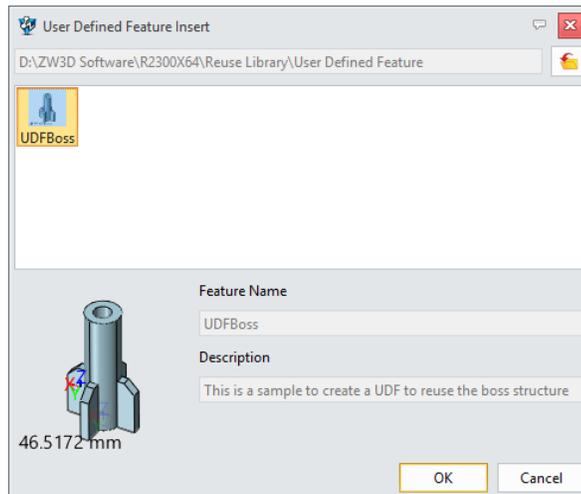


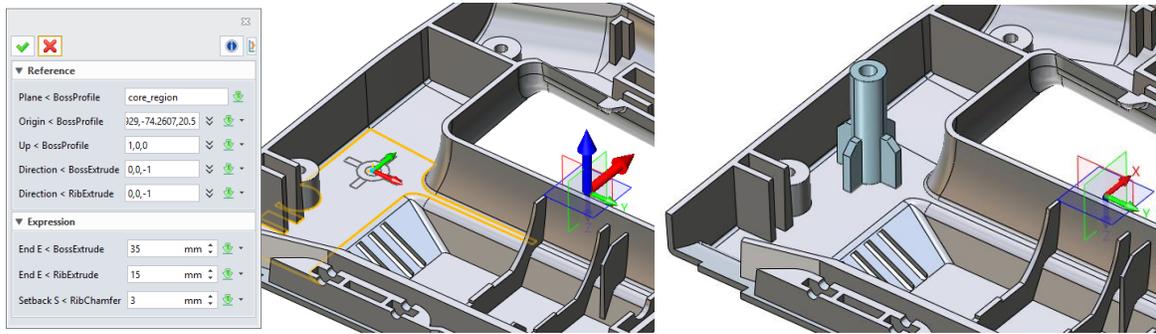
6. Recheck all info on “Summary” Tab



7. Use “Insert menu > Features > User Defined Feature > Insert User Defined Feature” to insert the UDF just created above

All defined UDF will be listed on the Insert User Defined Feature” form. Pick the UDF we just defined above, then re-input each field for each Reference and Expression, and finish the insertion. If default value is preferred for some field, users can click on middle-button of the mouse.





→ Where it is

Part/Assembly context > Insert menu > Features > User Defined Feature > User Defined Feature Wizard

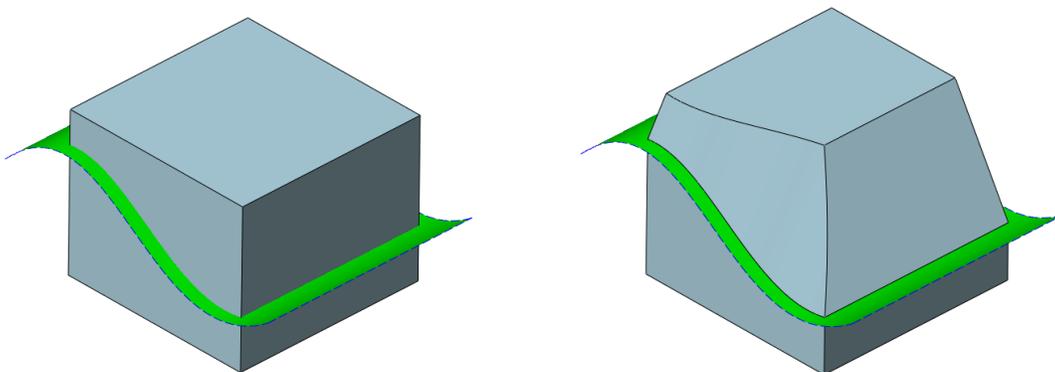
Part/Assembly context > Insert menu > Features > User Defined Feature > Insert User Defined Feature

3.3.10 Misc.

Basing on feedback, several small twists have been done as following:

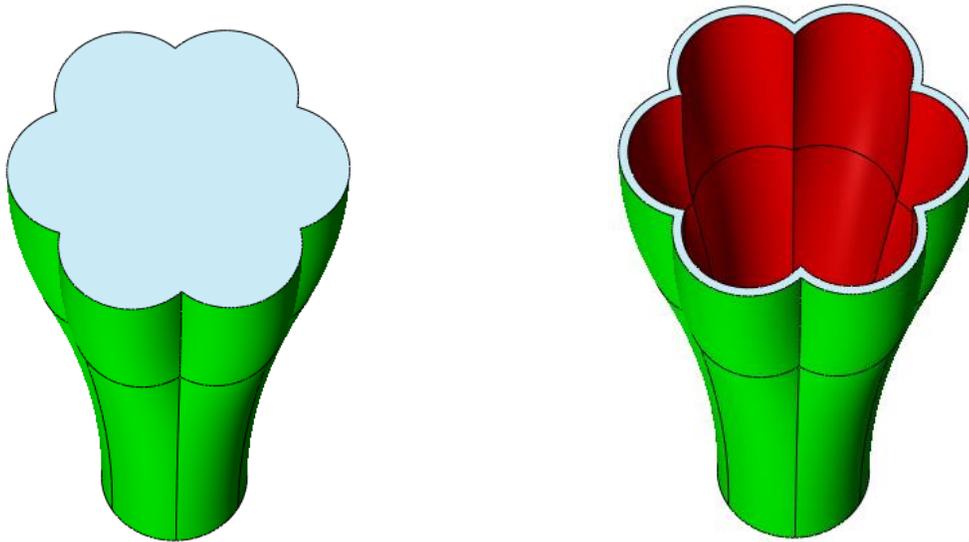
1. Surface can be picked as entity to draft about

If a surface is picked as entity to draft about, faces to draft will be split around the intersection curve with the picked surface and drafted from the new intersection edges.



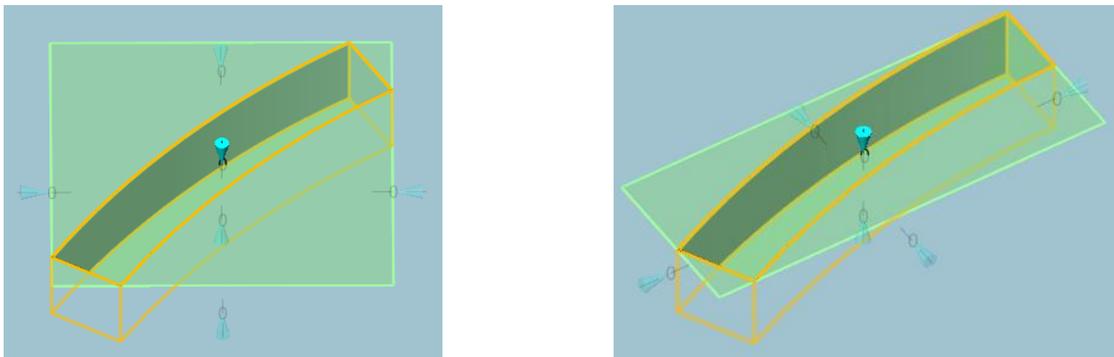
2. New “Keep primitive faces” in “Shell”

The option can primitive faces remaining as primitive after Shell. For example, the red faces generated from the green torus faces after shell will remain as torus.



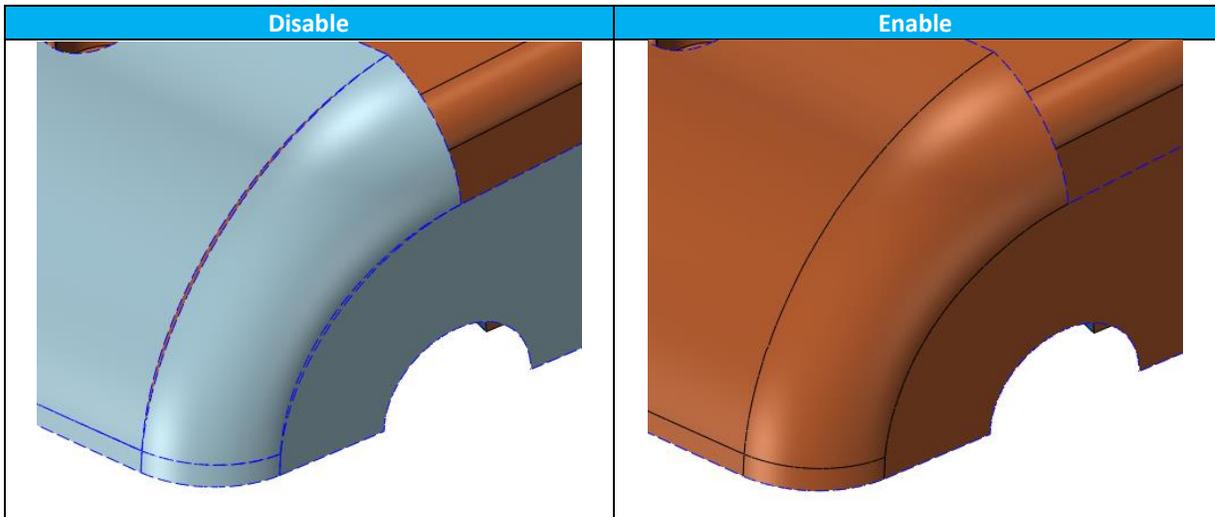
3. “Plane” field in “Stock” can re-pick selected planar face

“Plane” field can use the selected planar face to orient the stock.



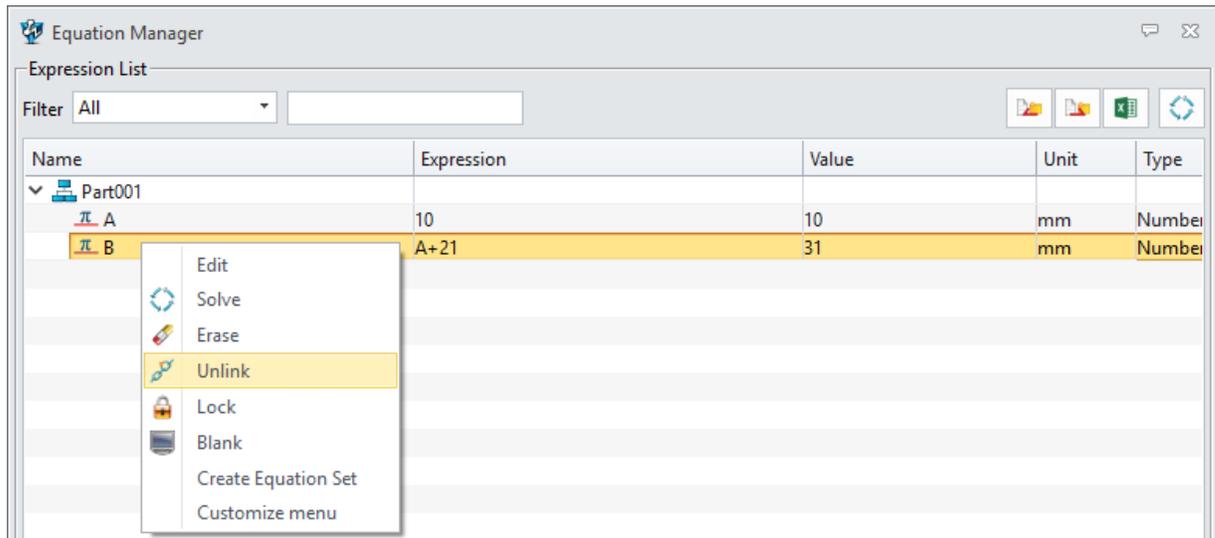
4. New “Keep edge connectivity” option in “Offset”

This new option can make sure the offset faces remain connected as they are offset before.



5. New “Unlink” option inside Equation manager

This unlink can converse all the variables inside an expression into constant so that the association can be broke.



3.4 Assembly Design

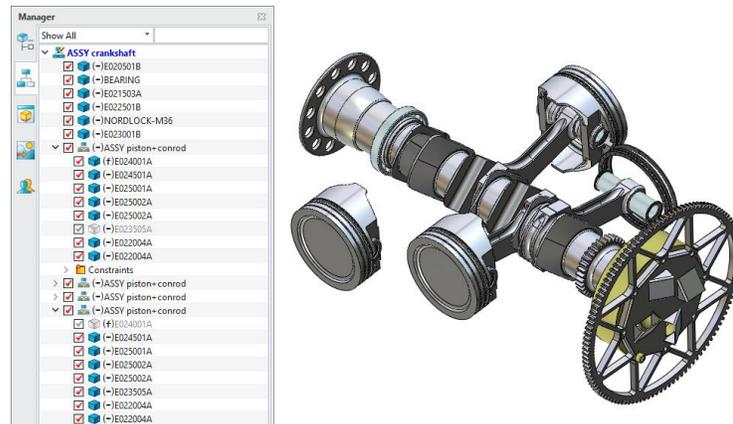
3.4.1 ★Expanded Assembly Manager

Assembly manager has expanded to record more component attributes to avoid modifying the original part object of the component, such as component visibility, color and position, so that different component instances can have different statuses.

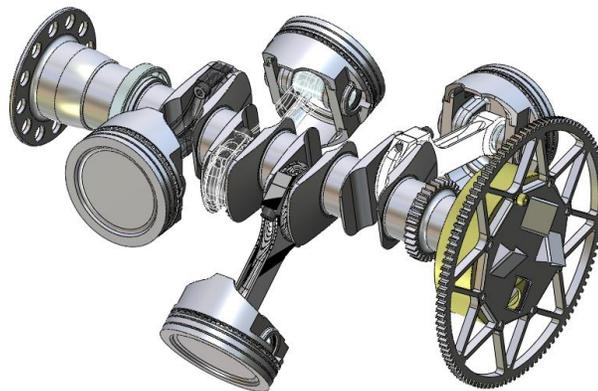
1. Component visibility of all assembly levels saved into the assembly object

Since the assembly object will record all the component visibility of all levels, different instances of same components can have different statuses.

The “Store current object visibility” option inside Custom View can record those component visibilities. If users need to toggle component visibility among different status, users can use this option to create several custom views, then activate specific one according to their needs.

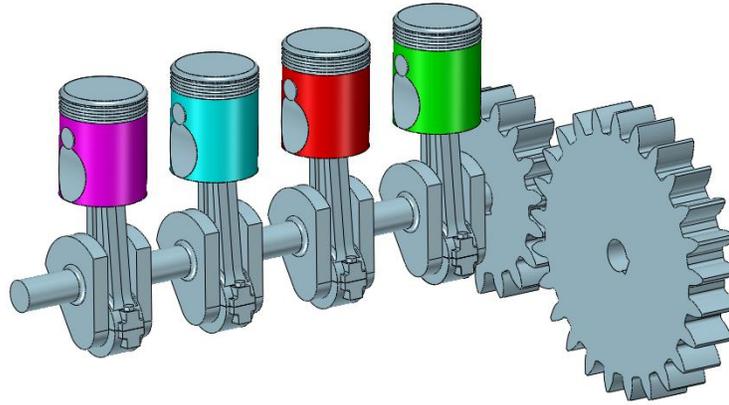


2. Component display mode of all assembly levels saved into the assembly object



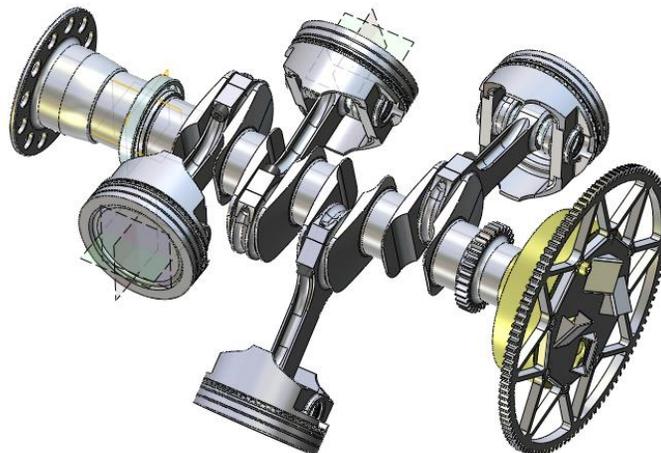
3. “Face Attribute” can work on picked faces of components

Same face from different instances of same component can be assigned different color without affecting the original part.



4. New “Blank/Unblank External Datum” option on the right-click menu of each component

Visibility of the datum planes of each instance now can be defined individually without affecting each other or the original part object. All datum planes are hidden by default even though the “External Datums On/Off” button on DA tool bar is on. Users need to turn on the datum plane of each instance through its right-click menu if they prefer to show this instance’s datum plane. The “External Datums On/Off” button only control whether external datum can be visible, and the datum plane of one component will be show if its own external datum is turned on.



5. New “Inherit Attributes” on the right-click menu of each component

“Inherit Attributes” can copy part attributes from different components quickly without any manual input. Attributes inside the original object of the component will be changed after this operation.

6. New “Part Attribute” on the right-click menu of each component

This “Part Attribute” can bring up the attribute form of the original object of the component without activating the component.

7. Component modified date is saved into the assembly object

The modified date of components in the top level is saved into assembly object, so that users can be noticed which components are modified when the assembly is open. If the component modified date is changed, the assembly will update its record, and users can decide whether to save the assembly or not.

8. New “Alt + Drag” behavior to insert picked component repeatedly

When pressing down Alt key to pick and drag a component, this picked component will be inserted at the new location.

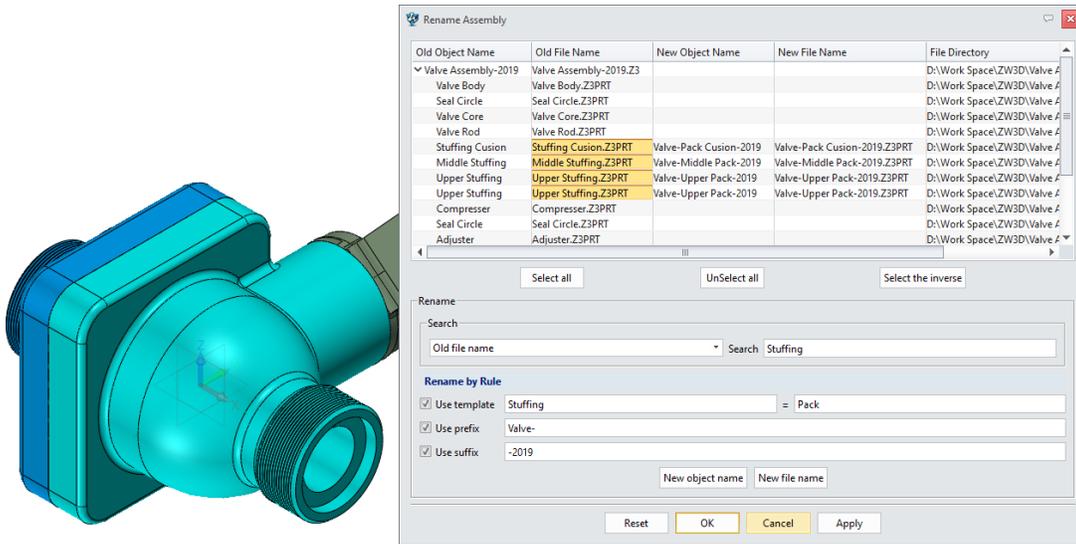
9. New “Toggle entity transparency” on the right-click menu of each component

This new option is used to toggle picked components transparent with pre-defined transparency, so that the component can be see-through. The transparency is pre-defined on “Configuration > Display> Numeric > Transparency used in On/Off (%)”.

This toggle won’t change the face transparency of the component, but the component display mode.

3.4.2 ★New “Rename Assembly”

“Rename Assembly” can rename object name and file name of all components of current assembly with assembly structure unchanged.



1. Object name and file name can be changed at the same time

Since object name and file name can be renamed at the same time, users need to input them separately. If the object is from Z3PRT, the object name will be auto synchronized with file name.

2. Multiple selection through search

Search is very useful to locate names with specified characters. Names with matching characters will be selected, then users can use fields from the “Rename by Rule” to change them all at one time.

3. Rename by Rule

3 rules are provided:

- “Use template” can find out specified characters in names and replace them with new characters.
- “Use prefix” and “Use suffix” can add extra characters to existing names.

4. Ctrl/Shift picking behavior is supported

Since rename can be done by rules, multiple selection with Ctrl/Shift behavior is supported.

→ **Where it is**

Part/Assembly context > right-click on blank area of the assembly tree > Rename Assembly

3.4.3 Changes in “Insert Component”

Several things have changed in “Insert Component” to help users to position component in batch.

1. New filter in object list to locate specific component

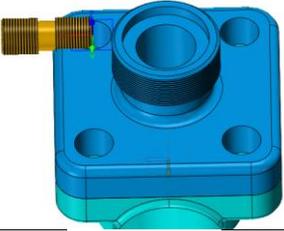
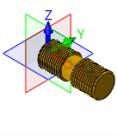
The new filter can distinguish part and assembly so that only part or assembly objects are listed as needed.

2. Double-click on Part lib to create and insert standard part on-place

The previous Part lib button beside the file folder is removed, but users can double-click the specified standard part item on the right docked Part lib form when “Insert Component” is on, then generate a new instance of standard parts and insert.

3. More placements are provided to insert multiple instances at one time

Placement	Description	Legend
Point	Old behavior to pick one point to place the inserted component	
Points	New behavior to pick multiple locations to place instances at one time.	

Active frame	New behavior to place the component by aligning its own frame to the assembly's active frame	
Default frame	New behavior to place component by aligning its own frame to the assembly's default frame	
Face/Datum	Old behavior to pick a face/datum and align the component with its XY datum	
Frame	Old behavior to pick a datum to place and align the component with Frame constraint	

4. New “Follow-up” option to define what to do next after insertion

3 actions are provided:

None – no follow-up action

Constrain after insertion – after insertion, “Common Constraint” will start up

Repeat after insertion - after insertion, more insertion points are required to input

5. New “Insert to layer” option to define which layer the inserted component will be put into

Users can pick an existing layer or create a new layer right away.

→ Where it is

Part/Assembly context > Assembly Ribbon > Insert

3.4.4 Assembly Constraint Improvements

1. Improved the precision of common constraint to get a more accurate solving result, especially for Angle and Concentric

Besides the accuracy improvement, a few more things have been done, such as Concentric constraint no long supports offset, and Middle constraint can pick one datum plane as input.

2. New “Use for position only” option

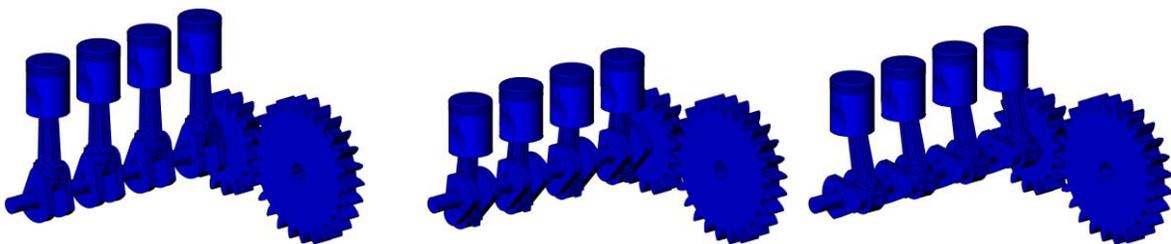
This option can control whether the constraint entry is registered or not. If enabled, no constraint will be added into the assembly tree.

3. Involving Pattern instances in constraint solving

After pattern instances are involved in the constraint solving, their constraint status will be the same as their parent component, and these instances can be treated as regular components, like dragging to move around, adding more constraints.

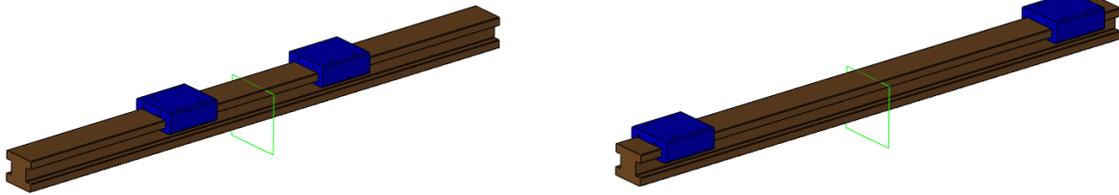
If pattern instances are not preferred to include in constraint solving to speed up, users can disable the “Inherit constraints” option.

Pattern instances with “At curve” and “At face” mode are still not involved in constraint solving.

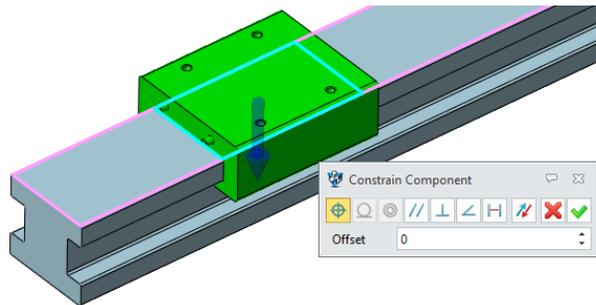


4. Involving Mirror instances in constraint solving

Similar with pattern instance, Mirror instances have the same constraint status as their parent component.

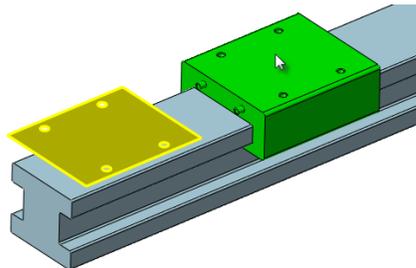


5. Offset field is provided on constraint context menu



6. Restore the drag-drop behavior during adding constraint to move around picked components

During adding constraint, users can directly pick (holding the left mouse key) and drag a component to move it to another position.



7. Keeping constraints related to datum plane when doing “Change component”

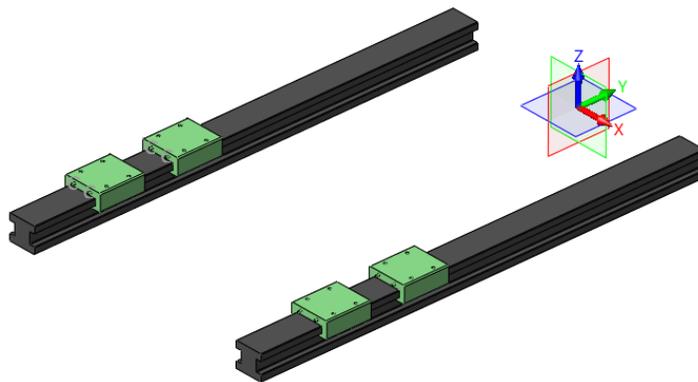
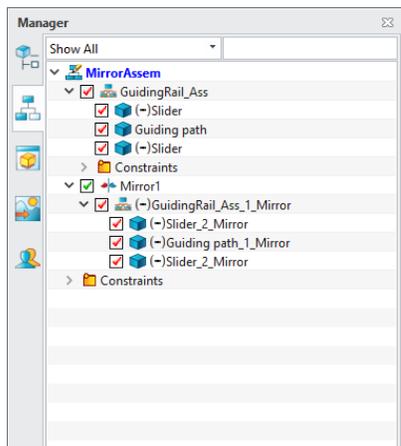
After executing “Change component”, existing constraint, which are using default datum plane as objectives, will update the objectives (datum plane of the component to change) to the corresponding ones from the target component to maintain the validity and avoid constraint failure.

3.4.5 New Support in “Mirror Component” on Duplicating Assembly

“Duplicate mirrored geometry” option in “Mirror Component” now can work on sub-assembly objects with 4 ways to do as following:

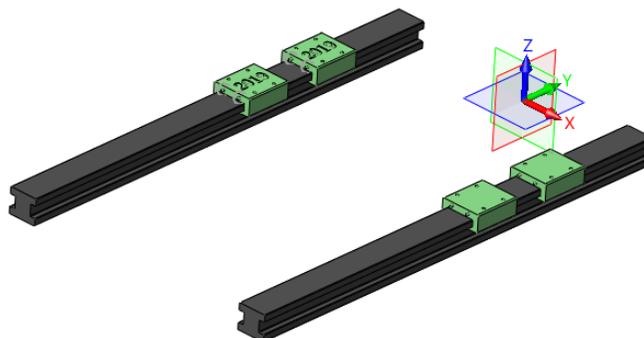
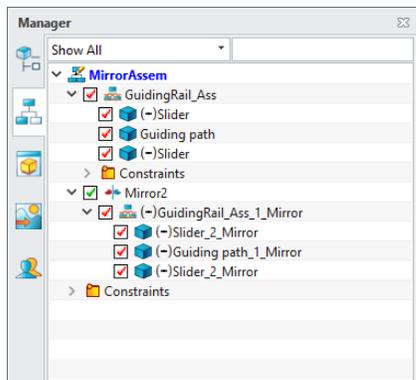
1. Keeping geometry and position associative

When both “Keep geometry associative” and “Keep position associative” options are enabled, a mirror feature will be registered into the assembly tree to make sure the new instance’s the position is following its parent object.



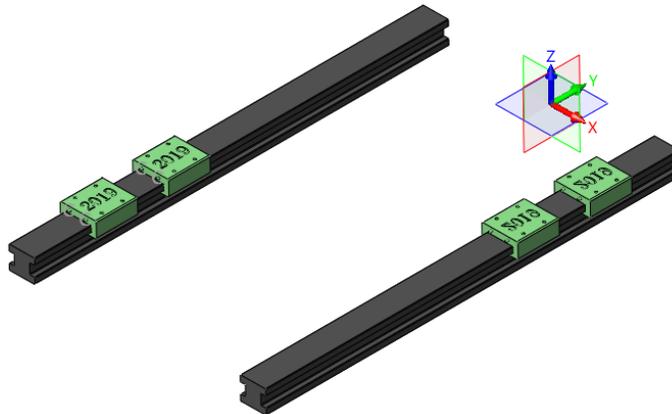
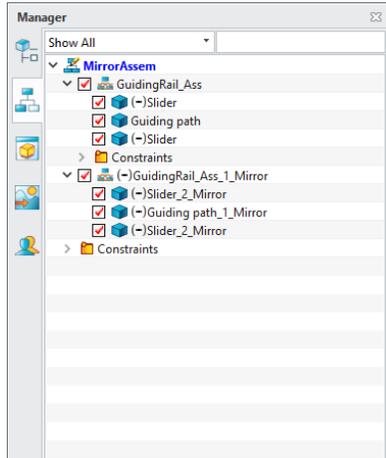
2. Only position is associative

The mirror feature is still registered into the assembly to make the position of the new instance and its parent synchronized when Only “Keep position associative” option is enabled, but the geometries of the instance can be different with the parent.



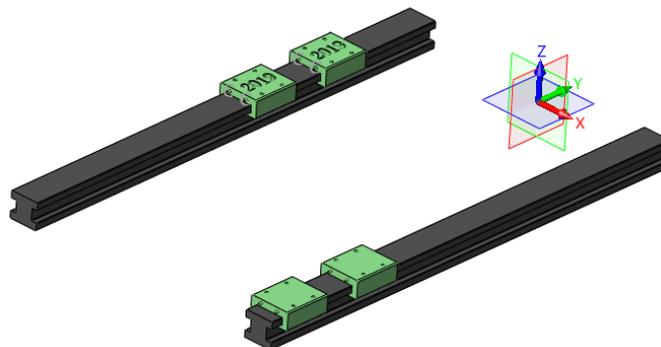
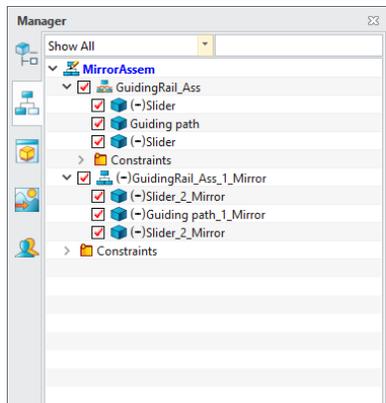
3. Only keep geometry associative

When only “Keep geometry associative” is enabled, no mirror feature is generated but only the new mirror instance is generated. The instance can be moved as whatever they like, but its geometry still follows what its parent looks like.



4. No association

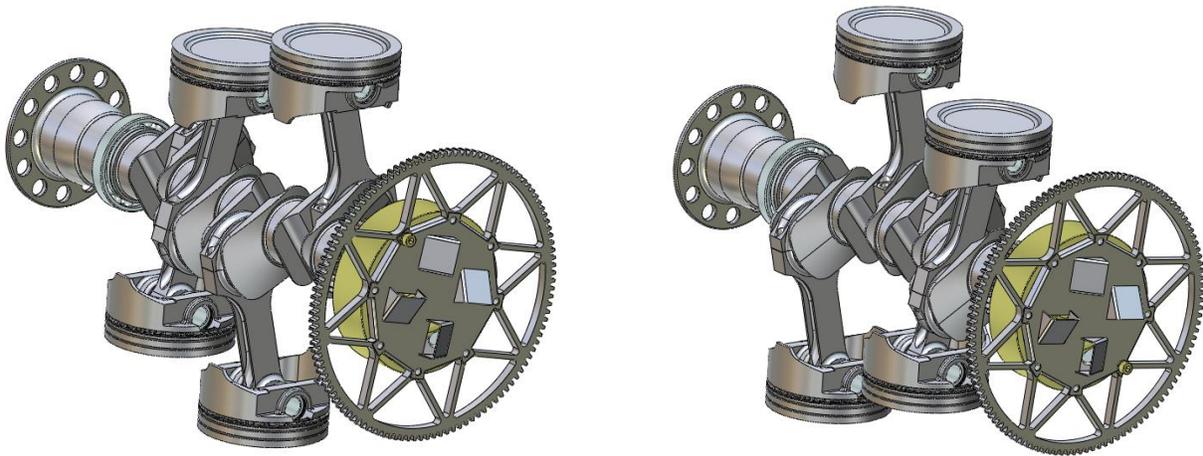
When both “Keep geometry associative” and “Keep position associative” options are not enabled, only the mirror instance is generated with no association with its origin.



3.4.6 ★New Flexible Assembly

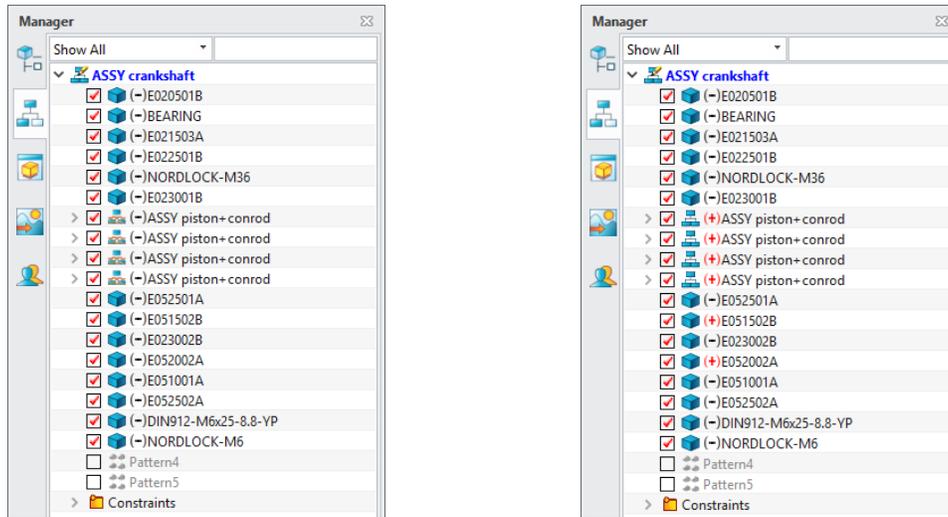
A normal sub-assembly is inserted into the assembly as one rigid component whose own parts can't move around each other under the assembly context, no matter those parts have or have no DOFs left. But the flexible sub-assembly is treated as movable component whose own parts can move around under the assembly context if those parts have DOFs.

For example, the part "Link rod" inside the sub-assembly "Piston" has a rotation DOF. After the Piston is inserted into the Crankshaft assembly and constrained with the crank part, and made as flexible, the "Link rod" can rotate along with the crank. All 4 instances of the Piston can move differently but accordingly with the crank.



Users can toggle sub-assembly into flexible through "Make flexible" command of its right-click menu. "Make rigid" is also available when the flexibility is not desired from a sub-assembly. There are different icons on the component node of the assembly tree when a sub-assembly is made flexible.

When a flexible assembly is turned back into rigid, the assembly may become over-constrained for its own parts can't be movable again. Users need to manually solve such issue, like disabling some of the constraints.



➔ Where it is

Part/Assembly context > right-click on sub-assembly no of the assembly tree > Make flexible/Rigid

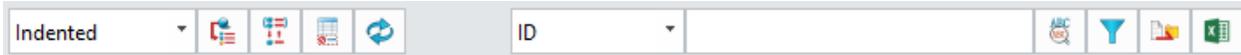
3.4.7 ★New 3D BOM

The new 3D BOM replaces the old “Part List” and uses spreadsheet to list out all assembly component info presenting a quite easy and known experience.

ID	Name	Material	BOM Struct	Mass (kg)	Stock Size	Quantity
1	E020501B	Steel	Normal part	7.43931	410.00x130.00x73.77(mm)	1
2	BEARING	Steel	Normal part	0.178293	13.00x80.00x80.00(mm)	1
3	E021503A	Steel	Normal part	0.11272	32.00x60.00x60.00(mm)	1
4	E022501B	Steel	Normal part	0.64493	55.00x125.00x125.00(mm)	1
5	NORDLOCK-M36	Steel	Normal part	0.0656451	55.00x55.00x6.60(mm)	1
6	E023001B	Steel	Normal part	0.280676	55.00x56.00x56.00(mm)	1
7	ASSY piston+conrod	Aluminum	Normal part	1.12061		4
8	E024001A	Aluminum	Normal part	0.470337	106.00x106.00x59.70(mm)	4
9	E024501A	Steel	Normal part	0.055757	105.60x105.60x15.25(mm)	4
10	E025001A	Steel	Normal part	0.126944	76.00x23.00x23.00(mm)	4
11	E025002A	Steel	Normal part	0.00144...	1.50x25.40x25.40(mm)	8
12	E023505A	Steel	Normal part	0.489593	23.00x74.00x179.50(mm)	4
13	E022004A	Aluminum	Normal part	0.0059758	19.00x51.79x25.53(mm)	8
14	E052501A	Steel	Normal part	0.969859	62.50x138.00x138.00(mm)	1

3.4.7.1 3D BOM Toolbar

3D BOM toolbar provides different tools to allow users to control what content will be shown in the list.



1. Component level selection

Similar with 2D BOM, user can select what level of components will be listed out.

Top-level only - only components from top level will be listed.

Part only - only components that is classified as part will be listed.

Indented - all components with the same assembly hierarchy will be listed.

Shape only - only shapes from current object will be listed.

When users use multiple shapes in one part to simulate components in an assembly, Shape only mode can list out all the shape as components, so that users can have a desired BOM.

2. “Include Shape” button to mix component and shape in one spreadsheet

This button can include shapes from current object into the list, so that the shapes can be treated as components.

3. “Display configurations of same part as one item” button to simplify the statistic

If there are different configurations for different instances from same component, users can choose to treat them as one item or leave them as individual.

4. “Show ‘do not list in BOM’ item” button to list out all components

Components with “do not list in BOM” attribute won’t be listed in the BOM. This button can list them together without modifying the attribute.

5. “Update calculated attributes” button to auto calculate all listed components

Listed calculated attributes like Mass, Volume, Stock size and so on, can be calculated all at one time for all listed components so that users have no need to calculate one by one. The more the calculated attributes and components are, more time will be needed.

6. Search function to locate specified item

Users can search input characters in specified attribute to locate desired row in character match or extract match. Found rows will be highlighted, and “Filter search result” can only list out the matched rows to simplify the spreadsheet.

ID	Name	Material	Quantity
1	Valve Body	ZG230-450	1
2	Valve Cover	ZG230-450	1
3	Seal Circle	聚四氟乙烯	2
4	Valve Core	40Cr	1
5	Adjuster	聚四氟乙烯	1
6	Valve Handle	40Cr	1
7	Stuffing Cusion	40Cr	1
8	Middle Stuffing	聚四氟乙烯	1
9	Upper Stuffing	聚四氟乙烯	2
10	Compressor	35	1
11	NH4O	ZG230-450	1
12	Bolt M12X30	Q235	4
13	Nut M12	Q235	4
14	Virtual Pipe1 in Assembly	Aluminum	1
15	Virtual Pipe2 in Assembly	Aluminum	1

ID	Name	Material	Quantity
14	Virtual Pipe1 in Assembly	Aluminum	1
15	Virtual Pipe2 in Assembly	Aluminum	1

7. “Export format” to save current setting to reuse next time

Spreadsheet format include all the things current BOM is using. Users can use this export to set up different formats for specific scenarios, then using “Columns from template” button to re-use these formats to generated desired content. There is [a sample link](#).

8. “Export Data” to output the list into an Excel file

“Export Data” can output all the content from the list and generate an Excel file with define format, so that users can share the spreadsheet with others.

The export form can be defined in “Export excel with template” on the Setting tab. if no format is defined, then it simply exports as what you see. [Here is a sample link](#).

3.4.7.2 3D BOM Spreadsheet

3D BOM list is a spreadsheet to list out all the component info for users to check on, or modify value of specified attribute, even to create new user attributes into every component.

1. Right-click menu with different edit option is provided on the table header

ID	Name	Material	BOM Struct	Mass	Stock Size	Quantity
1	E020501B		Normal part	7.43931	410.00x130.00x73.77(mm)	1
2	BEARING		Normal part	0.178293	13.00x80.00x80.00(mm)	1
3	E021503A		Normal part	0.11272	32.00x60.00x60.00(mm)	1
4	E022501B	Steel	Normal part	0.64493	55.00x125.00x125.00(mm)	1
5	NORDLOCK-M36	Steel	Standard part	0.0656451	55.00x55.00x6.60(mm)	1

ID	Name	Material	BOM Struct	Mass	Stock Size	Quantity
1	E020501B	Steel	Normal part	7.43931	410.00x130.00x73.77(mm)	1
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4	E022501B	Steel	Normal part	0.64493	55.00x125.00x125.00(mm)	1
5	NORDLOCK-M36	Steel	Standard part	0.0656451	55.00x55.00x6.60(mm)	1

- **Sort by Column**

Sorting the list with picked column in Ascending or Descending way.

- **Hide Column**

Remove the pick column from the list. Users can bring the hidden column back from the Column tab on the right side.

- **Edit Shape Names**

This option only appears when shapes are included in the list, and users can rename the shapes in batch. For detail, please refer to “[New Shape Attribute](#)” chapter.

- **Rename the column**

Each attribute column can have a descriptive name to replace the original one for better understanding. Users can do the same thing on the descriptive name column on the Column tab.

- **Auto Stock Size**

This option is provided on Stock size attribute column and to bring up the Auto Stock Size form.

- **Merge by Column**

This option is only provided on attributes which is string type, and can combine rows with the same value of picked column into one row.

2. Drag-drop the column to re-order is supported

3. Right-click menu is provided on row

The screenshot shows a '3D BOM' window with a table containing 5 rows. A context menu is open over the second row, showing options: 'Copy', 'Attributes', and 'Exclude Item'. The table columns are: ID, Name, Material, BOM Struct, Mass, Stock Size, and Quantity.

	ID	Name	Material	BOM Struct	Mass	Stock Size	Quantity
1	1	E020501B	Steel	Normal part	7.43931	410.00x130.00...	1
2			Steel	Normal part	0.178293	13.00x80.00...	1
3			Steel	Normal part	0.11272	32.00x60.00...	1
4			Steel	Normal part	0.64493	55.00x125.00...	1
5	5	NORDLOCK-M36	Steel	Standard part	0.0656451	55.00x55.00...	1

- **Copy**

Copying all data from the picked row, then Pasting them into another rows.

- **Attributes**

This option can bring up the attribute form of the picked items to edit.

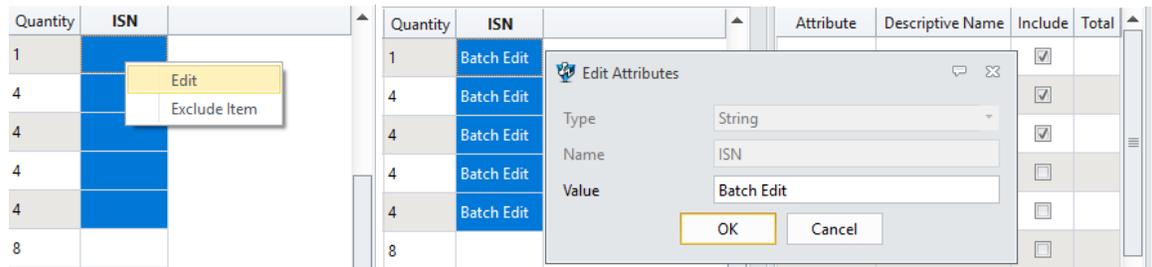
- **Exclude Item**

This option can remove the row from the spreadsheet. Users can bring the excluded rows back from the Row tab on the right side.

4. Behaviors on Cells

The right-click menu is provided for cells, and its content will be different on different cells.

Multiple selection by Ctrl/Shift picking is supported, and batch edit on selected cells

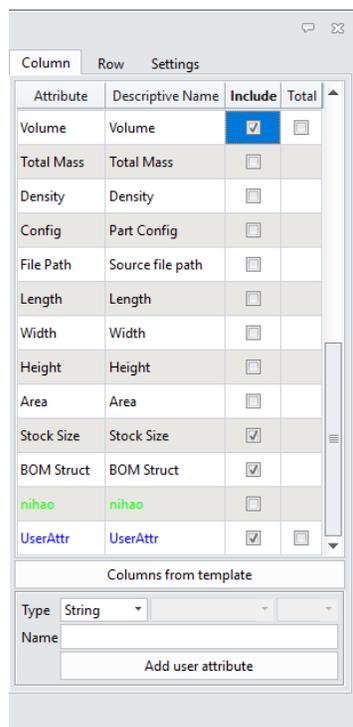


5. Summation row for numeric columns

All numeric columns can have its own summation which users can define on the Column tab.

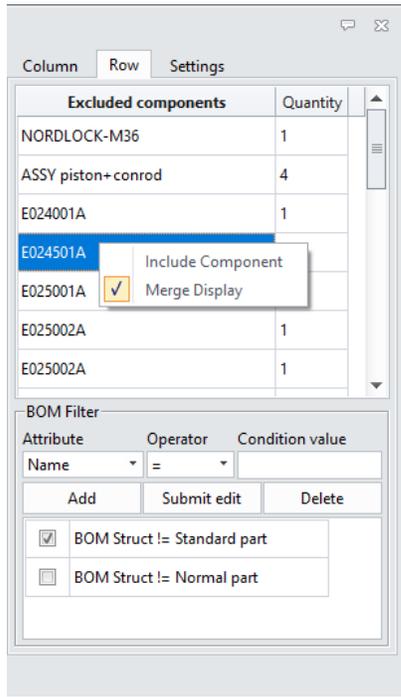
3.4.7.3 BOM Setting Tab

1. Column tab



- Attribute list**
 All available part attributes are listed here. Descriptive Name column can give each attribute a nickname. Include column can control whether the attribute should be included on the left side. Total column is only available for numeric attribute. Existing user attributes are marked in green, and newly-created but not-saved attributes are marked in blue.
- Columns from template**
 This button can read an external format file to update the Column list above and the spreadsheet on the left.
- User Attribute Creation**
 Users can create common user attributes for multiple components without activating each component to save time. Only when such common user attribute on each component's cell have explicit value, then corresponding user attribute on original component's part attribute form will be created.

2. Row tab



- **Exclude list**

All exclude components are listed here for users to check on or include again.

Same components can be listed in one row, or one for each.

- **BOM Filter**

Similar with the filter in 2D BOM, users can set up different filter conditions using different attributes.

Filter condition can be saved into BOM format.

3. Setting tab

Users can set up default templates or BOM, pre-defined attribute list and export Excel.

- **BOM Template sample**

There is a sample file Default.Z3DBOMTT under installation directory > supp

3D BOM Format

If want to include shape in the part (0/1)

SHAPE = 0

Traverse level (0 - top level only, 1 - parts only, 2 - indented)

LEVEL = 2

Set attribute name for sorting

SORT_KEY = []

Set sorting order (0 - ascending, 1 - descending)

```

SORT_ORDER = 0

# Merge various configurations of same part as one item

MERGE_CFG = 0

# Show "do not list in BOM" items

SHOW_DONOTLIST = 1

# Specify the attribute displayed in each column

COL = ID

    attrib = [Index]

    total = [0]

END_COL

COL = Name

    attrib = [Name]

    total = [0]

END_COL

COL = BOM Struct

    attrib = [BOM Struct]

    total = [0]

END_COL

COL = Mass

    attrib = [Mass]

    total = [1]

END_COL

END_COL

COL = Quantity
    
```

```

    attrib = [Quantity]

    total = [1]

END_COL

COL = ISN

    attrib = [Number]

    total = [0]

END_COL

# Set the exclusion condition for the BOM

# [Attribute name] (=,!=,>,<,>=,<=) Value

FILTER

[BOM Struct] != Standard part

    use = [1]

END_FILTER

# Set the attribute to merge by (default is null)

MERGE = [Number]

# Set the alias of the column name

ALIAS

[3] = ISN

END_ALIAS

```

- **Pre-defined attribute list sample**

There is a sample file Template.z3preset under installation directory > supp

```

# Attribute names and value lists must be placed between flags "PRESET_LIST" and
"PRESET_LIST_END"

PRESET_LIST

```

@[Designer]

Candy

Sophia

@[Number]

ISO123456

GB123456

@[Cost]

\$10

\$100

PRESET_LIST_END

- BOM Export Excel Sample**

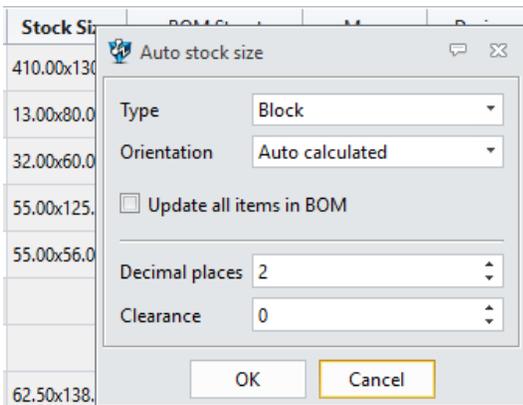
There is a sample file ExportConfiguration.txt under installation directory > supp.

Sample	Result																																																																																																																																																										
<p># Configuration to export part list # Set the template excel file TEMPLATE = zw3d.xlsx # Write the value of the attribute of the current assembly to the specified cell [Name] = C4 [Designer] = E4 [End Date] = G4 # Write the name of the attribute to the specified cell @[Name] = B5 @[Material] = D5 # Set the start row of the data to write (1,2,3...) START_ROW = 6 # Set the column to write for the part attributes # (if no column is specified, write all the data as you see in 3D BOM) DATA [Index] = A</p>	<table border="1"> <thead> <tr> <th>No</th> <th>Name</th> <th>Material</th> <th>Material</th> <th>Name</th> <th>Qty</th> <th>Note</th> </tr> </thead> <tbody> <tr><td>1</td><td>E020501B</td><td>Steel</td><td>410.00x130.00x73.77(mm)</td><td></td><td>7.43931</td><td>1</td></tr> <tr><td>2</td><td>BEARING</td><td>Steel</td><td>13.00x80.00x80.00(mm)</td><td></td><td>0.178293</td><td>1</td></tr> <tr><td>3</td><td>E021503A</td><td>Steel</td><td>32.00x60.00x60.00(mm)</td><td></td><td>0.11272</td><td>1</td></tr> <tr><td>4</td><td>E022501B</td><td>Steel</td><td>55.00x125.00x125.00(mm)</td><td></td><td>0.64493</td><td>1</td></tr> <tr><td>5</td><td>NORDLOCK-M36</td><td>Steel</td><td>55.00x55.00x6.60(mm)</td><td></td><td>0.0656451</td><td>1</td></tr> <tr><td>6</td><td>E023001B</td><td>Steel</td><td>55.00x56.00x56.00(mm)</td><td></td><td>0.280676</td><td>1</td></tr> <tr><td>7</td><td>ASSY piston+conrod</td><td>Aluminum</td><td></td><td></td><td>1.12061</td><td>4</td></tr> <tr><td>7.1</td><td>E024001A</td><td>Aluminum</td><td>106.00x106.00x59.70(mm)</td><td></td><td>0.470337</td><td>4</td></tr> <tr><td>7.2</td><td>E024501A</td><td>Steel</td><td>105.60x105.60x15.25(mm)</td><td></td><td>0.055757</td><td>4</td></tr> <tr><td>7.3</td><td>E025001A</td><td>Steel</td><td>76.00x23.00x23.00(mm)</td><td></td><td>0.126944</td><td>4</td></tr> <tr><td>7.4</td><td>E025002A</td><td>Steel</td><td>1.50x25.40x25.40(mm)</td><td></td><td>0.00144608</td><td>8</td></tr> <tr><td>7.5</td><td>E023505A</td><td>Steel</td><td>23.00x74.00x179.50(mm)</td><td></td><td>0.489593</td><td>4</td></tr> <tr><td>7.6</td><td>E022004A</td><td>Aluminum</td><td>19.00x51.79x25.53(mm)</td><td></td><td>0.0059758</td><td>8</td></tr> <tr><td>8</td><td>E052501A</td><td>Steel</td><td>62.50x138.00x138.00(mm)</td><td></td><td>0.969859</td><td>1</td></tr> <tr><td>9</td><td>E051502B</td><td>Aluminum</td><td>26.00x132.01x132.00(mm)</td><td></td><td>0.0623392</td><td>1</td></tr> <tr><td>10</td><td>E023002B</td><td>Steel</td><td>50.00x40.00x40.00(mm)</td><td></td><td>0.147044</td><td>1</td></tr> <tr><td>11</td><td>E051002A</td><td>Aluminum</td><td>20.00x20.00x7.00(mm)</td><td></td><td>0.00579464</td><td>2</td></tr> <tr><td>12</td><td>E051001A</td><td>Steel</td><td>8.00x245.00x244.98(mm)</td><td></td><td>0.656252</td><td>1</td></tr> <tr><td>13</td><td>DIN912-M6x25-8.8-YP</td><td>Steel</td><td>10.00x10.00x31.00(mm)</td><td></td><td>0.00733716</td><td>1</td></tr> <tr><td>14</td><td>NORDLOCK-M6</td><td>Steel</td><td>11.00x11.00x1.80(mm)</td><td></td><td>0.000863017</td><td>1</td></tr> <tr><td></td><td></td><td></td><td></td><td></td><td>15.20675864</td><td>50</td></tr> </tbody> </table>	No	Name	Material	Material	Name	Qty	Note	1	E020501B	Steel	410.00x130.00x73.77(mm)		7.43931	1	2	BEARING	Steel	13.00x80.00x80.00(mm)		0.178293	1	3	E021503A	Steel	32.00x60.00x60.00(mm)		0.11272	1	4	E022501B	Steel	55.00x125.00x125.00(mm)		0.64493	1	5	NORDLOCK-M36	Steel	55.00x55.00x6.60(mm)		0.0656451	1	6	E023001B	Steel	55.00x56.00x56.00(mm)		0.280676	1	7	ASSY piston+conrod	Aluminum			1.12061	4	7.1	E024001A	Aluminum	106.00x106.00x59.70(mm)		0.470337	4	7.2	E024501A	Steel	105.60x105.60x15.25(mm)		0.055757	4	7.3	E025001A	Steel	76.00x23.00x23.00(mm)		0.126944	4	7.4	E025002A	Steel	1.50x25.40x25.40(mm)		0.00144608	8	7.5	E023505A	Steel	23.00x74.00x179.50(mm)		0.489593	4	7.6	E022004A	Aluminum	19.00x51.79x25.53(mm)		0.0059758	8	8	E052501A	Steel	62.50x138.00x138.00(mm)		0.969859	1	9	E051502B	Aluminum	26.00x132.01x132.00(mm)		0.0623392	1	10	E023002B	Steel	50.00x40.00x40.00(mm)		0.147044	1	11	E051002A	Aluminum	20.00x20.00x7.00(mm)		0.00579464	2	12	E051001A	Steel	8.00x245.00x244.98(mm)		0.656252	1	13	DIN912-M6x25-8.8-YP	Steel	10.00x10.00x31.00(mm)		0.00733716	1	14	NORDLOCK-M6	Steel	11.00x11.00x1.80(mm)		0.000863017	1						15.20675864	50
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[Name] = B [Material] = C [Stock Size]= D [Mass] = E [Quantity] = F [Remark] = G END_DATA	
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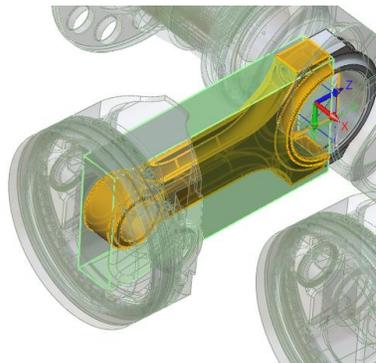
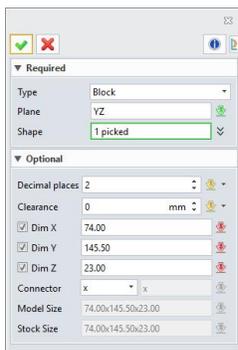
3.4.7.4 Stock Size Calculation

“Auto Stock Size” and “Edit Stock Size” are only available for stock size attribute. Users can have them through the right-click menu. “Auto Stock Size” is meant to get the result without much manual operation, and “Edit Stock Size” provides options to allow users to manually define their desire stock size.



What “Auto Stock Size” can do:

- 3 types of stock: Block, Circumscribed and inscribed cylinder.
- 2 ways to orient the stock: using the default frame, and auto calculation to have longest edge as axis to orient
- Update all stock sizes of components
- Extra clearance can be defined to add to stock



What “Edit Stock Size” can do:

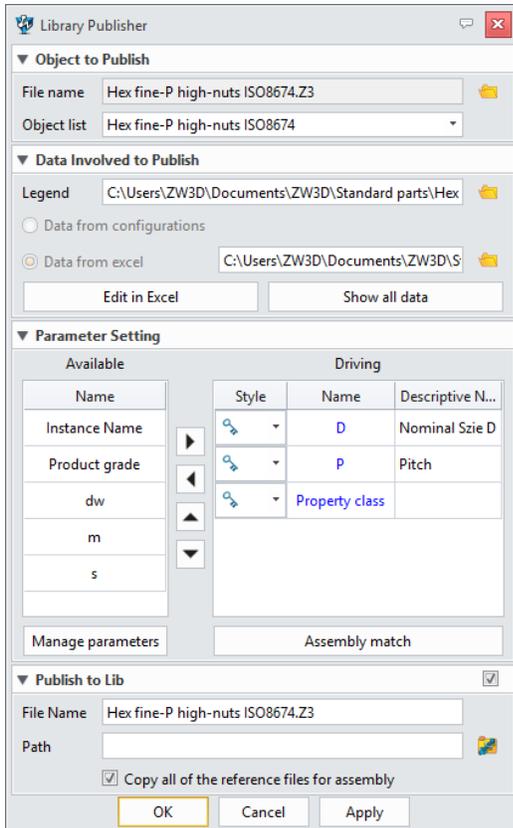
- Same 3 types of stock as above.
- Manually picked datum plane to orient and shapes to calculate.
- Values for X, Y, Z axis can be modified and included or not in the result.

→ Where it is

Part/Assembly context > Assembly Ribbon > 3D BOM

3.4.8 “Part Table” Update into “Part Publisher”

Basing on feedback, “Part Table” have updated into “Part Publisher”. Here are the details.



1. Workflow has changed into vertical

2. External Excel data can be redefined

If the data is from Excel, users can specify a new external Excel file as the source data.

3. Data can be edited on-place

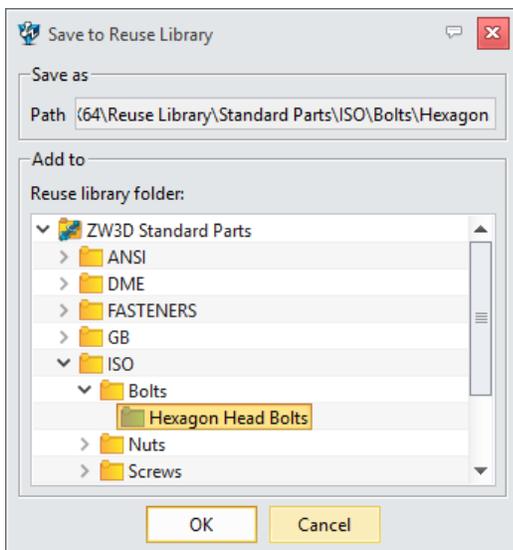
- “Edit in Excel” button can open the data in Excel, then users can edit anything they like.
- “Show all data” button is a data viewer to facilitate users that have no Excel installed.

4. Configured items modification

- Available parameter list shows all configured parameters so that users can pick anyone to define as driving. “Manager parameter” can bright the “Config Table” to allow users to add or remove configured items.
- Driving parameter list shows all key or custom parameter so that users can change their style or descriptive name. The sequence of driving parameters is the same sequence they appear on “Add reuse part” form. Users can change the sequence with the Up and Down buttons.
- Parameters can travel between Available and Driving lists with the Right and Left buttons, or double-clicking.

5. New “Publish to Lib”

This option can publish the part into default standard part lib.



6. Changes in Z3L file

Z3L file now can record the sequence of the driving parameters with following format:

#keysSequence=\$var1, \$var2

Meanwhile Z3L can save the directory of the component files of the current assembly file to make sure the new assembly instance can use the right components. The format is as following:

#components

Plain large washers ISO7093=..\folder name

Hex fine-P high-nuts ISO8674=..\folder name

7. Changes in Data Excel file

The data Excel file can add Descriptive name for common configured items, not only for the driving parameters, to work with the same change in [Config Table](#).

New “ATTRIBUUTES” keyword is meant to set specified value into defined attributes with format as: Attribute name=value.

New “COMPONENT” keyword can define how the assembly’s component will be selected when new assembly instance is generated. Here is the format:

Full name of component	Selection condition1	Selection condition2
------------------------	----------------------	----------------------

The full name of component should be in the first column and include file name and object name, like Full file name:Object name.

Condition should start with the component’s own configured parameter and following with relation operators (<, >, =, <=, >=, =>, =<), then ending with the assembly’s configured parameter, such as following picture shown,

	A	B	C	D
1	##KEYS			
2	<NAME>	\$M	\$P	
3	<TAG>	Nomial Diameter	Pitch	
4	##CUSTOMS			
5	<NAME>			
6	<TAG>			
7	<MIN>			
8	<MAX>			
9	##DESCRIPTIONS			
10	<NAME>	\$PClass		
11	<TAG>	Property Class		
12	##ATTRIBUTES			
13	Number=N123456	UserAttribute=Demo Sample		
14				
15	##COMPONENTS			
16	Plain large washers ISO7093.Z3:Plain large washers ISO7093	\$d=>\$M		
17	Hex fine-P high-nuts ISO8674.Z3:Hex fine-P high-nuts ISO8674	\$D=>\$M	\$P=>\$P	
18				
19	##PARAMETER			
20		\$M	\$P	\$PClass
21		8	1	"A"
22		10	1	"A"
23		10	1.25	"A"
24		12	1.25	"A"

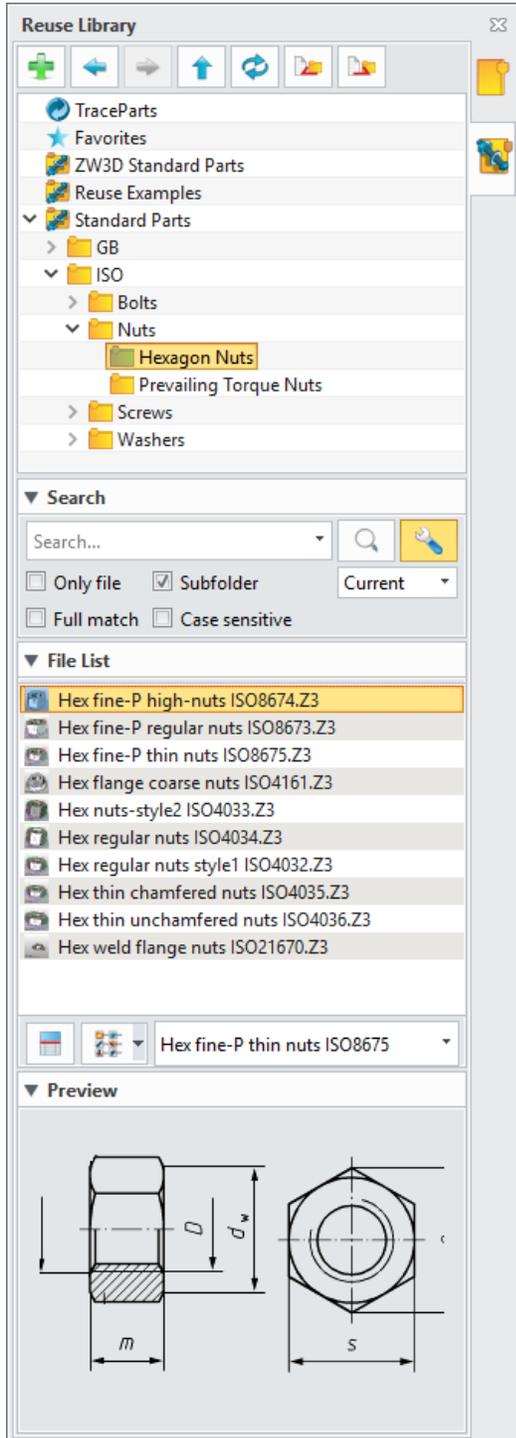
➔ Where it is

Part/Assembly context > Assembly Ribbon > Part Publisher

Part/Assembly context > Tools Ribbon > Part Publisher

3.4.9 “Reuse Library” Improvements

Basing on received feedback, “Reuse Library” has optimized its form and workflow, and added more new functionalities.



1. Reuse Lib has embedded into File browser on the right

Since the Reuse lib is together with the File browser, its form won't block the graphic area anymore, and users can insert different standard parts repeatedly with simple dragging-drop.

2. New “Export” and “Import” buttons

“Export” can output current file structure tree into an XML file so that users can re-use it on different PC or next time with “Import” button.

3. More new options for Search

Here are the new options in Search for users to more accurate locate specified object.

- Only file – only files are searched excluding any folders.
- Subfolder – only current or all subfolder will be included in search
- Full match – all input characters must be matched.
- Case sensitive

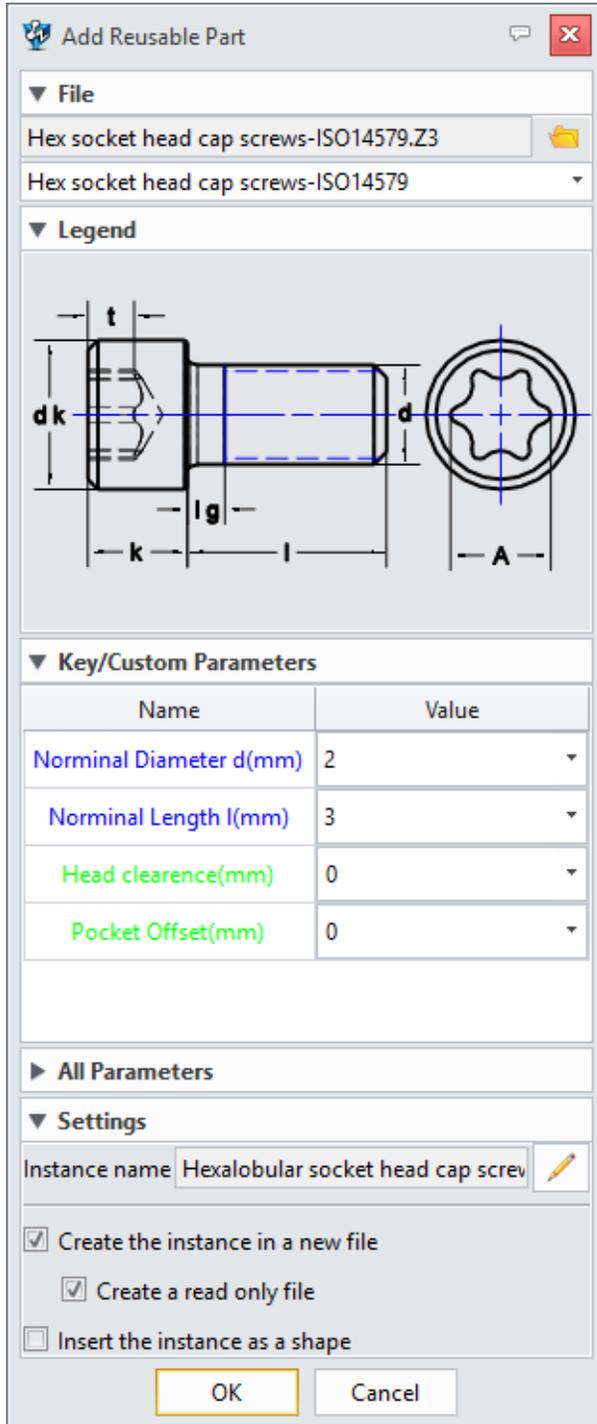
4. More insertion behaviors

Double-clicking on a file can insert its new instance into current object, or directly drag a file into graphic area and drop to insert.

The “Insert as component” from the right-click menu of the picked file can insert the original object of the file as component, not using “Add Reusable Part” to generate new instance.

3.4.10 “Add Reusable Part” Adjustment

Since “Reuse Lib” and “Part Publisher” have updated, “Add Reusable Part” has to update along with following details:



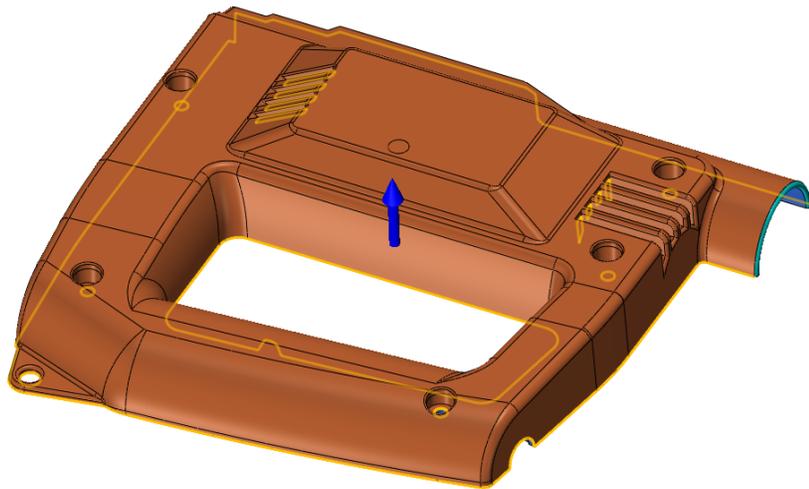
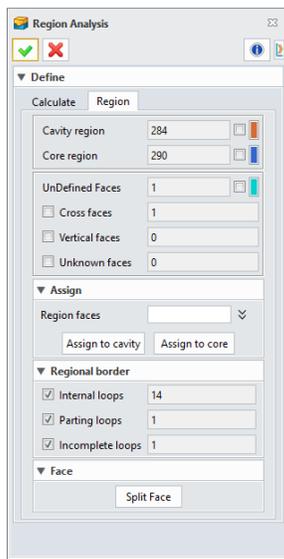
- 1. Single key/custom parameter list**
 Key and custom parameters now are merged into one list which users can distinguish them with their colors: blue is the key and green is the custom.
- 2. New way to change part type**
 The old “Transfer to Part Lib” button is removed since there is no form for “Reuse Lib”. Users can change the part type by double-clicking on the item inside the Reuse Lib when “Add Reusable Part form is open.
- 3. New “Create a read-only file**
 The instance file is created with read-only attribute by default. If this is not desired, users can turn this option off.
- 4. New feature for “Insert the instance as a shape”**
 A specified feature will be generated when the instances is inserted as shape. Users can manipulate the feature as any other features, and “Add Reusable Part” form will be brought up when redefining the feature.

3.5 ★Mold Design

3.5.1 ★Renovated Parting Design

3.5.1.1 ★Update in “Region Analysis”

“Region Analysis” now can perform another new analysis on modified faces while keeping previous result with a much faster speed as following details:



1. Update result while keeping last analysis result

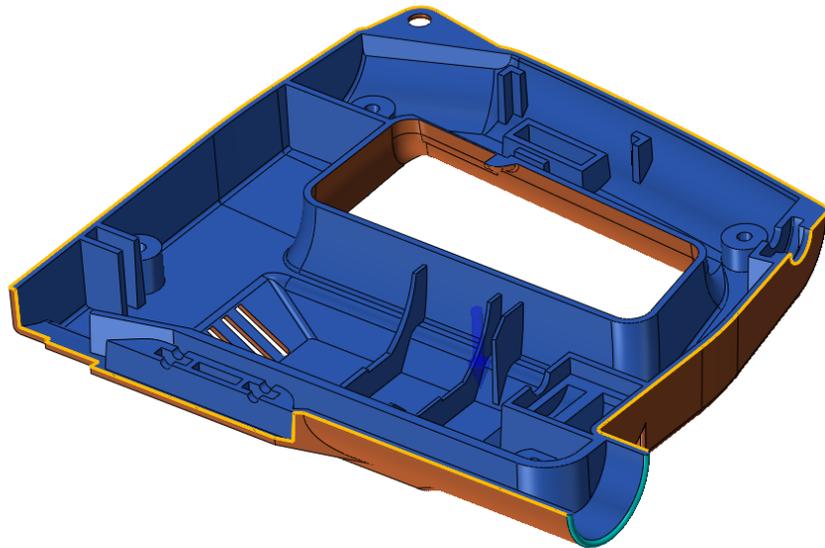
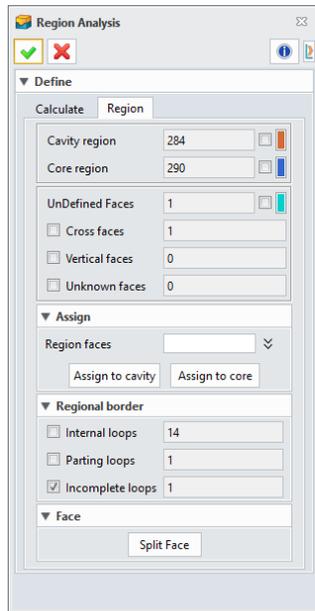
The new “keep existing” option can remain the last analysis result and only update the new faces, while “Reset all” option just operates a fully new analysis.

2. New “Regional border” analysis

“Regional border” provides 3 kinds of results:

- Inner loop - all inner openings inside the model
- Parting loop - the only outer loops
- Incomplete loop - incomplete inner and outer loops. After fine region classification, this field should be zero.

There is one incomplete loop for there is one cross face at the sample below.



3. New “Split Face”

“Split Face” is meant to divide the cross face as above, so that all regions can be defined explicitly. 3 ways are provided by “Split Face”:

- Face Silhouette

Curves are generated at face silhouette with given direction, then cross faces are divided with those curves.

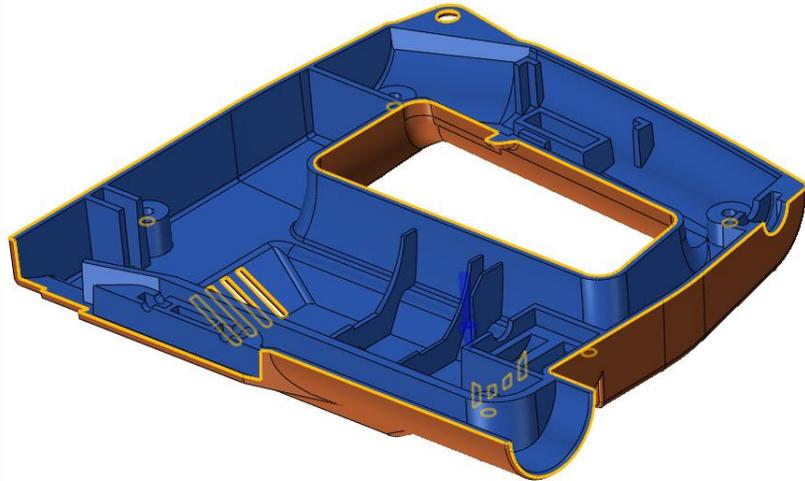
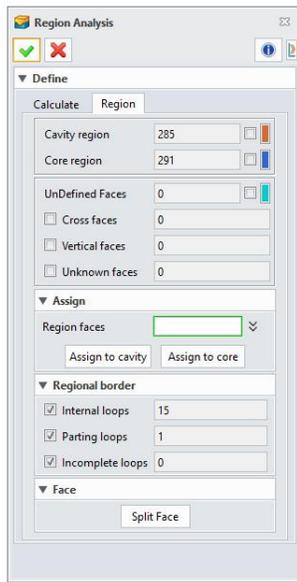
- Select Curves to divide

Cross faces are directly divided with picked curves. Users can generate curves if there is no suitable curve to pick with the “Add Line”.

- Face/Datum

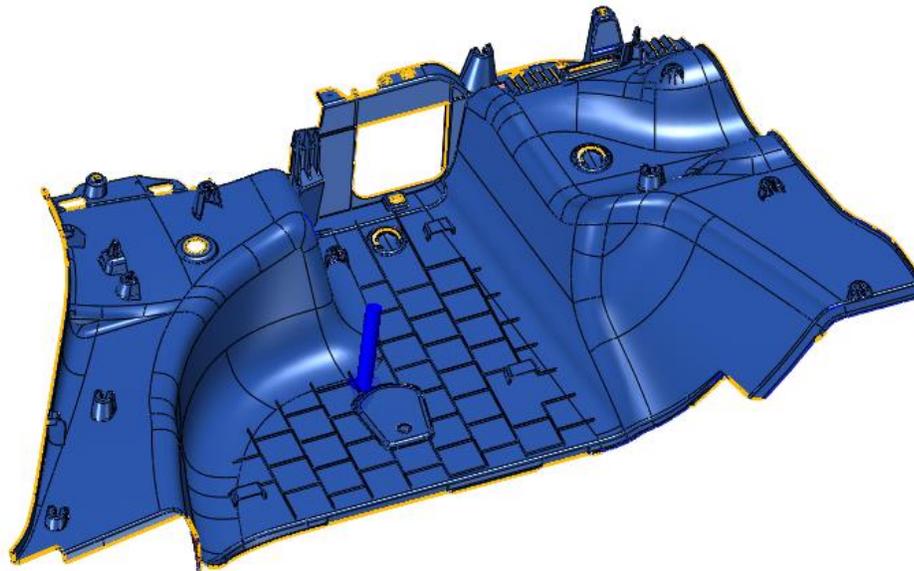
Cross faces are divided with picked faces.

The cross face from above model can be divided with “Face Silhouette”, then the resultant faces can be assigned to core and cavity region separately.



4. Better result and efficiency

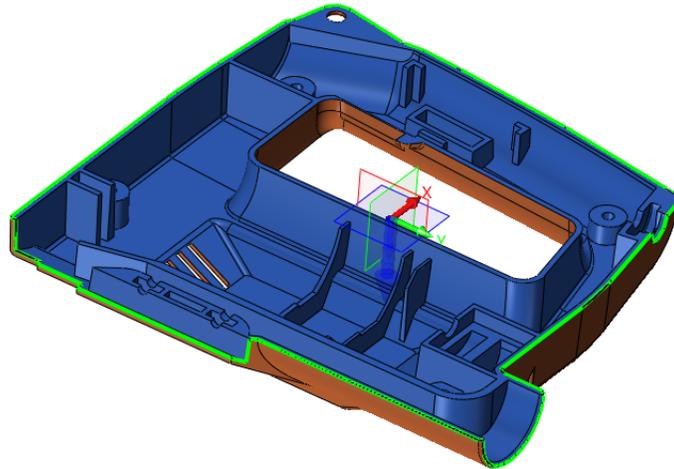
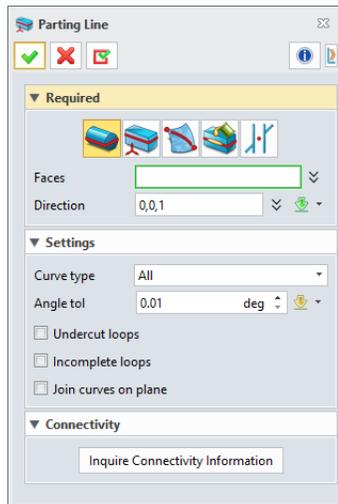
“Region Analysis” has optimized its algorithm to produce a better and stable analysis result with faster speed. Analysis result will remain the same as much as the one before regeneration, and user can perform another analysis to update the region classification at time as needed.



3.5.1.2 Better Parting Line Tools

A few improvements have done for parting lines.

1. Previous parting line commands have integrated into single one parting line command

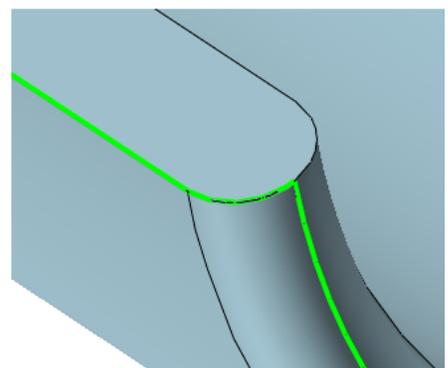
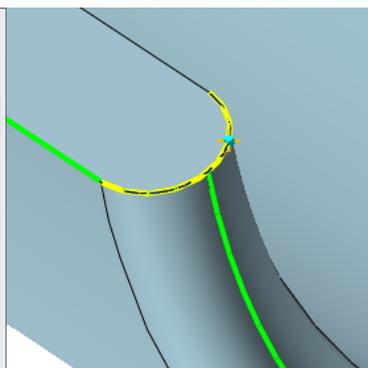
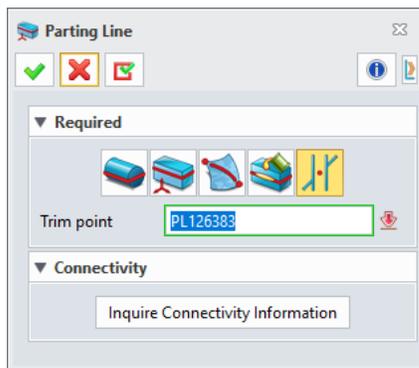


2. Connectivity can be performed during

Connectivity is embedded into Parting Line form so that users can check existing parting lines.

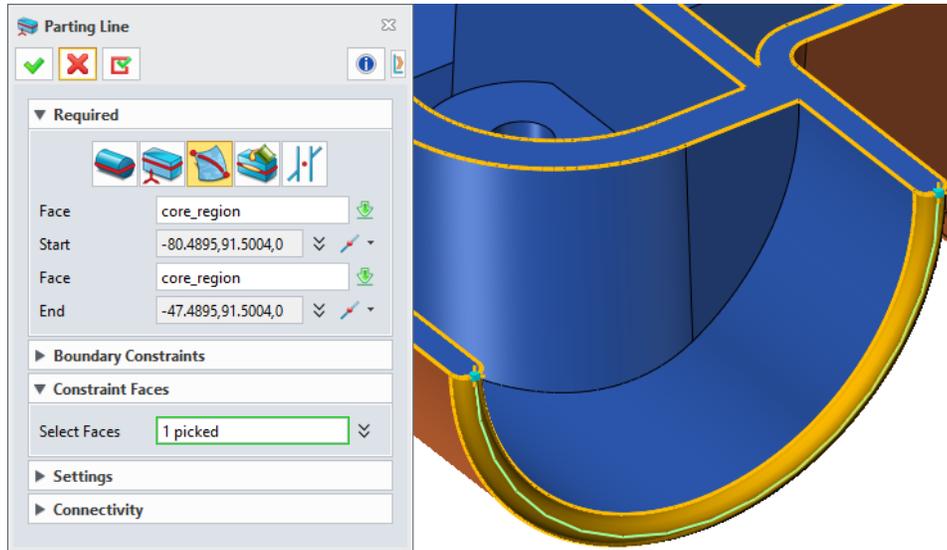
3. New “Click to Trim Parting Line”

Users can delete any extra or unwanted existing parting lines with this “Click to trim parting line”.



4. New “Constraint Face” in “Parting line between two points”

Same as the new function in [“Blend on Face”](#), “Constraint Faces” can guide how the blend curve will pass through, especially when the curve needs to cross faces that are from different shapes. A parting line can be generated on the following cross face.



3.5.1.3 ★New “Region Definition”

“Region Definition” defines which faces belong to core and cavity, or other regions, and then extract faces of those regions into individual shapes leaving the product model unchanged.

1. 2 ways to define regions

- Basing on defined regions

This mode allows users to pick faces manually into different regions to classify. Normally core and cavity regions are defined after region analysis, so that users can use this mode to define extra regions other than core and cavity.

- Basing on parting lines

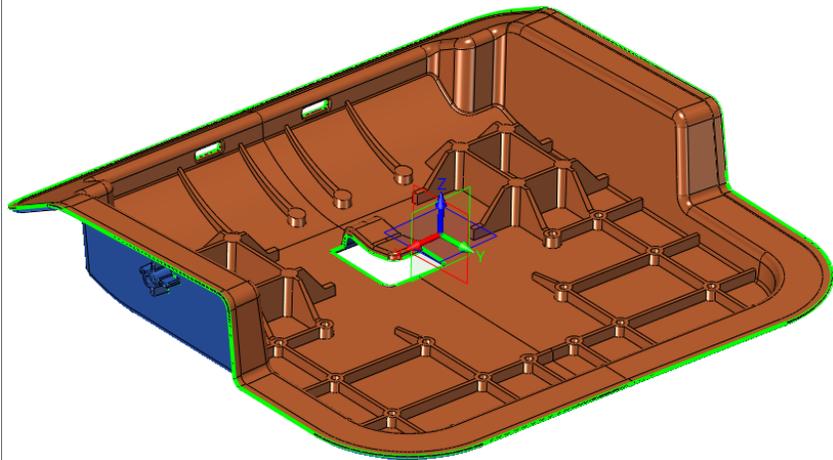
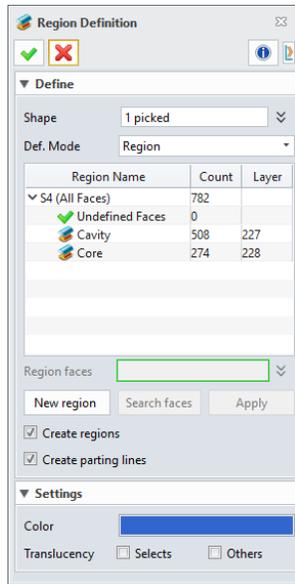
This mode uses defined parting lines to define core, cavity and other regions, no need to manually pick faces to classify.

2. Region list

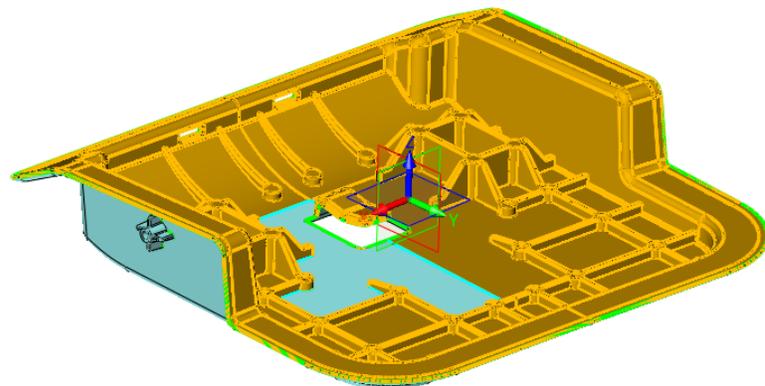
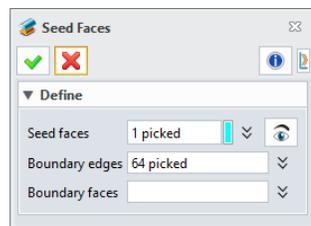
3 default regions are provided by default: Undefined, Core and Cavity. Users can define which faces belong to what region, or add more regions.

- **Under region mode**

Defined core and cavity regions will be read out after the model is picked.

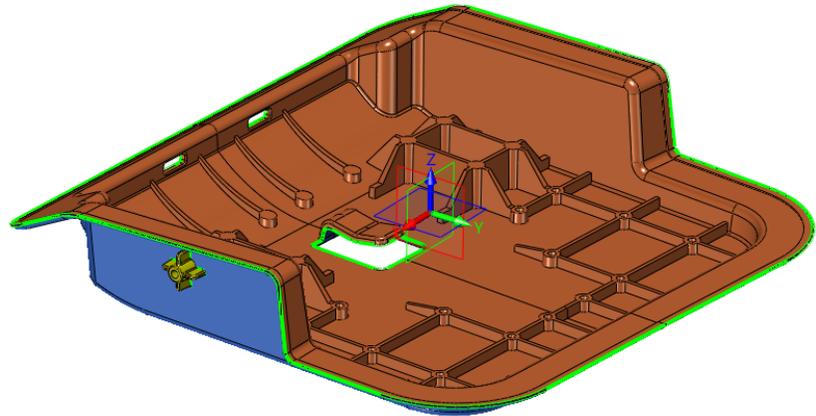
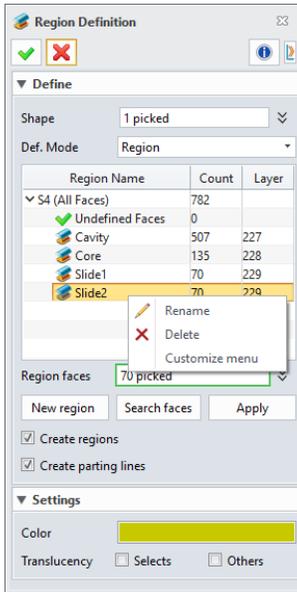


If no region is defined, users can pick the Cavity and Core region to activate the Region face field to manually pick faces to define. And the “Search Faces” button can help users pick faces by a seed face with boundary edges/faces.



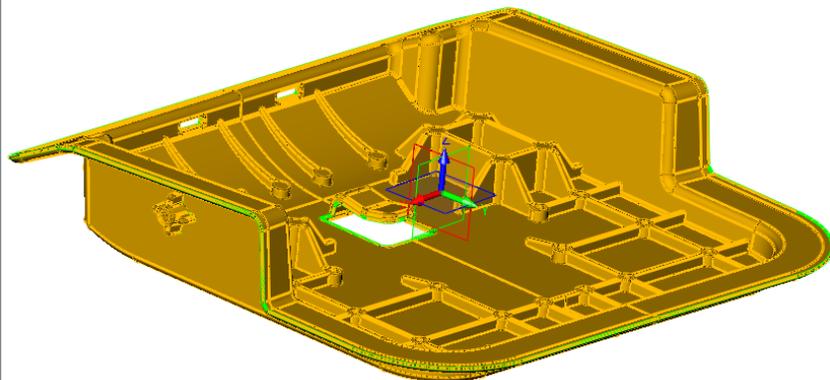
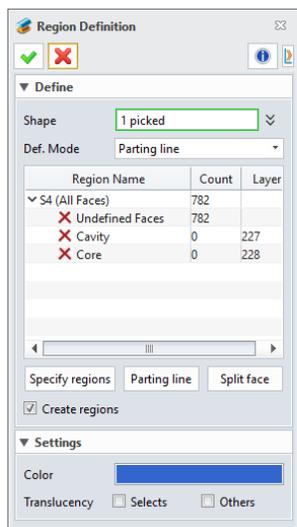
“New Region” button is available for users to create new region other than core and cavity.

All regions on the list can be renamed and deleted through their right-click menu, and have defined layers to put their own geometry which users can modify the layer ID with double-clicking.



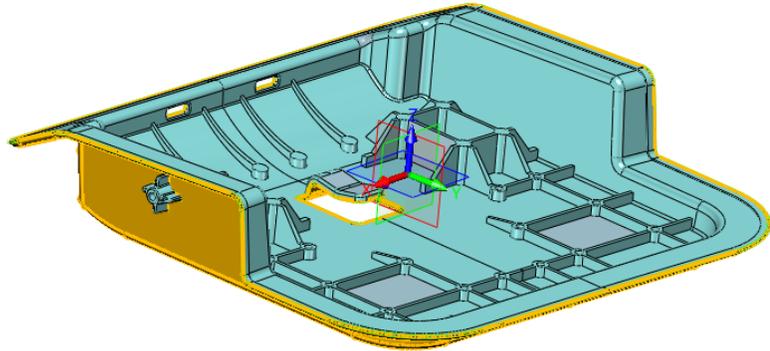
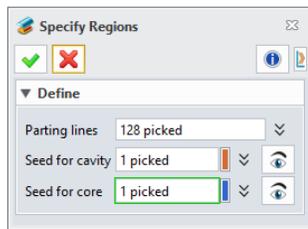
- **Under parting line mode**

Normally there is no region defined before for parting line mode. So, users need to use “Specify regions” to classify the model.

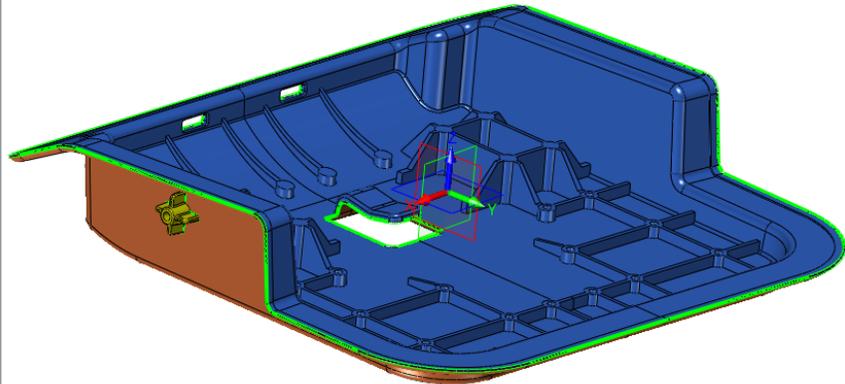
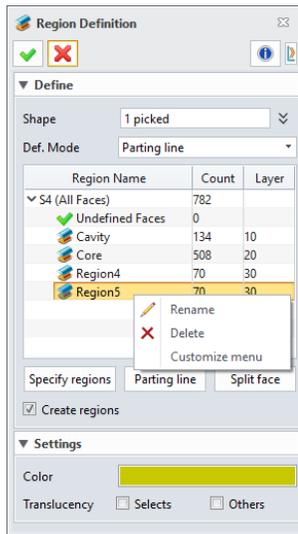


“Specify region” will pick all parting lines automatically, and use the seed faces picked separately by users for core and cavity to classify faces into different regions.

The preview button beside Seed field can be used for users to check on which faces will be picked and put into core and cavity before execution.



Faces, that can't be classified into core or cavity region, will be classified as other regions as following picture demonstrates.

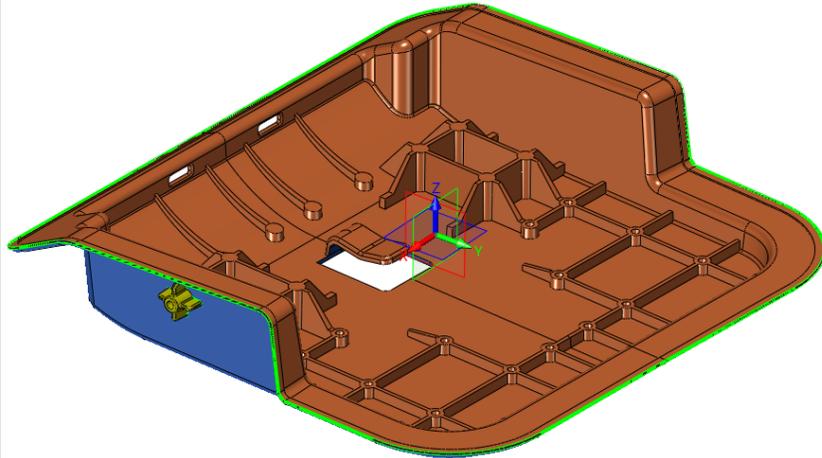
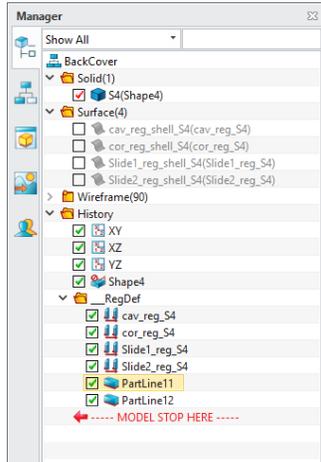


If no parting line is generated, “Parting Line” button can be used to create right away.

This parting line mode can't create new region manually, but “Specify region” can generate new region automatically basing parting lines.

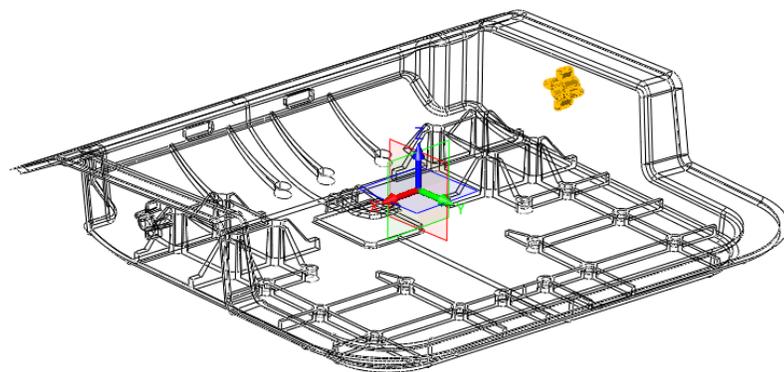
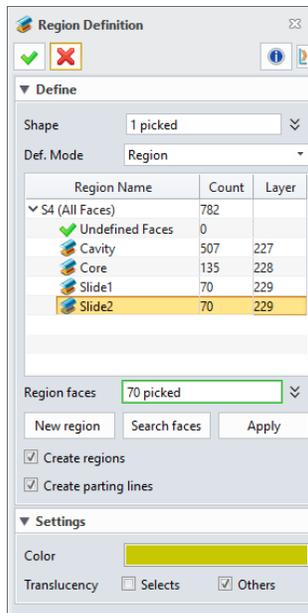
3. Extracting region shapes

After all regions are well defined, the quantity of the undefined faces should be zero. Then users can check on the “Create region” option to extract region shape copy for each region from the product model leaving the product model unchanged.



4. Region color

“Color” field can give different color for different regions, and users can use the “Translucency” to block region undesired to see.



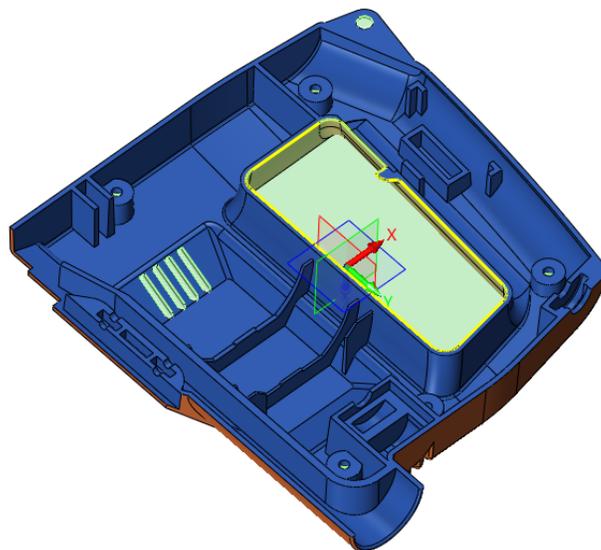
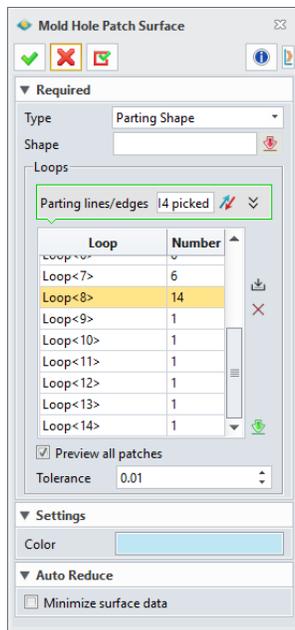
3.5.1.4 ★More Powerful “Patch”

“Patch” has done great deal of things on auto patching, instant preview, specific patch editing and workflow.

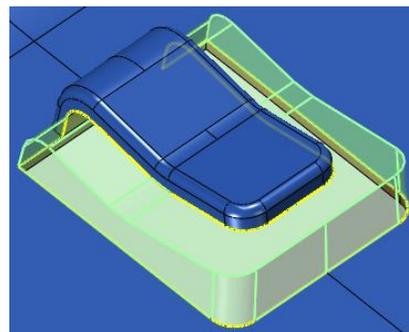
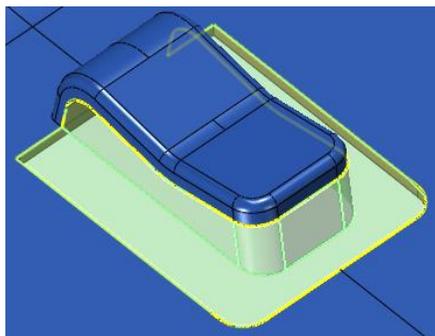
1. New patch loop list to improve the patch generation and editing

Under Parting shape mode, all inner loops will be searched and listed inside this patch loop list after the produce model is picked.

Patch faces will be automatically generated for each loop inside the list. Users can pick each loop to preview the faces or check the “Preview all patches” option to see all the patch faces.



The switch button next the Part lines/edges field can toggle which region the patch face will generate from between core and cavity regions, so that the patch face can have different form.

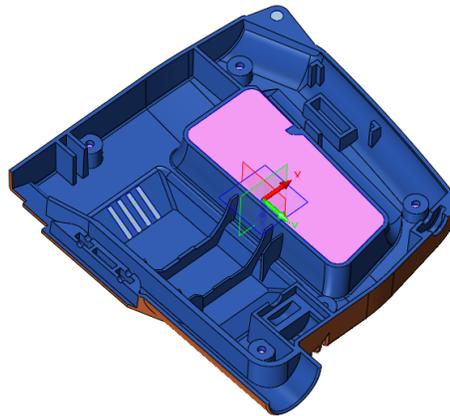
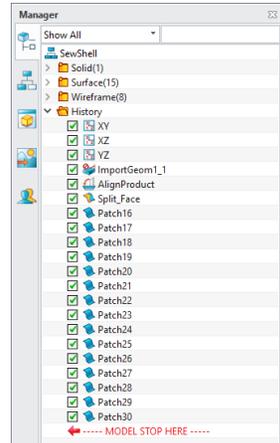


2. New “Color” field

This field can define what color the patch faces will be used.

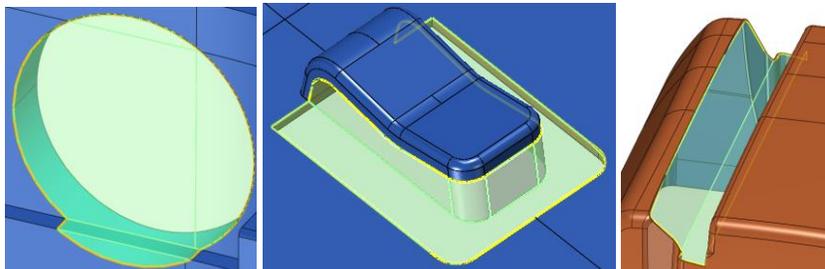
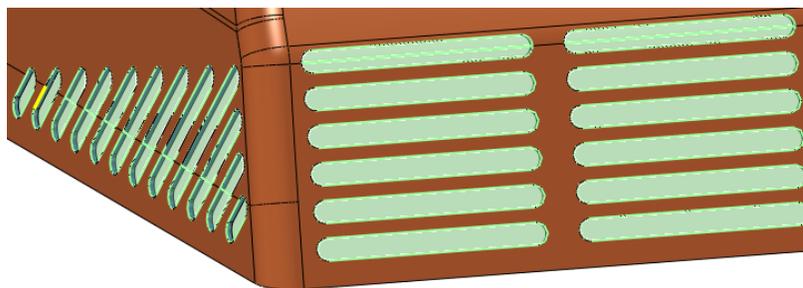
3. One feature entry on the history tree for one loop

Each patch for each loop will have one feature entry on the history tree so that users can locate each loop, and then redefine or delete separately.



4. More powerful patching result

Patch has greatly improved its algorithm to analyze various holes, slots and other openings, and implement various approaches to generate suitable patch faces for all inner loops as following samples demonstrate.

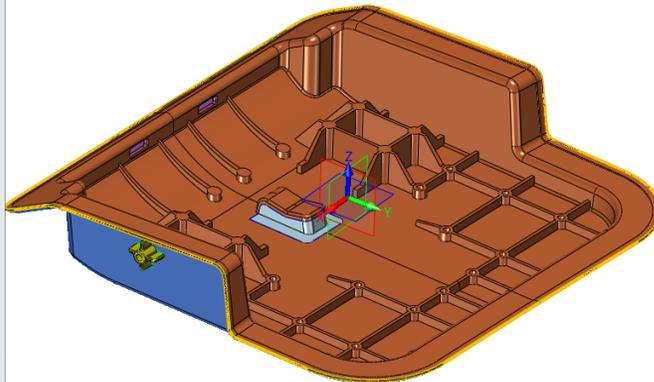
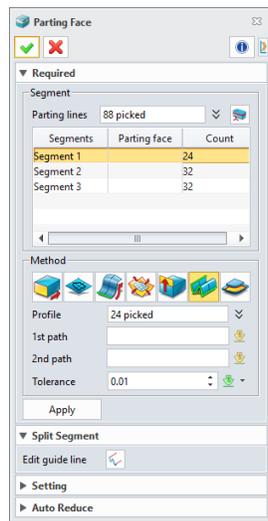


3.5.1.5 ★Blank-new “Parting Face”

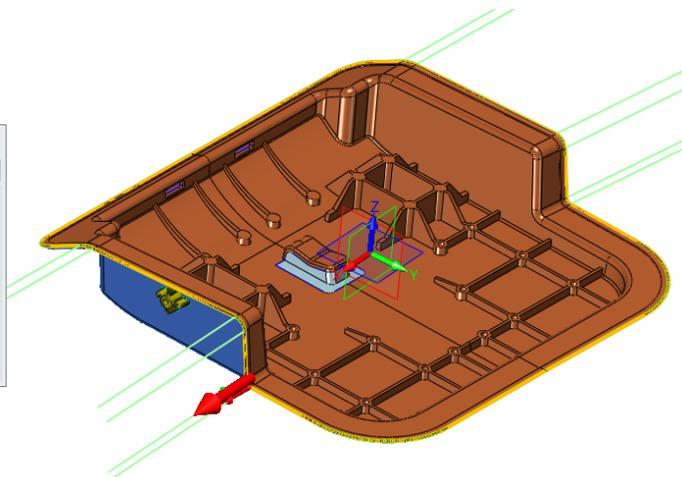
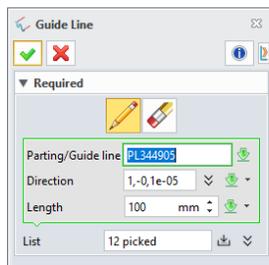
“Parting Face” have re-done completely with rich face creation methods and easy parting lines segmentation to meet quite various and complex parting face creation.

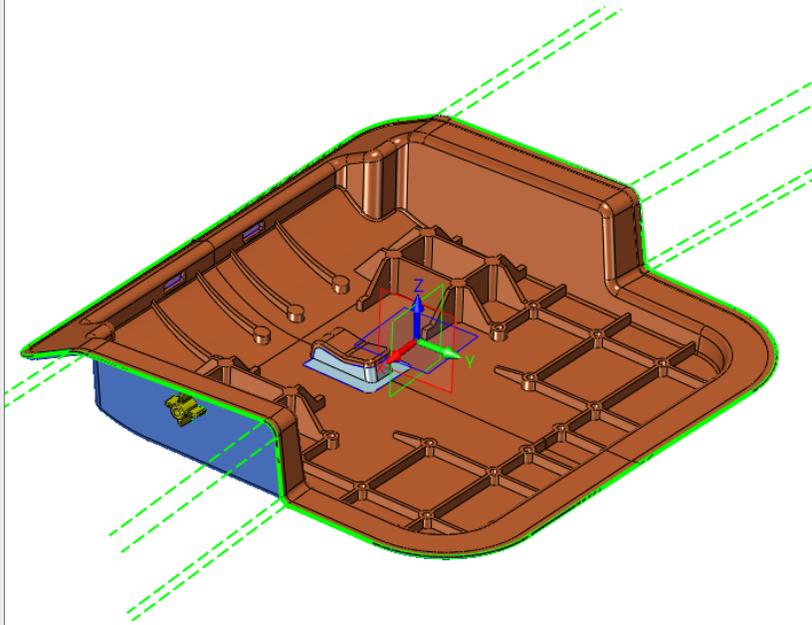
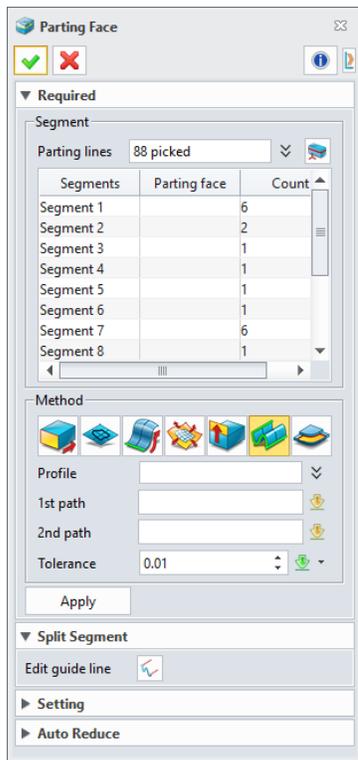
1. Parting line segmentation basing on parting face creation methods

Connected parting lines are listed as one parting segments to use one method to create parting face. Since different methods are needed in parting faces of a model, “Edit guide line” button is meant to divide connected parting lines into different segments.



On the “Guide Line” form, a guide line will be created on the end of the picked parting line with defined direction and length. Guide lines should be created if parting face creation methods are different among parting lines.



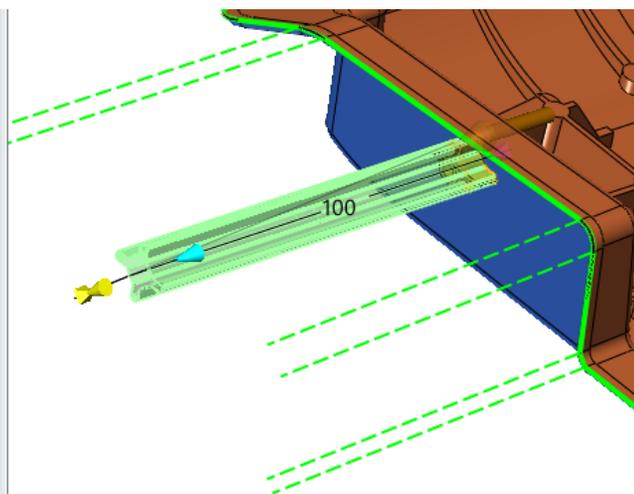
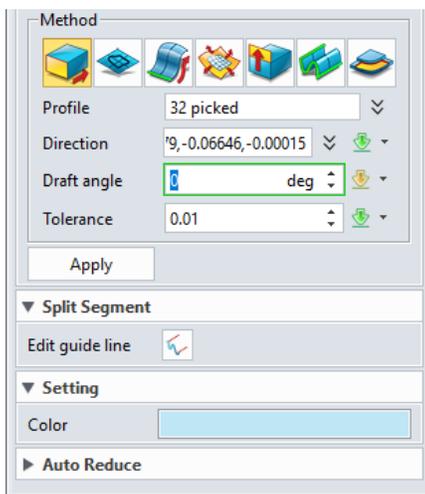


2. Using suitable methods to create desired parting faces

Several methods are provided to help users to create desired parting faces.

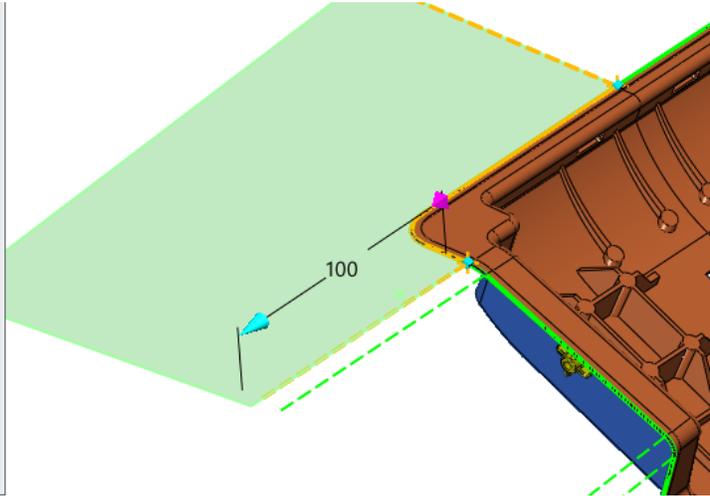
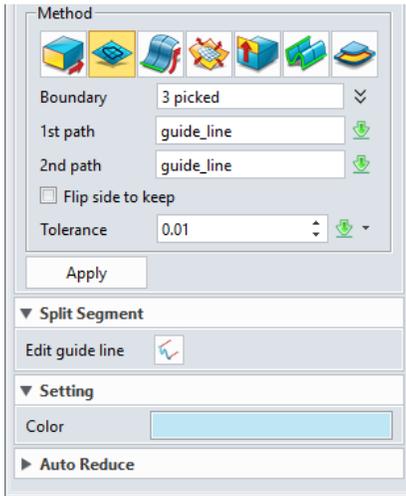
1) Extrude

Parting lines are picked as the profile and extruded with define direction and length.



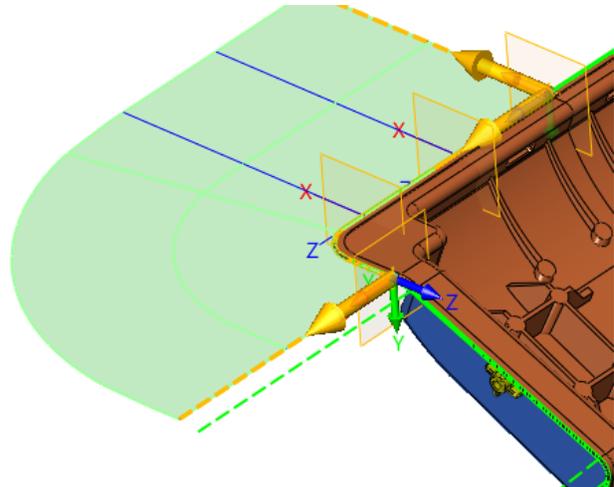
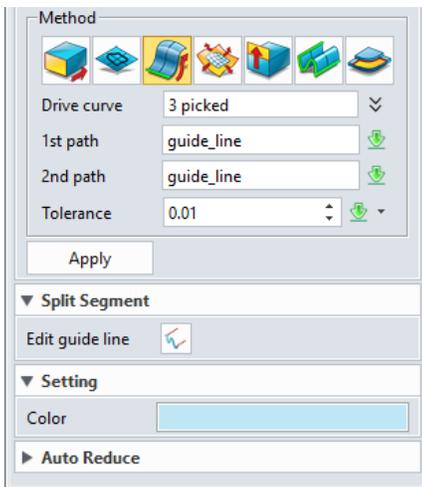
2) Trimmed Plane by Boundaries

This method only works when the parting lines and the guide lines are on one plane. A planar face will be trimmed with parting lines and guide lines as boundaries to generate as the parting face.



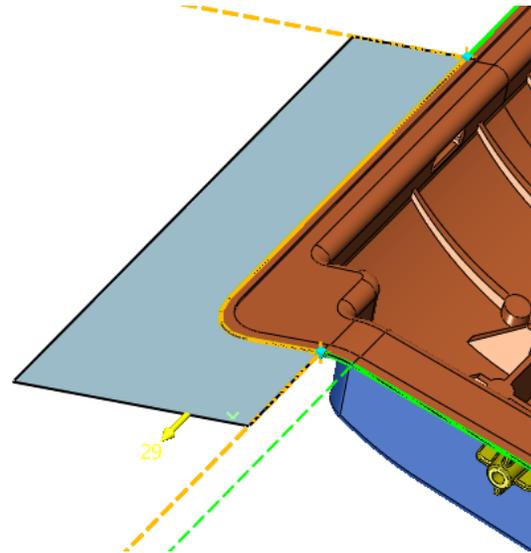
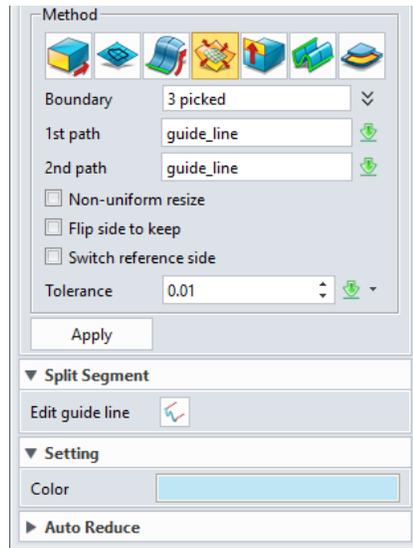
3) Spine Loft

Parting face is lofted with parting lines as profile and guide lines as paths.



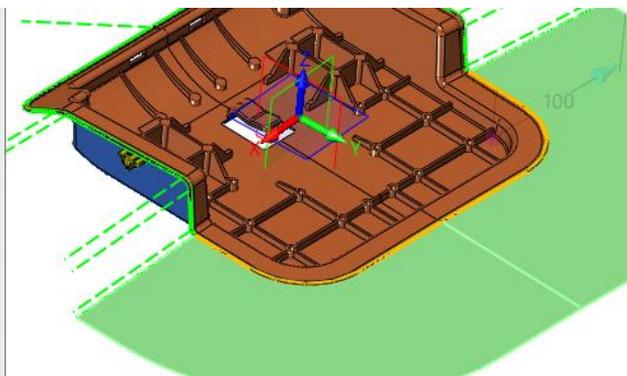
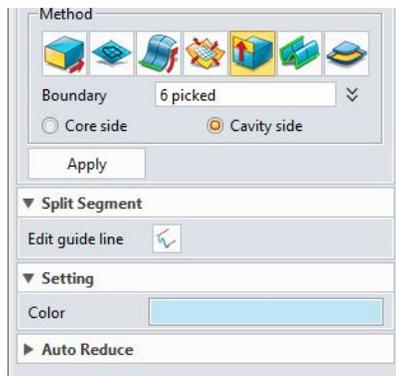
4) Enlarge and Trim Face

This method only works when all parting lines from one segment lie on one face. The face along the parting lines is enlarged and trimmed with parting lines and guide lines.



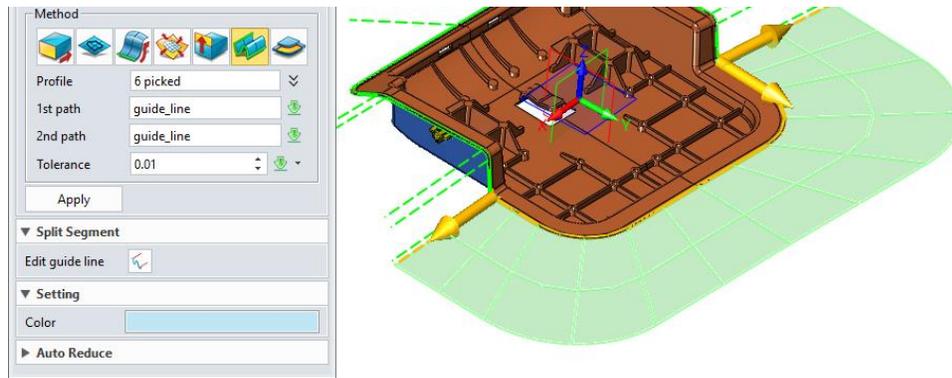
5) Extend Shape

The parting face is extended from the face along the parting lines and trimmed by the parting lines and guide lines.



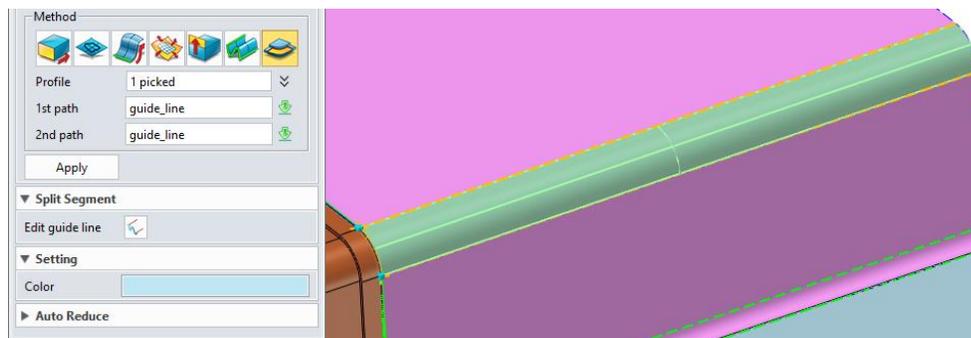
6) Extrude Edges by Rails

The parting face is lofted with parting lines as profile and guide lines as paths.



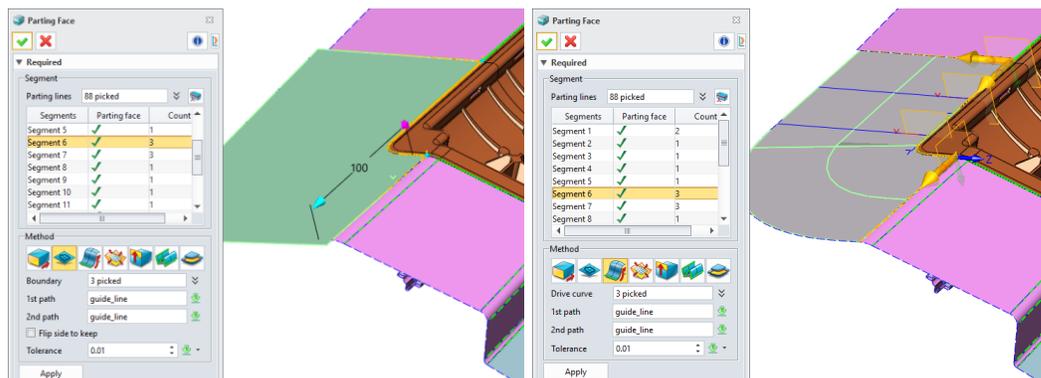
7) Bridge Face

This method only works on single parting line which is tangent with connected curves from both ends and whose connected parting lines have their own parting faces created. The parting face is created with Curve Mesh with parting lines, edges and bridge curve from adjacent parting lines.



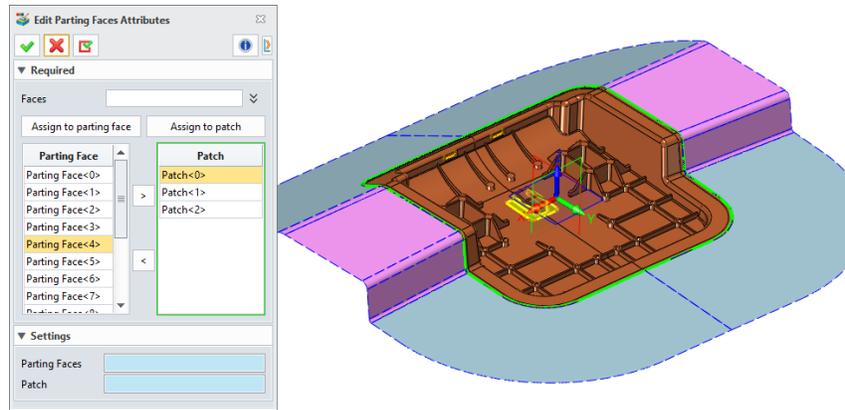
3. Parting face redefinition

Parting face redefinition should use the same "Parting Face", not the regular redefinition picked from the history tree, so that existing parting faces can update normally.



3.5.1.6 New “Edit Parting Face Attributes”

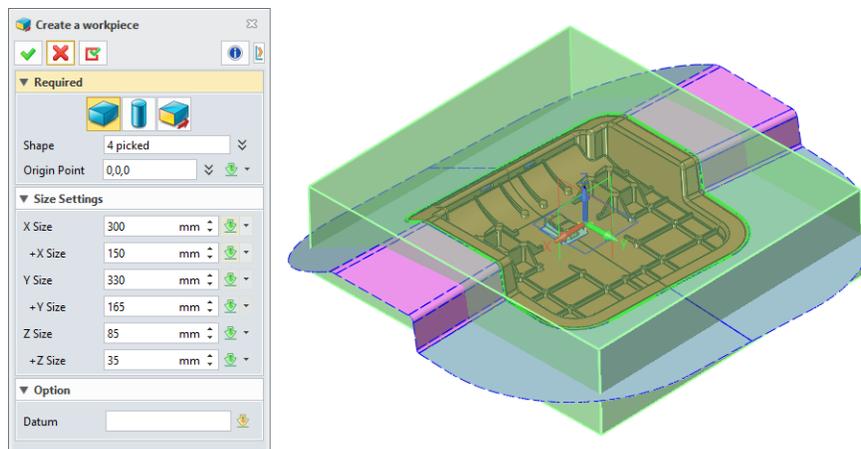
“Edit Face Attributes” can label unmarked faces as parting face or patch face as what previous commands do, or switch labeled faces between parting faces and patch faces. Only labeled faces can be identified by “Trim” command, so before trimming workpiece with parting faces into core and cavity, please use this command to label related faces. Only open shape can be selected for parting face can’t be solid shape.



3.5.1.7 Adjustments in “Workpiece”

A few adjustments have been done in Workpiece to meet the new mold design workflow.

1. Auto roundup to integer
2. Auto Mark the shape with the workpiece label
3. New layer (ID 233) is automatically created to contain the workpiece.



3.5.1.8 Changes in "Trim"

"Trim" now can select workpiece, paring faces automatically according to the labels attached to the shapes, then divide the workpiece into core and cavity, and slides.

Trim to Cavity and Core

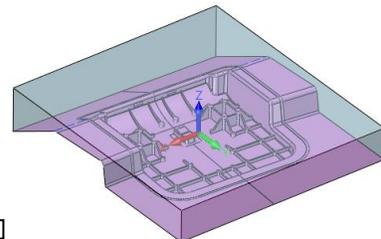
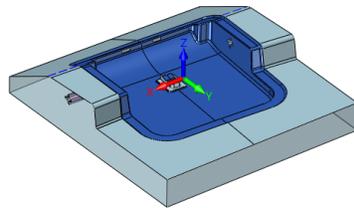
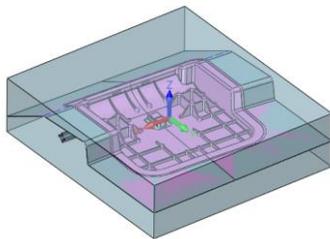
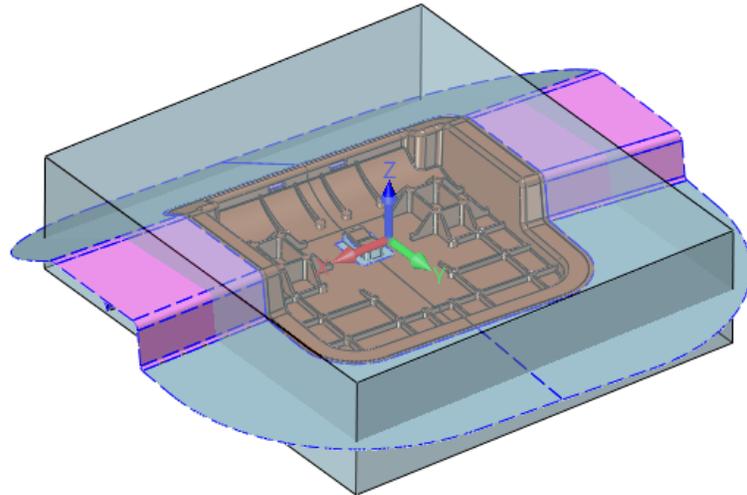
Required

Region	Count
<input checked="" type="checkbox"/> Cavity	18
<input checked="" type="checkbox"/> Core	20
<input checked="" type="checkbox"/> Slide1	2
<input checked="" type="checkbox"/> Slide2	2

Merge planar faces

Tolerance

Tolerance: 0.01 mm



Only selected shapes will be displayed when clicking on the region list for better observation.

Trim to Cavity and Core

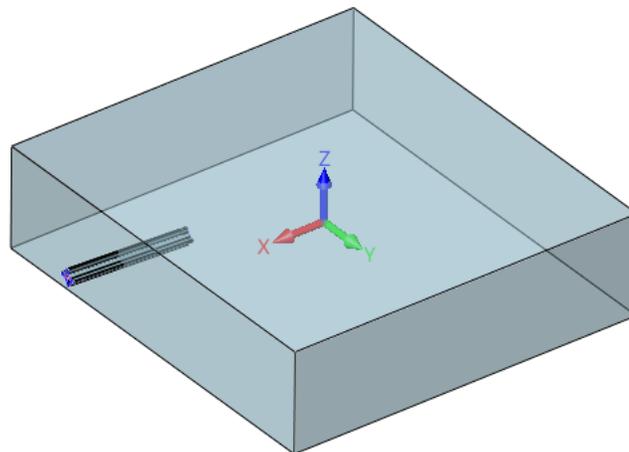
Required

Region	Count
<input checked="" type="checkbox"/> Cavity	18
<input checked="" type="checkbox"/> Core	20
<input checked="" type="checkbox"/> Slide1	2
<input checked="" type="checkbox"/> Slide2	2

Merge planar faces

Tolerance

Tolerance: 0.01 mm



3.5.2 Electrode Improvements

3.5.2.1 Reformed “Electrode Blank”

“Electrode Blank” optimized its workflow with following details:

1. Simplified electrode selection

Only a planar face from an electrode needs to pick to define what electrode to operate and how it should orient.

2. Custom blank origin

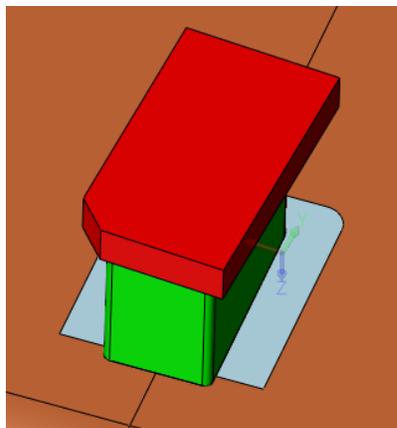
The new “Custom origin” option can activate the XYZ field of the origin to allow users to input.

3. New “Stock Size”

Stock size is composed of the stock base size and the offset value defined by “Blank greater than Base in single side” and “Blank greater than Base in Height” on “Blank Configuration”, or directly from the specified size from the standard size also defined by “Apply Standard Size” option on “Blank Configuration” form.

4. New color settings

New “Base color” and “Tip color” as following:



5. New “Blank Configuration”

“Blank Configuration” can define all default valued used by “Electrode Blank” form.

- 1) “Assembly” option defines which way the electrode blank should be generated
- 2) “Z0 Auto Offset” option defines the offset value of the origin on the Electrode Drawing.
- 3) 3 Round methods for electrode base:

Round: 4.25 -> 4, 4.55 -> 5

Round up: 4.25 -> 5, 4.55 -> 5

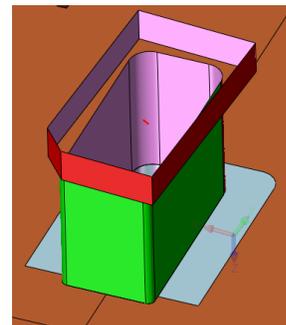
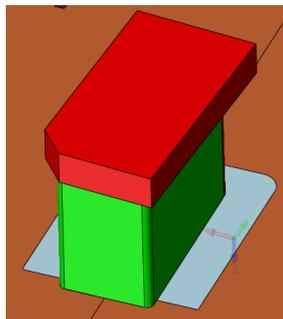
Round down: 4.25 -> 4, 4.55 -> 4

Base size = rounding tip size + “Base increase in single side”

- 4) “Base increase in single side” defines the increment of each side of the electrode base from the rounded tip size.
- 5) “Blank greater than Base in single side” defines the increment of each side of the electrode blank from the base.
- 6) “Blank greater that Base in height” defines the increment of the height of the electrode blank from the base.
- 7) “Apply Standard Size” defines the Stock size should use the standard size which is close to the base size.

Standard size is defined in installation directory > supp > ElectrodeCopper.txt,
ElectrodeGraphite.txt.

- 8) “Base Bottom Transparent” option has following effect.



3.5.2.2 Reformed “Electrode Drawing”

“Electrode Drawing” provides more options to help users to specify how their EDM drawings are generated.

1. New “Method” to define which electrode should generate drawing

3 ways are provided:

Default - manually pick electrodes from graphic area to create drawings

By layer - layer list is provided to pick which layer to create drawings

Name - name list is provided to pick which electrodes to create drawings

2. More new options to define details of the EDM drawing

1) New “MOLD NO.”

2) New “Name of Wp”

3) New “EDM Dimension style” to define dimension to use Ordinate or Linear form.

4) “New isometric view” to generate an isometric view within the EDM drawing

5) New “Auto Drawing Configuration” form to setup view and dimension details inside EDM drawing

Fields inside View Layout are meant to define the exact XY location of every projection views basing the ordinate of the drawing.

Options inside EDM Style are meant to define the dimension text style.

3. New “Center of WP” in Frame

This new option provides a Z0 field to define the zero point in Z axis.

4. New “Drawing Template”

Users can use these options to define which template should be used.

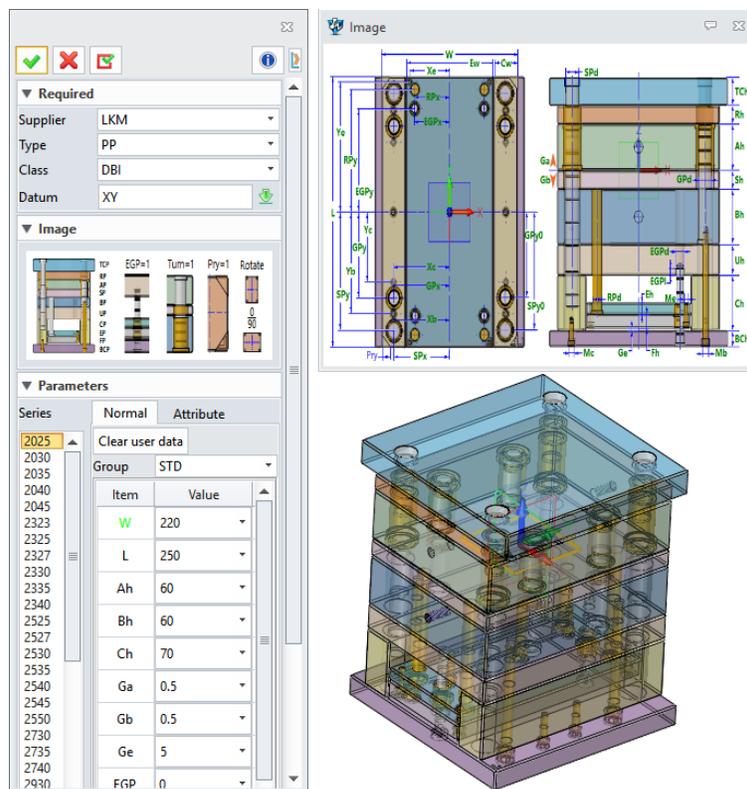
3.5.3 Reformed Standard Library

Mold library commands for mold standard parts have re-implemented with the “Part Publisher” (i.e. part table) function so that those standard parts can be more flexible to customize and easier to edit with Excel.

All standard files under installation directory > ZWMold have updated. If users want to add your own standard parts into ZWMold, please use the new “Part Publisher”.

3.5.3.1 Updated Mold Base to Support Customization

This updated Mold Base can greatly support customization during insertion.



1. Instant model generation

As soon as the MoldBase starts, the 3D model will be generated instantly within graphic area, so that users can observe the model closely and clearly.

When changing the model type, the 3D model will be refreshed right away. And if some parameters have been modified, the “Refresh” button can update the model.

2. New “Image” to show different structure of various model

When users change the option of the Type field, the image will be refreshed at once to show corresponding picture. And the popup image on the right will show specific parameter of very detail.

3. New “Clear user data” button

This button can restore the custom value users put into specific parameters into default.

4. New “group” to control the content of the parameter list

The parameter list becomes very long since most of the parameters are listed out for users to customize. Users can use options from the group to change what parameters to see.

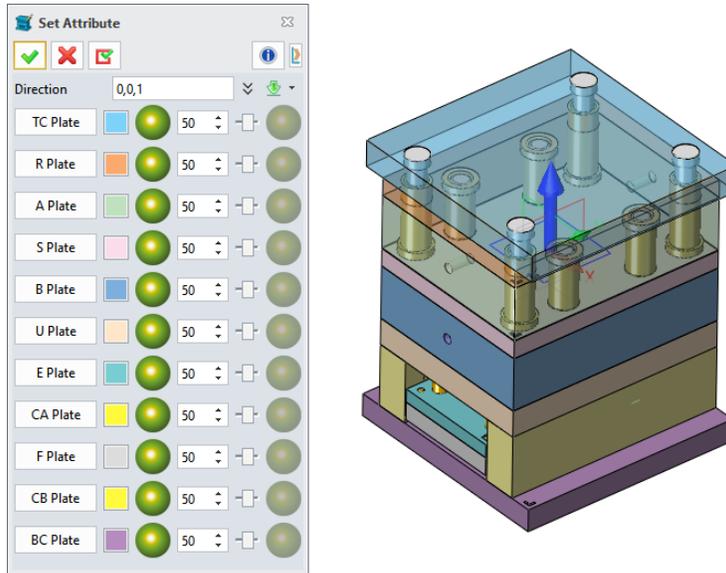
5. Blank-new parameter list

Parameters now are classified into 2 types:

- Key parameter - marked with the key icon and its value can only pick from the defined list
- Custom parameter - marked with pen icon and its value not only can pick from the defined list, but also accept new input. Customized parameter will be marked green to identify itself.

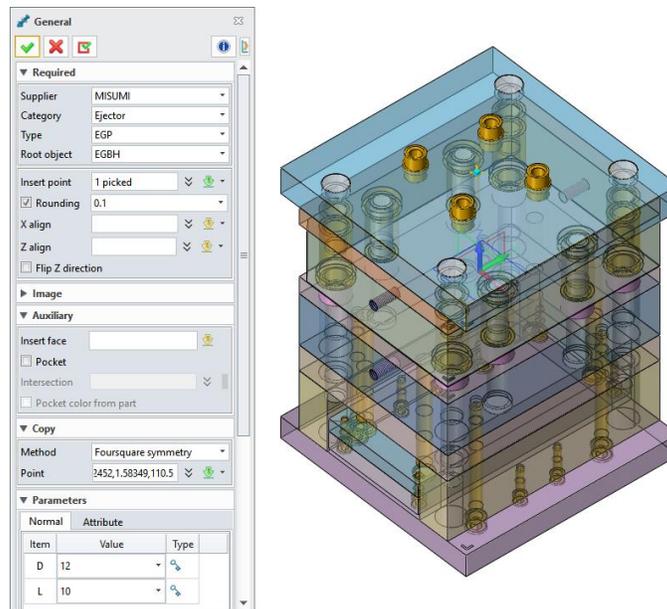
3.5.3.2 New "Set Attribute"

"Set Attribute" command is meant to set up regular mold base attribute into imported moldbase model, so that the model can be used as native moldbase.



3.5.3.3 Updated Mold Standard Part Commands

Same as the MoldBase command, all mold standard part commands, such as General, Screw, Epin and so on, have updated with the new "Part Publisher". Here are the major changes:



1. New “Rounding” option

Rounding is meant to round down the coordinates of insert points to expected values as following:

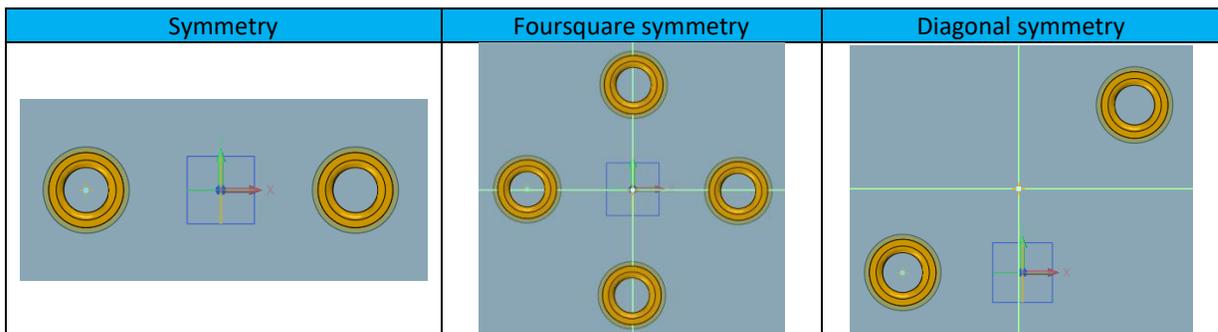
- 0.01 is to round down the value with only 2 decimal number left
 $21.057 = 21.05$; $21.979 = 21.97$
- 0.1 is to round down the value with only 1 decimal number left
 $21.0877 = 21.00$; $21.7713 = 21.70$
- 1 is to round down the value to an integer
 $28.999 = 28.00$; $65.000 = 65.00$
- 2 is to round down the value to an integer which is close to the multiple value of 2
 $28.999 = 28.00$; $65.001 = 64.00$
- 3 is to round down the value to an integer which is close to the multiple value of 3
 $28.999 = 27.00$; $65.001 = 63.00$

2. New orientation option for inserted component

“X Align” can orient the x axis of the inserted component to the defined direction. Same things can be done by “Z Align”.

3. New “Copy” to multiple insert

kinds of multiple insertion



3.6 Drafting Design

3.6.1 View Projection Improvement

3.6.1.1 ★New Projection Source Settings

When picking a 3D model to project a view, all data will be included by default. Now new added “Object select mode” can allow users to select which components or shapes will be included or excluded from view projection. 3 ways are provided.

1. Exclude mode

This mode records which components or shapes are picked to exclude from view projection, so that no matter how the 3D model changes, updated views will always exclude the picked objects.

2. Include mode

This mode records which components or shapes are picked to only include in view projection, so that no matter how the 3D model changes, updated views will always only project the included objects.

3. By layer mode

This mode records which layers and which entities from picked layers are included in view projection.

Inside the object list, the “Hide component” from the right-click menu of picked items can define which one is excluded. For projected views, if preferred to change which items are excluded or included, users can do that on the view’s attribute form.

→ Where it is

Drafting context > Standard view > Settings > Comp > Object select mode

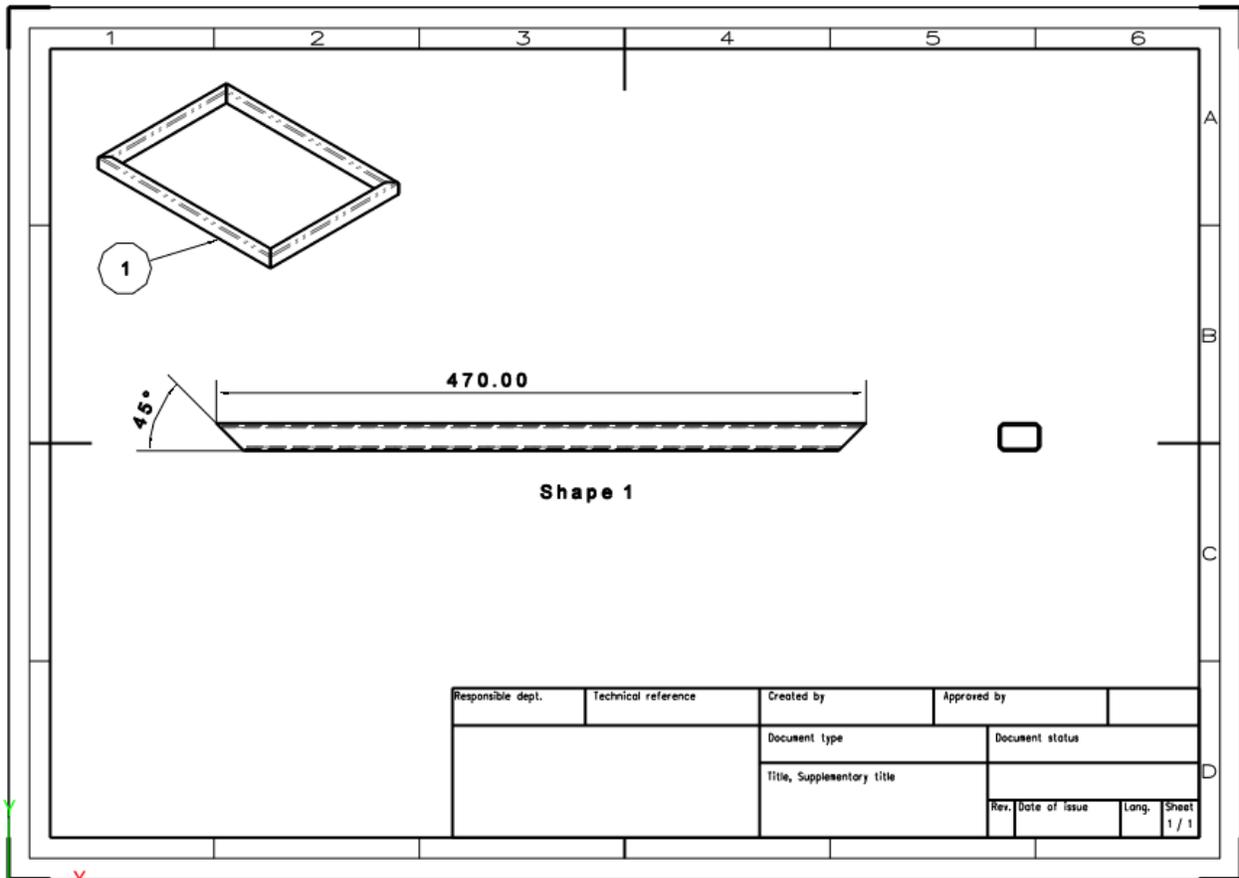
Drafting context > Layout view > Settings > Comp > Object select mode

Drafting context > Right-click on views > Attribute > Comp

3.6.1.2 ★Projection Views for Picked Shapes

Normally view projection is from part or assembly objects, but now it can work on picked shape, one or multiple, so shapes can have its own individual views.

There is a new “Show shape” option on the Comp tab which can show all shapes from the picked object. Users can pick unwanted shapes to exclude from projection through the “Hide component” option from the right-click menu, so that specified shape can have its own projection.



➔ Where it is

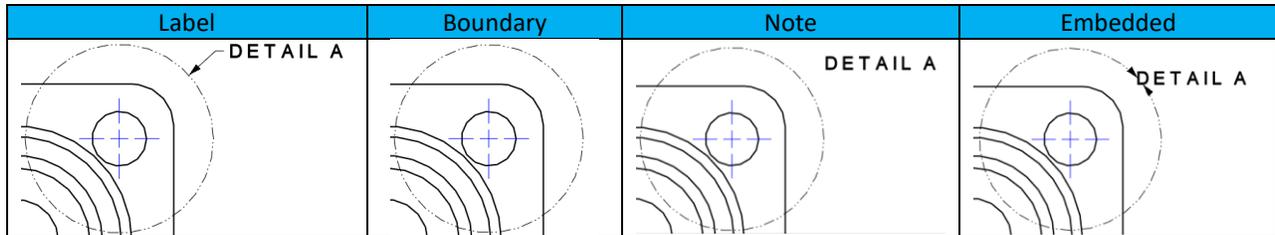
Drafting context > Standard view > Settings > Comp > Object select mode

Drafting context > Layout view > Settings > Comp > Object select mode

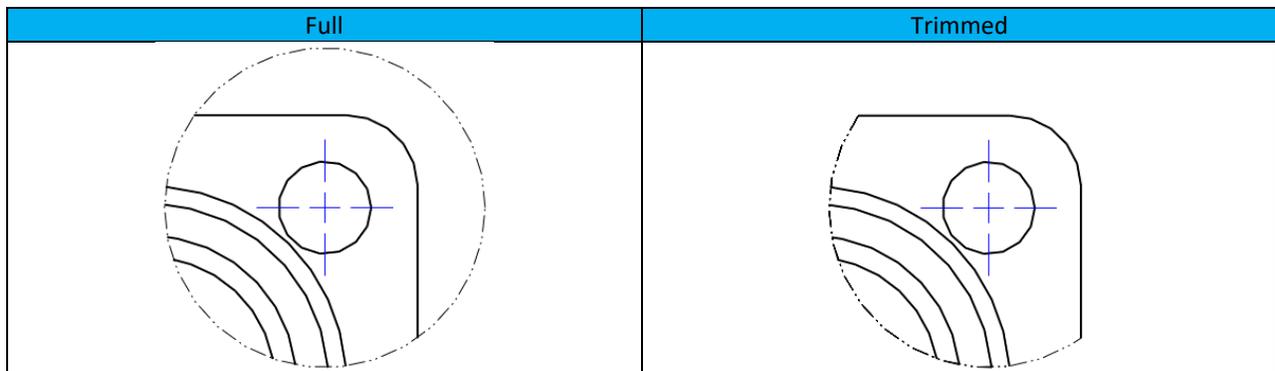
Drafting context > Right-click on views > Attribute > Comp

3.6.1.3 More Boundary Styles in Detail View

Here are the new boundary styles:



New “Full outline” options to trim the view boundary as needed.



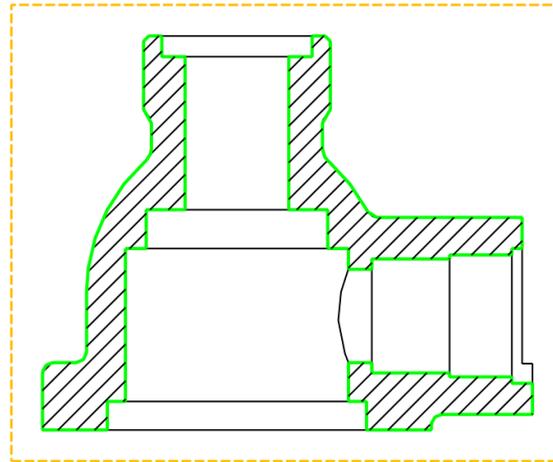
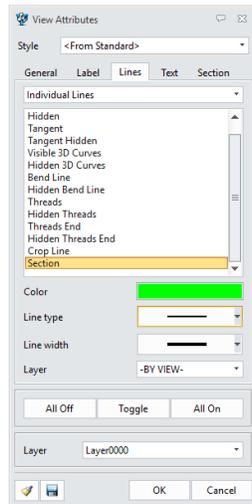
➔ **Where it is**

Drafting context > Layout Ribbon > Detail View > General

Drafting context > Right-click on Detail view > Redefine

3.6.1.4 New “Section contour line” in Section Line

There is a new line class “Section contour line” in “Section view > Attribute > Lines tab” to allow users to define different line type with different color and width to outline the section profile, so others can identify the section profile instantly.

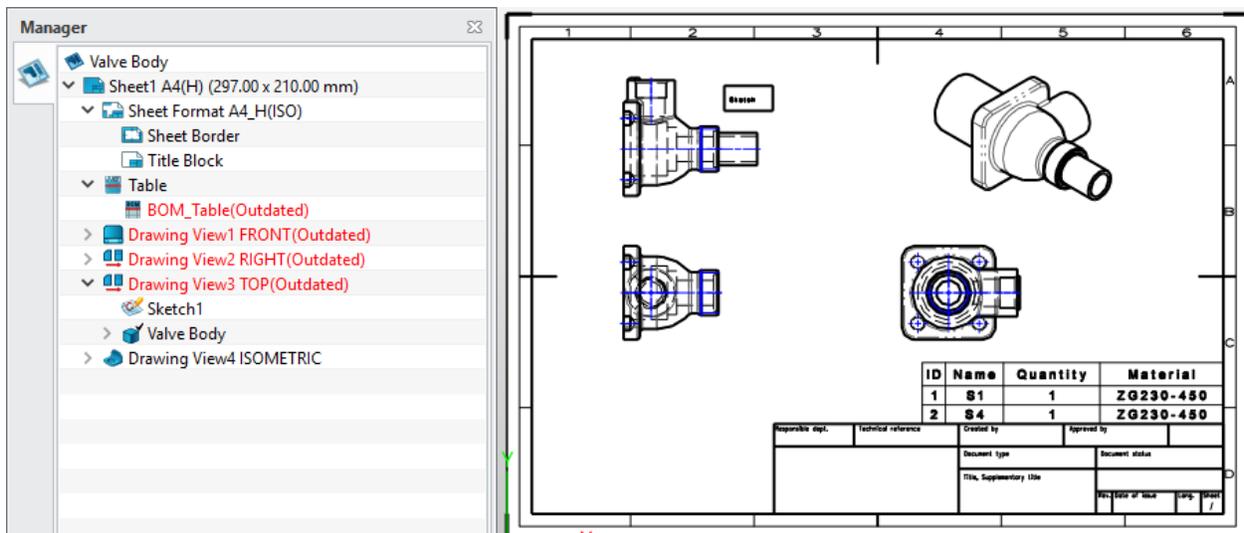


→ Where it is

Drafting context > Right-click on Section view > Attribute > Lines tab

3.6.1.5 New "Outdated" mark in Sheet Manager

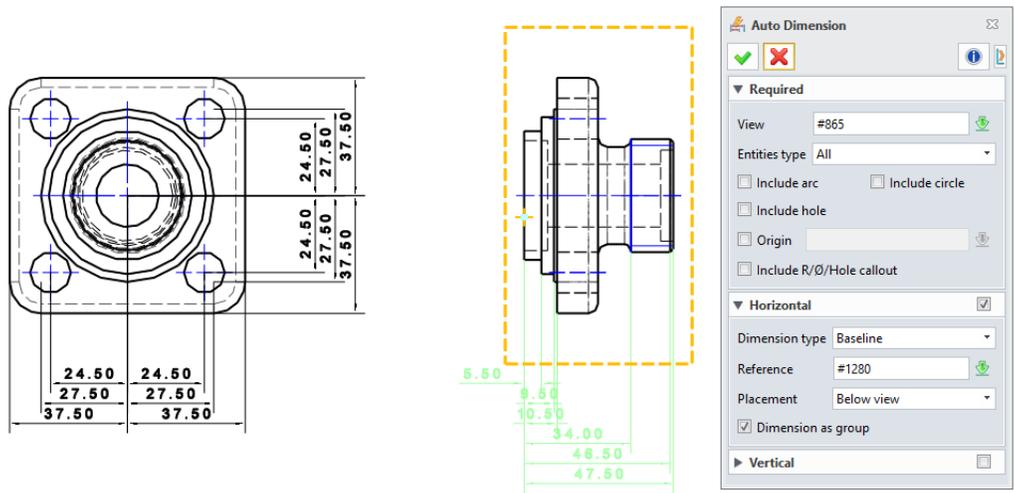
There is a "Outdated" mark on view, table sketch nodes on the sheet manager tree to mark which object is not updated when the 3D object is modified, to guide users to update them individually.



3.6.2 Enhancements in Dimension

3.6.2.1 ★New “Auto Dimension”

“Auto Dimension” can generate linear dimension, Radial/Diametric dimension, hole dimension on picked entities or entire entities from a picked view to help users to finish the dimensioning work quickly.



1. Entities to auto dimension can be very adaptive to various purpose

“Entities type” field provides 3 options:

- All - all entities of the view will be dimensioned
- All arc/circle - only arc or circle entities of the picked view is dimensioned.
- Picked - only picked entities will be dimensioned.

2. Dimension type can be pointed for auto generated dimensions

Baseline dimension, continuous dimension, ordinate dimension, label ordinate dimension, Radial/Diametric dimension and hole callout can be generated by this “Auto Dimension”. And since some of the dimensions are group dimension type, “Dimension as group” option can break the group dimension into single dimension type.

3. A point can be the origin or 2 vertical lines can be the origin

An origin should be defined for Baseline dimension, continuous dimension, ordinate dimension, label ordinate dimension. “Origin” field can define a point as datum for XY directions, or “Reference” defines the X and Y direction individually instead.

4. Dimension placement can be pointed

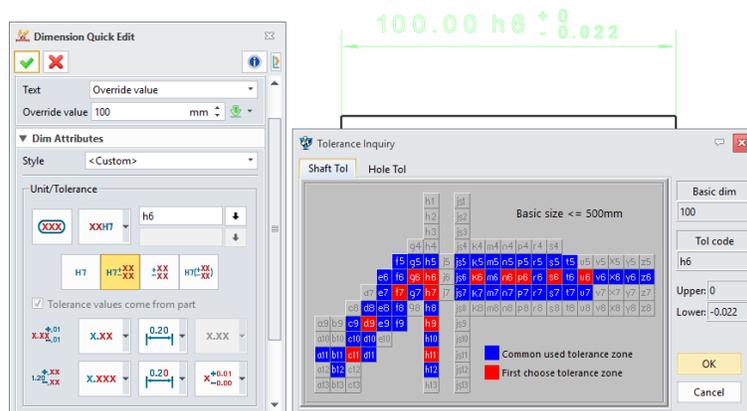
Auto generated dimension can be placed on the 4 directions of the view.

→ **Where it is**

Drafting context > Dimension Ribbon > Auto Dimension

3.6.2.2 ★New “Override Value” in Dimension Text

This new “Override Value” can replace the original dimension value with a defined number or a referred variable keeping all dimension attributes imposed on the original value untouched. The difference to the “User Text”, which makes the input characters as string, is that “Override Value” works as a new numeric value replacing the old one so that all attributes for the old numeric value can work the same. Such the tolerance zone setting before can work for the override value.



→ **Where it is**

Drafting context > Dimension Ribbon > Linear dimension > Settings > Text

3.6.2.3 Improvements in Dimension Attribute

Basing on feedback, a few of improvements have been done for dimension attributes as following:

1. New “Inclined Angle” in Linear dimension

There is a new option “Inclination of Angle” in linear dimension attribute to tilt the dimension extension lines away from the dimensioned entities to better distinguish each other.

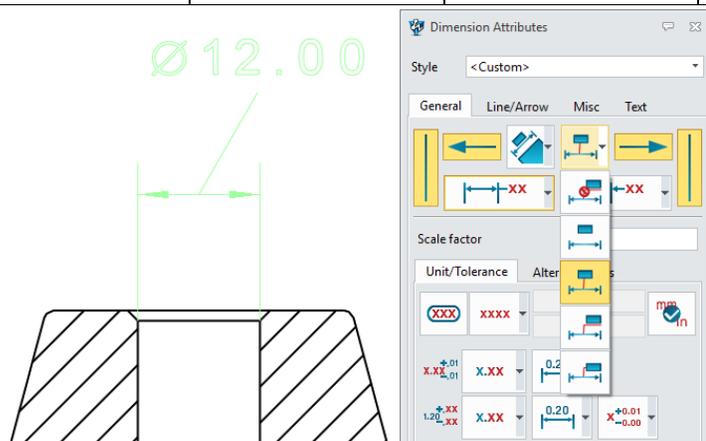
Users can directly drag the dimension extension lines to make the angle.



2. New leader for narrow dimension to avoid overlap

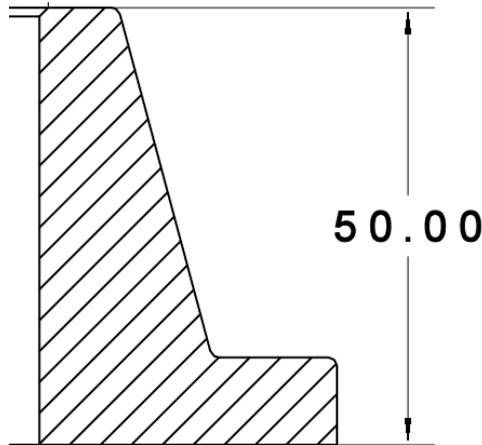
When there is no enough space between the dimension extension lines to accommodate dimension text, users can use this new dimension leader to move the text away from the dimension line, so that there is no overlap occurring. 5 ways are provided:

None	No leader	With leader	On leader	After leader



3. New “Perpendicular text” for text orientation

This new option can make the dimension text always perpendicular with the dimension line.



4. New “Tolerance height ratio”

There is a new “Height ratio of tolerance text” to allow users to define how height the tolerance text is comparing to the dimension text.

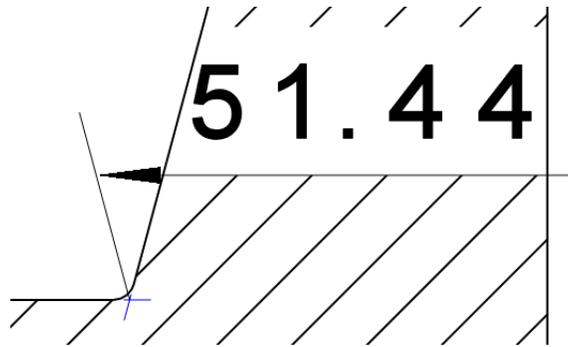


→ Where it is

Drafting context > Dimension commands > Settings > Dimension attribute > Text

3.6.2.4 New “Intersection Symbol”

“Intersection Symbol” can mark the intersection point of 2 picked dimensions so that other dimensions can use it as object to dimension more. “Intersection Symbol” only work on line/line, line/arc and arc/arc intersection.



→ Where it is

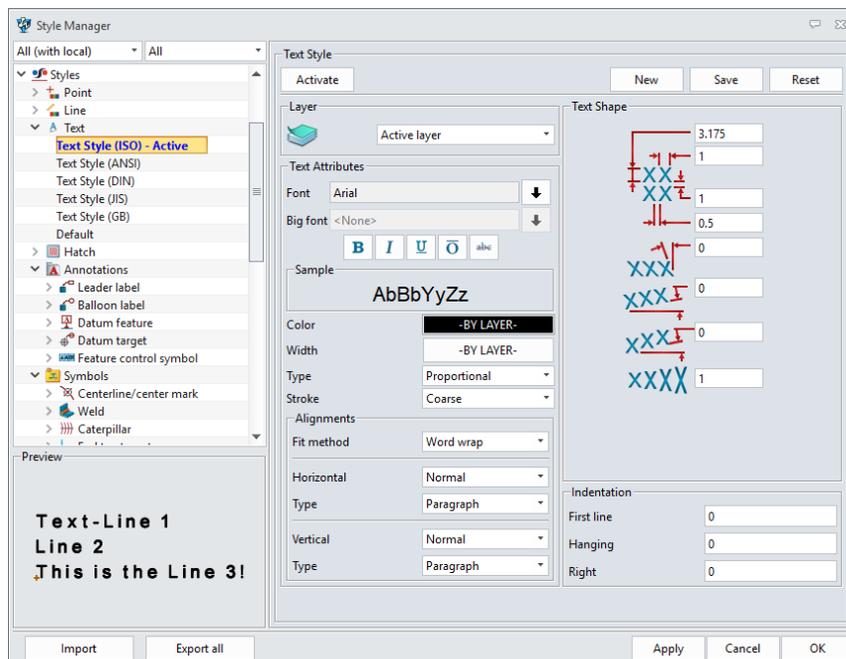
Drafting context > Dimension ribbon > Symbol > Intersection Symbol

3.6.2.5 ★Changes in “Text”

There are a few things enhanced in “Text” object:

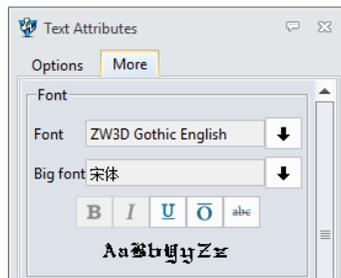
1. Style manager can create different text styles to reuse

Text in style manager now can define its styles so that dimension, views or tables can use them to control their own texts easier.



2. New “Big font”

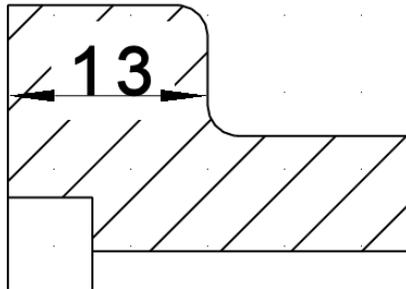
This “big font” defines a font for characters that are not supported by the “Font” so that all the characters can be displayed as desired. The “Big font” field can be available only when the “Font” field is defined with ZW3D own fonts.



ZW3D 2019
中望3D 2019

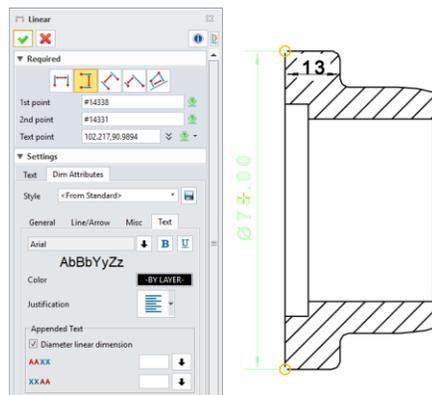
3. Text over hatch

When text is on top over the hatch area, it won't be blocked.



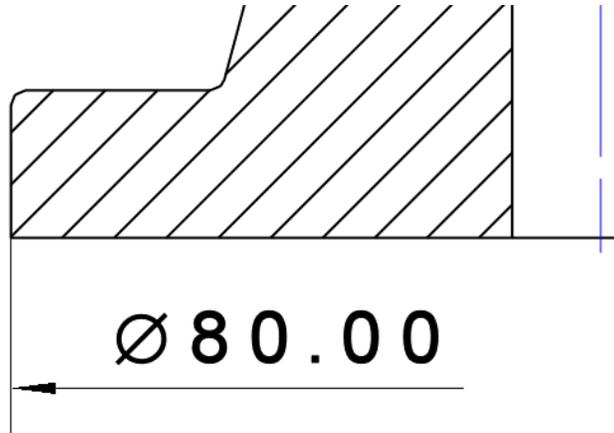
4. New “Diameter linear dimension”

This option can put the diameter symbol Φ and the dimension value together as a whole without mixing with the prefix.



3.6.2.6 ★New “Symmetry Dimension”

“Symmetry Dimension” normally work on symmetric part, especially on symmetric views. Users only need pick the entity and a center line to generate. Any line can work as centerline. For lines projected from edges of revolving faces, the symmetric dimension will automatically have Φ symbol and no need pick any centerline.

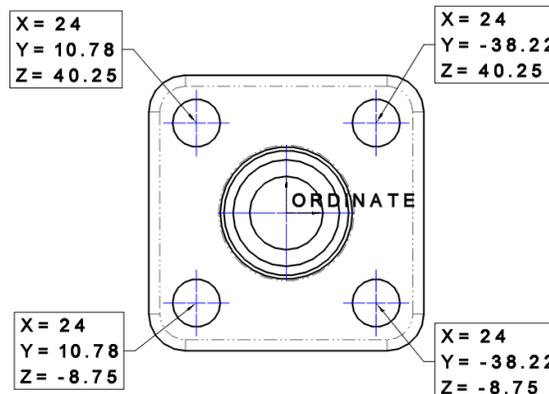


→ Where it is

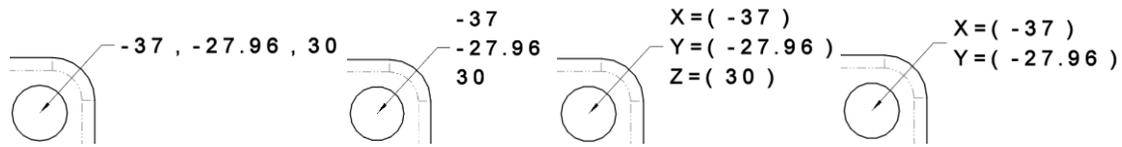
Drafting context > Dimension ribbon > Symmetry dimension

3.6.2.7 ★New “Label Coordinate”

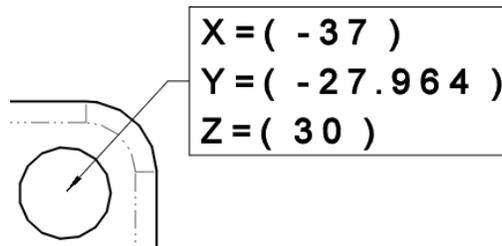
“Label Coordinate” can read out the XYZ coordinates of picked points according to the picked origin.



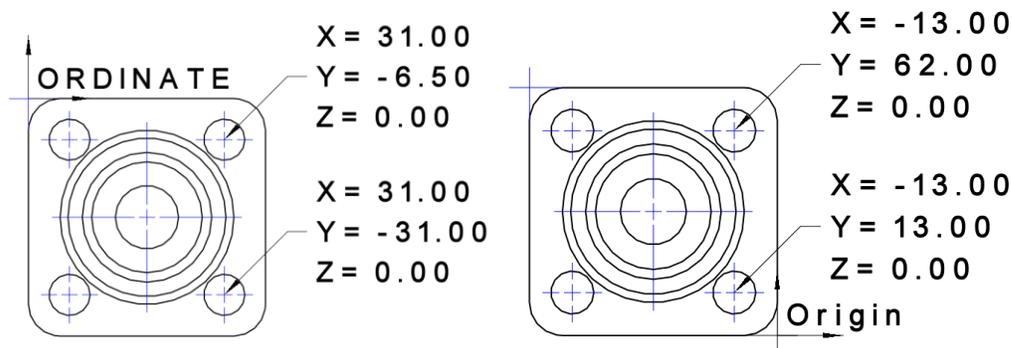
1. 2 ways to place the text
2. Besides the XYZ value reading out automatically, users input any extra characters into the label, and can edit how the text should be arranged.



3. the dimension style of this “Label Coordinate” shares the defined style of common “Label”, but has additional options “Decimal” and “Bounding box”.



4. The origin of this “Label Coordinate” will be marked automatically, and the origin text can be changed with double-clicking. Dragging-drop the origin mark to another placement can change update the XYZ values of its label dimensions. Deleting the origin mark will erase all its label dimensions.



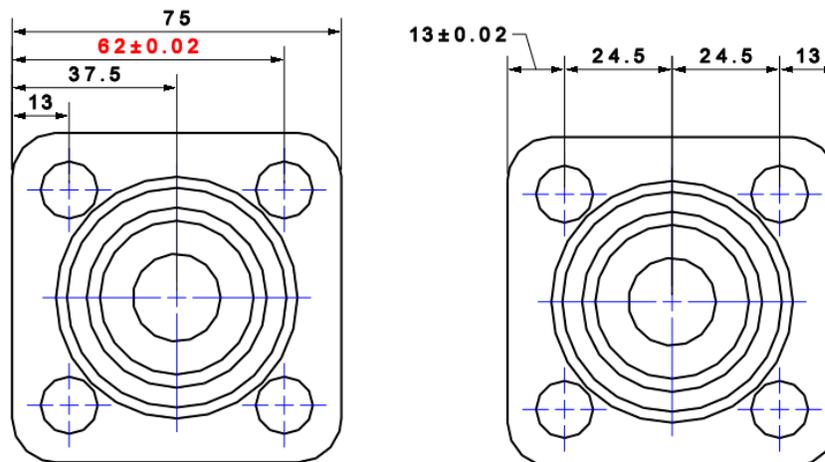
➔ Where it is

Drafting context > Dimension ribbon > Label Coordinate

3.6.2.8 Changes in “Baseline dimension” and “Continuous Dimension”

Dimensions generated by “Baseline dimension” and “Continuous Dimension” are within a group. Now the relation between group and dimension members has improved as following:

1. Dimension members can be selected individually, or multiple picked with Ctrl key. When a member is picked with the Shift key holding down, then all the members of the same group will be picked automatically.
2. Attributes of dimension members can be defined individually, so that dimension members can have various dimension form.
3. Dragging any one of the members will move the whole dimension group together.

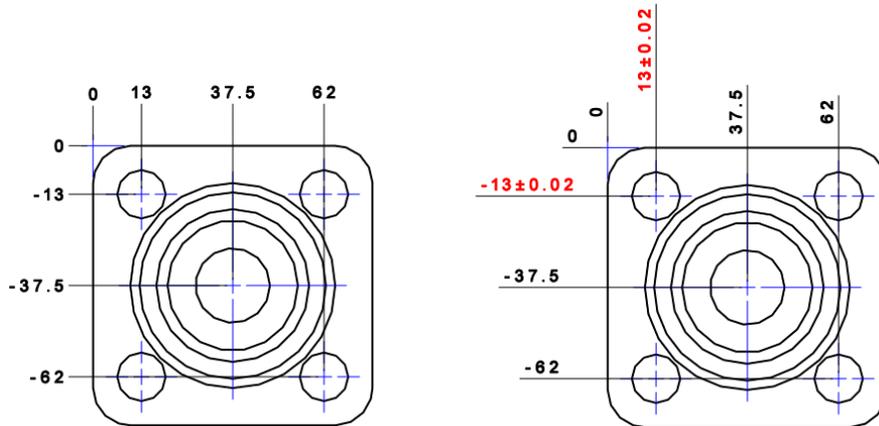


3.6.2.9 Changes in “Ordinate Dimension”

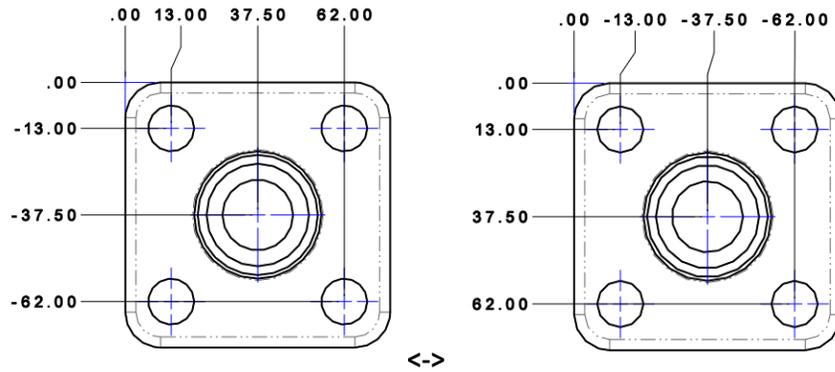
Since “Baseline dimension” and “Continuous Dimension” have changed the relation between members and group, “Ordinate Dimension” also inherit such change. Besides this update, more options have been added into “Ordinate Dimension”.

1. **Dimension member can be more flexible to set up its own attribute**

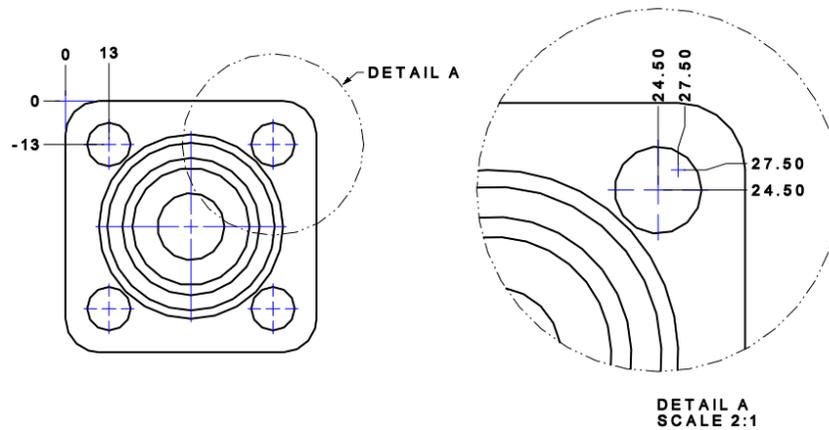
Same behavior is supported as “Baseline dimension” and “Continuous Dimension”.



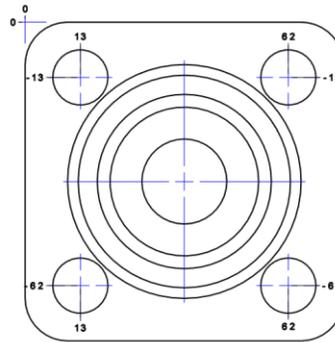
2. New "Direction" field to switch positive and negative direction of X/Y axis



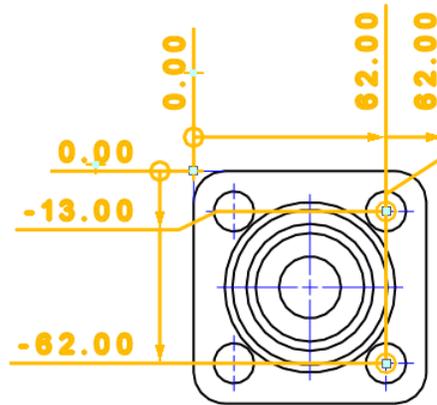
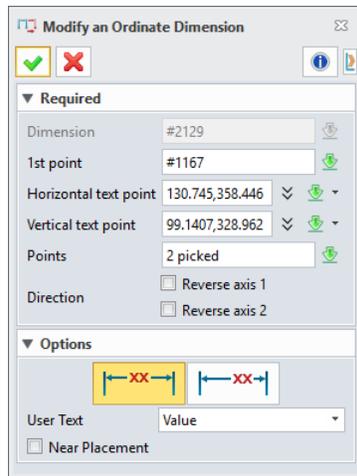
3. New "Share origin" field to re-use the origin of the existing ordinate dimension, such as ordinate dimension on detail view re-uses the one in regular view



4. New "Near placement" field to place the ordinate value beside the dimensioned entities



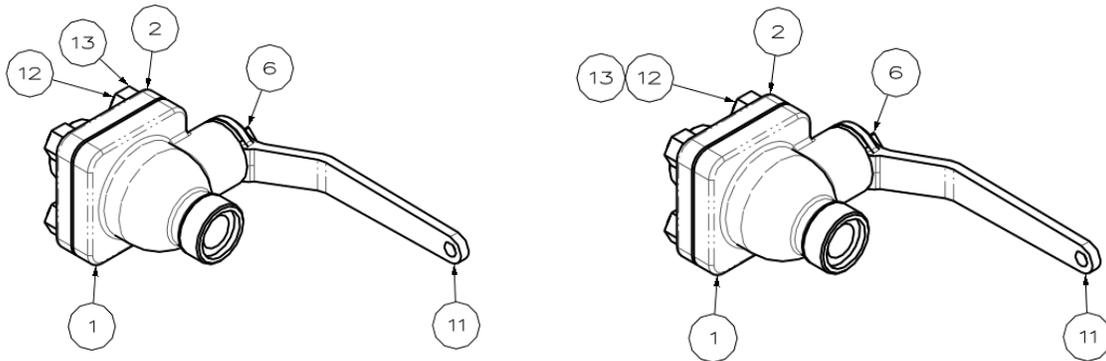
5. “Modify dimension point” command can edit ordinate dimension, such as the origin, dimension text placement, or adding or remove dimension points and so on.



6. “Add to Group” and “Remove from Group” can work on ordinate dimension.

3.6.2.10 ★New “Stack Balloons” and “Sort Stack Balloons”

“Stack Balloons” can gather balloons that are too close to each other into one balloon to avoid overlapping.



1. Users can pick balloon dimensions to stack, or directly select projection entities of components, or component entries from sheet manager.
2. 4 ways to place the stacked balloons are available from the right-click menu of the stacked balloon

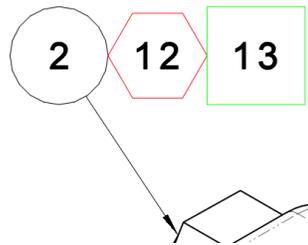
Stack right	Stack left	Stack up	Stack down

3. Various styles are provided

Stack right-underline	Stack up-underline	Stack right-none	Stack right-hexagon

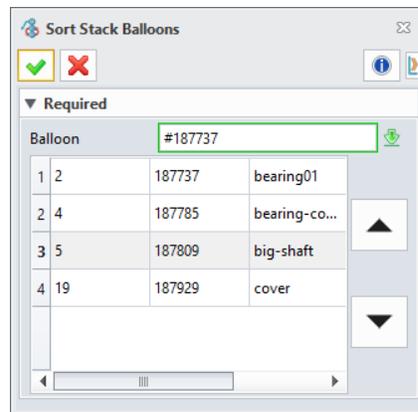
4. "Auto sort" can re-arrange the stacked balloons ascendingly during stacking. The new "Sort Stack Balloon" command can re-order existing stacked balloons.

- 5. “Add to Group” and “Remove from Group” can work on stacked balloons, or users can delete the picked balloon directly.
- 6. “Explode Group” can dissolve the stacked balloons into individual ones.
- 7. Each balloon from the stacked balloons can have its own attributes



- 8. Each balloon member from the stacked balloons can be selected individually. Multiple picking with Ctrl and Shift keys is supported.

“Sort Stack Balloons” offers a form to allow users to re-order stacked balloons manually.



➔ Where it is

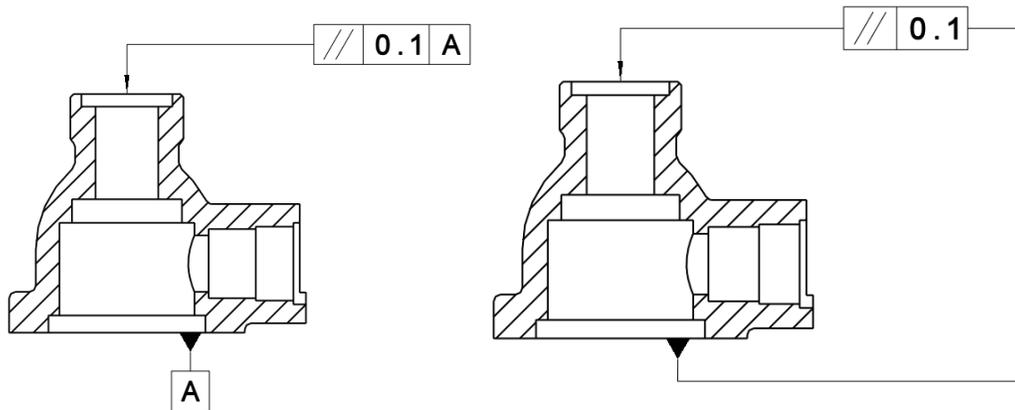
Drafting context > Dimension ribbon > Annotation > Stack Balloon

Drafting context > Dimension ribbon > Edit Dimension > Sort Stack Balloon

3.6.2.11 New "Auxiliary datum" in "Feature Control"

The new "Auxiliary location" field in "Feature Control" can define an extra datum feature to indicate which entity will work as datum.

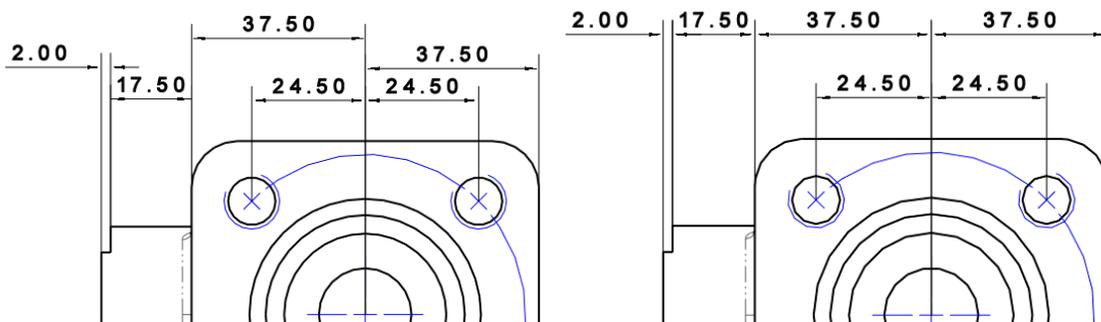
Multiple points can be defined in "Auxiliary location", and first point should be on specific entity which should work as datum, and subsequent points are to define how the leader should be broke and turned. "Modify dimension point" can modify those points, or users can use the "Insert branch" and "Delete branch" from the right-click menu to modify.



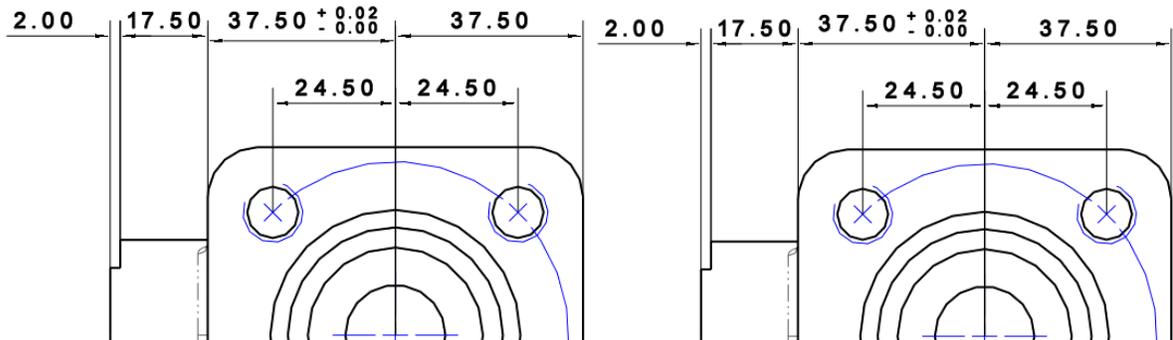
3.6.2.12 Multiple Dimension Supported in "Modify Text Alignment"

Basing on feedback, "Modify Text Alignment" has done following changes:

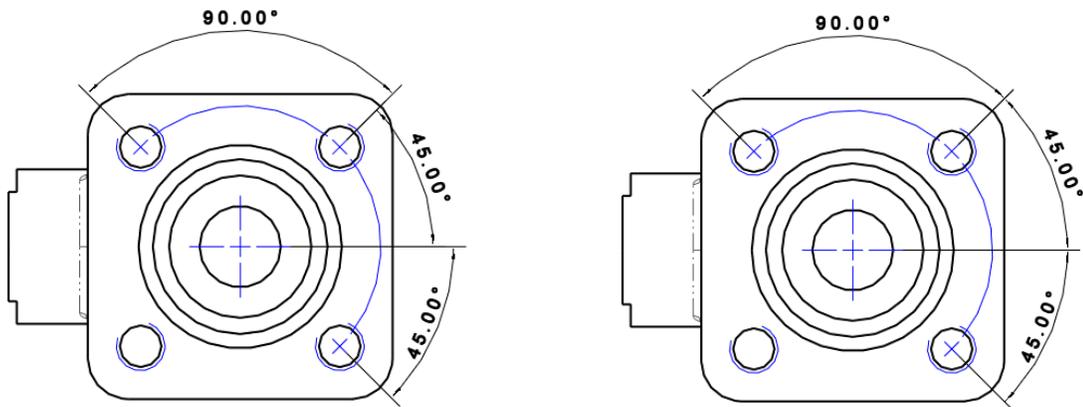
1. Multiple dimension can be modified at one time



2. New “Linear dimension align to line” option to change the alignment base from text to dimension line



3. Angle dimension now can be handled



3.6.2.13 Enhancements in BOM

Since 3D BOM comes out, 2D BOM has done updates as following:

1. New “Inherit 3D BOM” in Template field

If 3D model has a defined 3D BOM, users can use this option to copy the setup into 2D BOM to make sure these 2 BOM have the same outcome.

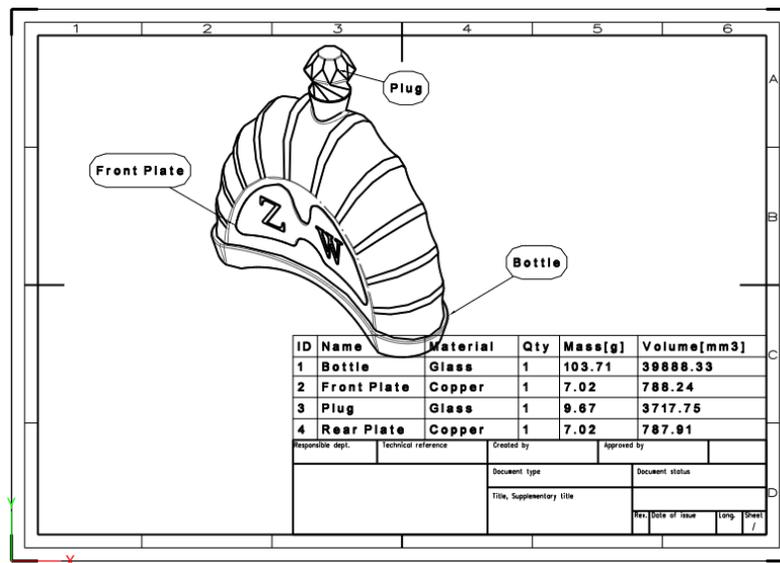
2. New “Merged by” field to combine rows with defined attributes

If the values of the defined attributes are the same, rows with that value will be combined into one row, so that the table can be simplified.

3. New “Show shapes” option to include shapes in BOM

Since the shapes can be projected individually as components, BOM table needs to include shapes to make them as components with this new option.

If “Balloon” and “Auto Balloon” is expected to work on shapes, this option must enable.



3.6.2.14 ★Improvements in Style Manager

Style manager has included more dimension objects and supported batch edit to improve work efficiency with following details:

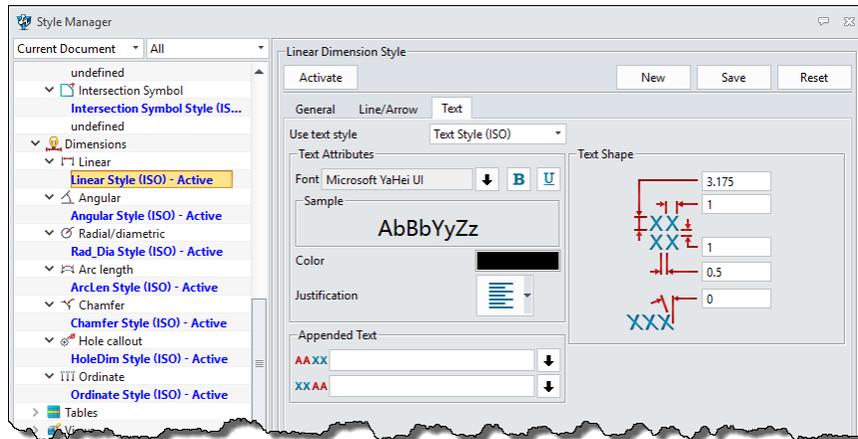
1. More dimension objects can be defined inside style manager

Following dimension objects are newly added into style manager.

Text, Hatch, Weld symbol, Roughness and Intersection symbol.

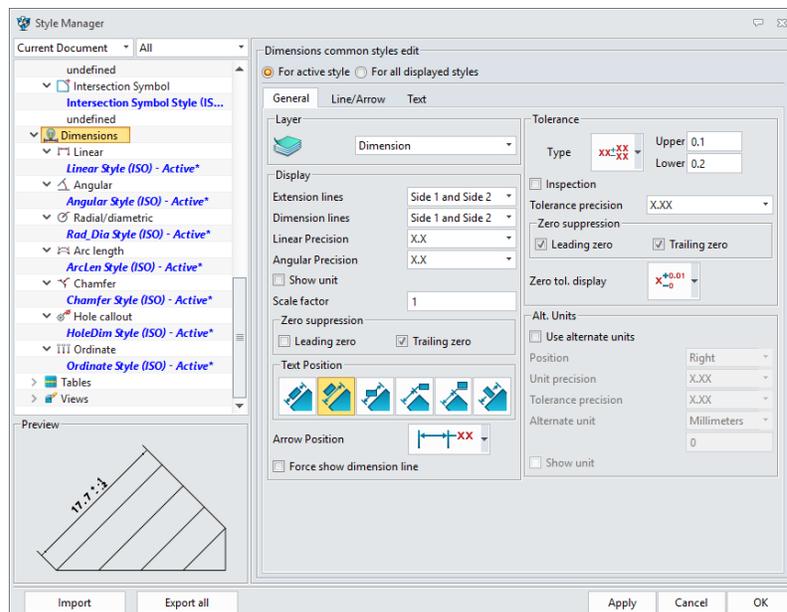
2. Text in each dimension object can use the defined text style

After text can set up different styles, all dimension objects now can directly use those style, so that users can make sure all texts in drafting use the same format and work easier and more efficient.



3. Batch edit on root node for each dimension object

On every root node of very dimension object, the common options of those dimension objects are provided so that users can modify all the settings of all styles at one operation.

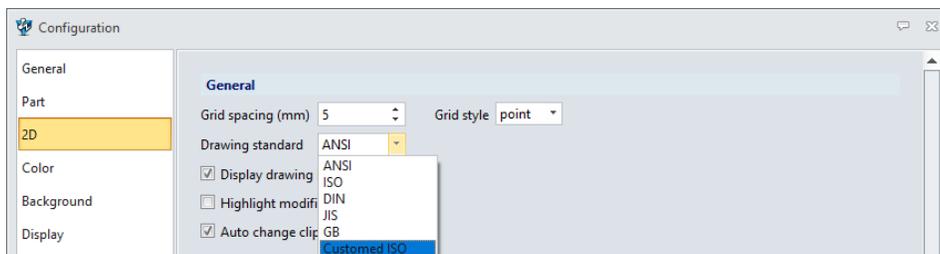


4. Custom style behavior changed from auto to manual

When changing some dimension settings on the dimension attribute form, the Style field will not change into Custom as before remaining.

5. Drawing standard can use custom standard as default

“Configuration > 2D > General > Drawing standard” option now can read the custom standard saved in user folder, and set as default.



4 CAM

4.1 Summary of CAM New Features

ZW3D CAM contains 2~5axis machining modules with flexible and rich milling strategies. It's an easy to learn and easy to use NC CAM software which is widely used in tool making, automotive industry, and mold industry. This chapter describes the most significant CAM improvements of ZW3D 2019 including the following new features and enhancements:

- 1) **Volumill 2X and 3X Integration** – An ultra-high-performance technology applied to roughing of 2 axis and 3 axis operation.
- 2) **Optimization of the “Link” Function in 2X Operation**
- 3) **Subdivision of Tool Path Type in 2X and QM Operation**
- 4) **Enhance the Custom CL Command Function**
- 5) **Enhance the Tool Path Verification Function**
- 6) **Enhance the Corner Control Function in 2X Operation**
- 7) **Support 3D Arc Output in 2X and QM operation**
- 8) **Linear Array of Tool Path can Output NC Code with Sub-program format**
- 9) **Enhance the Flat Region Recognition**
- 10) **Enhance the Big Stepper in 3X Roughing**
- 11) **5X Swarf Cut Enhancement to Support 5X Laser Machine**
- 12) **Misc.** – some enhancements to users' work more efficient and more stable, such as:
 - A. **New Quantity Function and Automatic Sort Function in Hole Feature**
 - B. **New Tool Path Status Icon**
 - C. **Optimize the Definition Process and Redefine the method of Engage and Retract**
 - D. **Enhance the Safe Ramp Option**
 - E. **Offset Normal of Surface Feature Supports in QM**

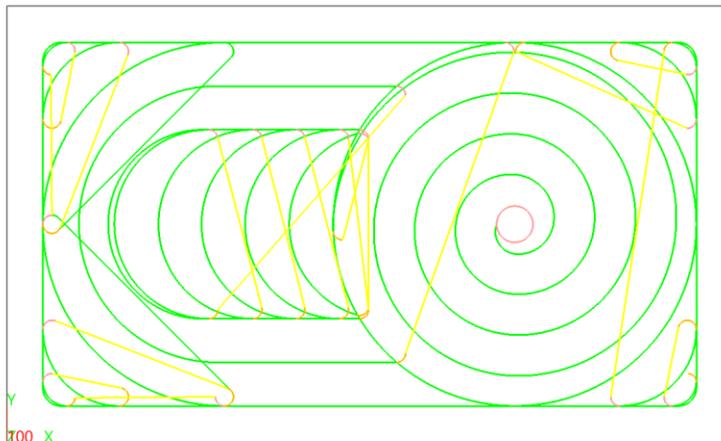
- F. Regulate the Tool Manager Parameters Tap**
- G. Enhance the 2X Profile Operation and Chamfer Feature**
- H. 2X Operations do not Support the Profile Features of “Contain” Type**

4.2 Volumill 2X and 3X Integration

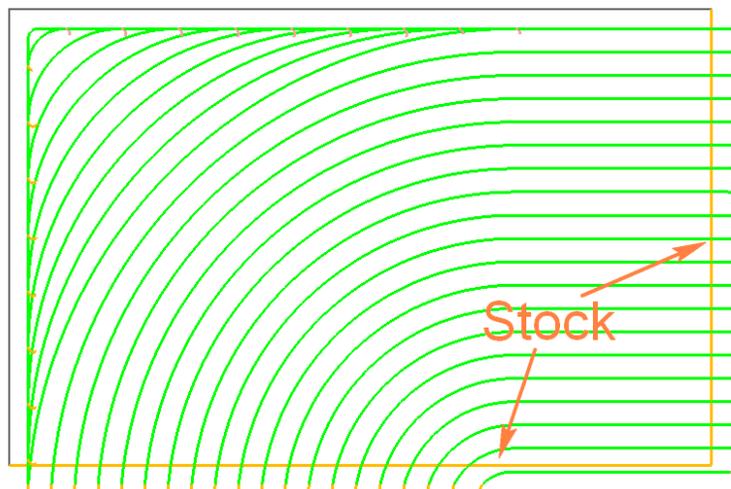
Volumill is an ultra-high-performance technology. A constant material removal rate enables the machine and the tools to operate under ideal milling conditions, reducing cycle time, thereby reducing machine workload and prolonging tool life.

4.2.1 Volumill 2X Operation

By defining the profile features, material removal rate and other parameters, to generate tool path.

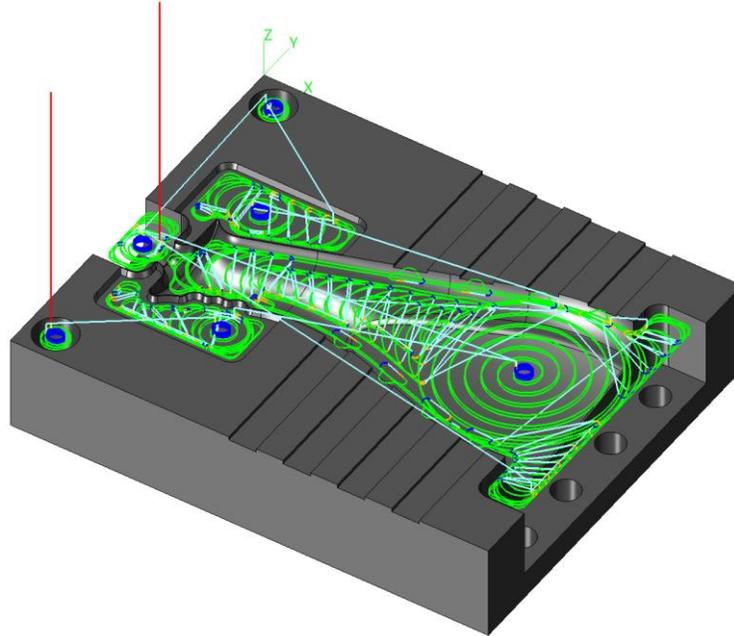


When set the profile as “Stock” Type, the external engage can be implemented.



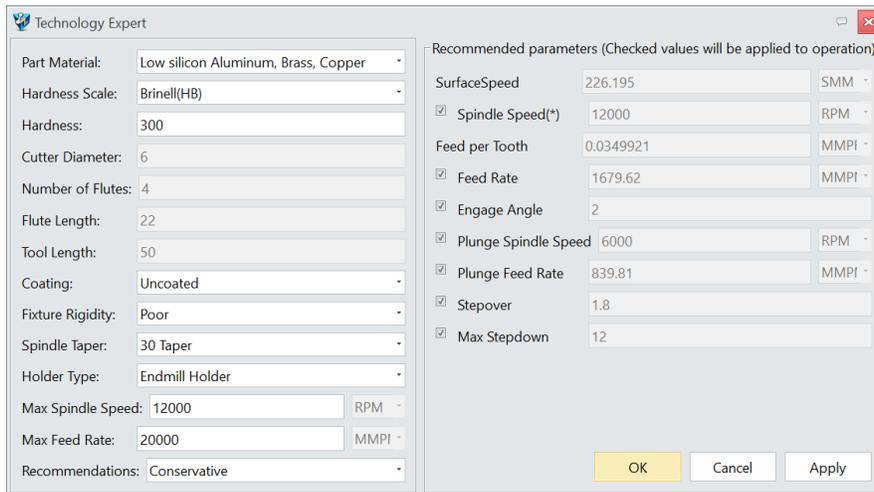
4.2.2 Volumill 3X Operation

By defining the profile features, material removal rate and other parameters, to generate tool path.



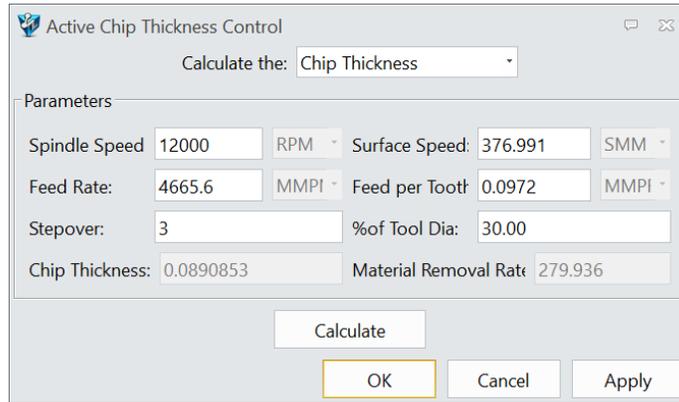
Technology Expert

According to the actual situation, the user selects different materials, sets the maximum speed and feed rate of the machine, as well as the type of tool and fixture. The expert system will automatically calculate the speed, feed rate and step size of the process according to the user's settings. Users can choose whether to use these recommended values.



Active Chip Thickness Control:

The cutting rate control function allows the user to modify the cutting parameters and control the material removal rate, thereby further improving the cutting efficiency and reducing the processing time.



4.3 Optimization of the “Link” Function in 2X Operation

Optimization of link between the 2X tool path, user can use different link type in different positions of tool path in operation. Optimize the tool path and improve the processing efficiency. The parameters are shown as follows:

Inside Region Type — Control the link tool path in same region.

Between Layer — Control the link tool path between different layers in same region.

Between Region — Control the link tool path between different regions.

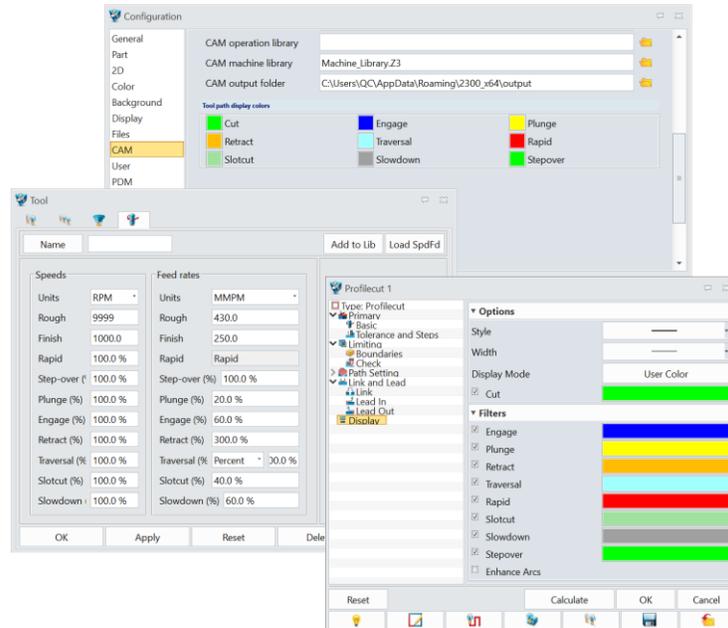
Clearance Z — Define the Clear Z of the operation.

Safe Height — Define the safe height for link tool path.

▼ Link	
Inside Region Type	Clearance Z
Between Layer	Clearance Z
Between Region	Clearance Z
Clearance Z	100
Safe Height	2.5

4.4 Subdivision of Tool Path Type in 2X and QM Operation

Divided the tool path type of 2X and QM operations into 9 types, respectively: Engage, Plunge, Retract, Traversal, Rapid, Cut, Slotcut, Slowdown and Stepmover. Each tool path can be set with different colors according to the user's needs to facilitate user identification.



4.5 Enhance the Custom CL Command Function

Optimized the original custom CL command function, redesigned the user interface, and supported editing the head and end commands for multiple operations at the same time and changing their state.

Custom CL Command										
	NC name	Op name	No.	Head CMD	Parameters	On/Off	No.	End CMD	Parameters	On/Off
1		Rough Offset 2d 1	1	MAX SPEED	15000	ON	1	COOLANT OFF ACTIVE		OFF
2			2	CLAMP AXIS	Xaxis	ON				
3		Rough Offset 2d 1	1	AXIS	ZAXIS	OFF	1	LOCK PLANE	ZPLAN	OFF
4		Rough Offset 2d 2	2	AXIS	YAXIS	ON	2	AXIS	YAXIS	ON
5		Lace 1	1	LOCK PLANE	XYPLAN	ON	1	LOCK PLANE	XYPLAN	OFF
6		Zigzagcut 1	1	SEQ NUMBER	3	OFF				
7		Spiralcut 1	1	Tool PRESELECT	6	ON				

4.6 Enhance the Tool Path Verification Function

Optimized the original toolpath verify function, redesigned the user interface, added wireframe display function, displayed tool holder and various display toolpath modes.

Tool path display mode:

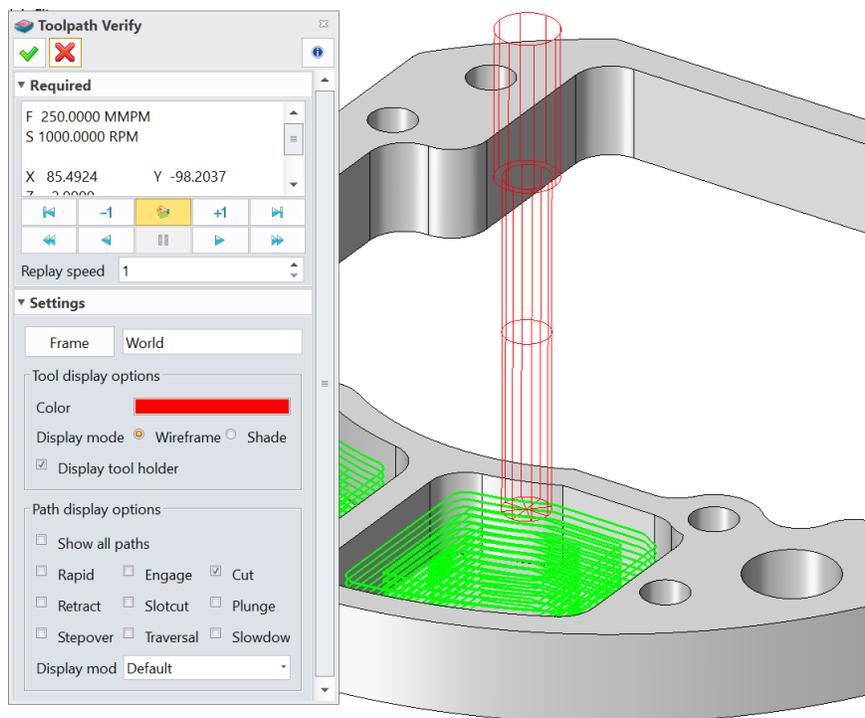
Default — Default tool path display mode

Current motion — Display current motion tool path

Absolute Z-height — Display the tool path of Z-direction height of the absolute coordinate

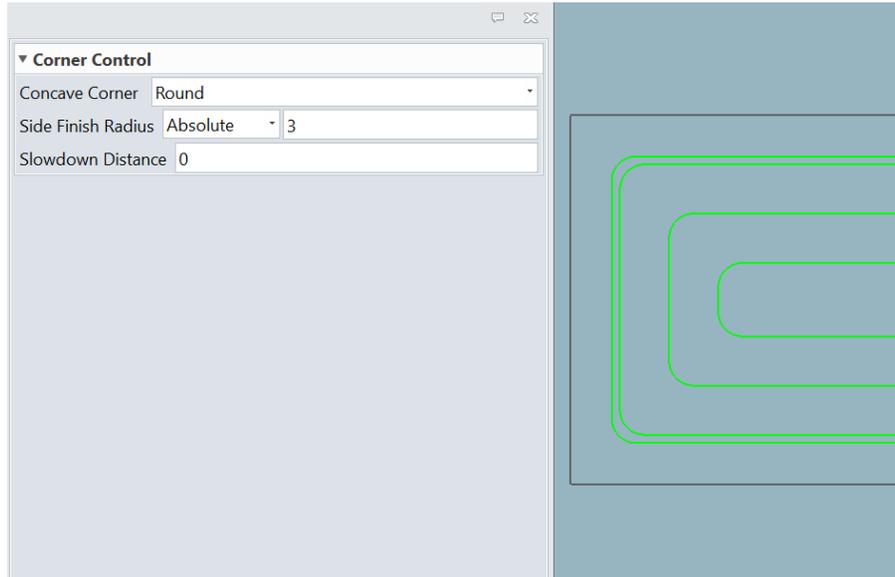
Relative Z-height — Display the tool path in the Z direction of the current coordinate relative to the tool position

Prev/Next location — Display the previous and next motion tool path



4.7 Enhance the Corner Control Function in 2X Operation

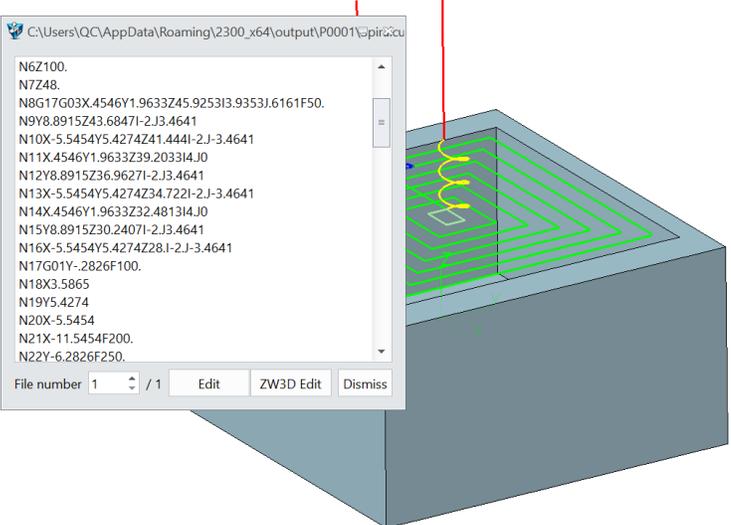
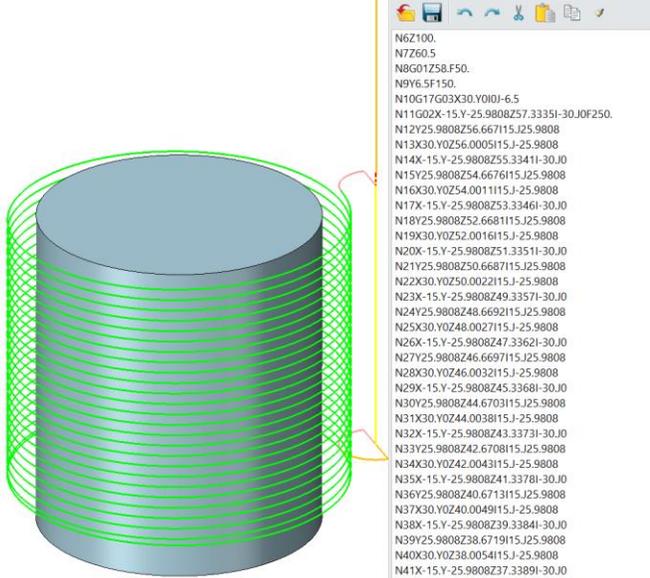
For the corner control function, a new parameter is added, which allows the user to set the corner radius of the outermost and side finishing to optimize the tool path. It can reduce tool wear during roughing and improve the quality of corners in finishing.

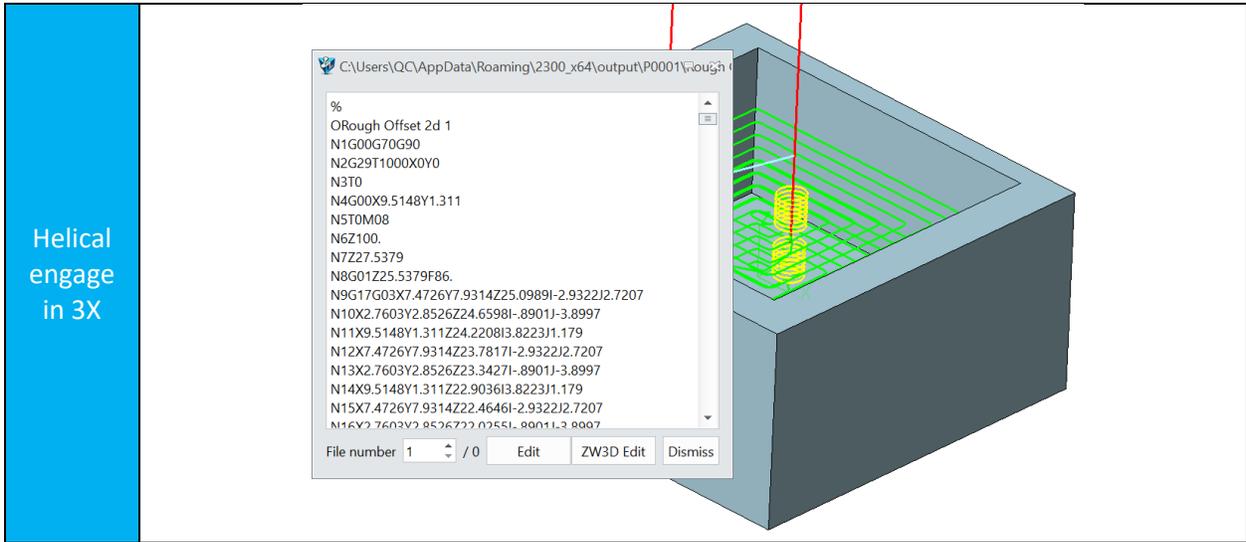


4.8 Support 3D Arc Output in 2X and QM operation

This version supports the 3D arc in 2X and QM operation to output with G02 or G03 command, which makes the tool smoother during machining and greatly reduces the number of NC codes. The specific scenarios are shown in the following table:

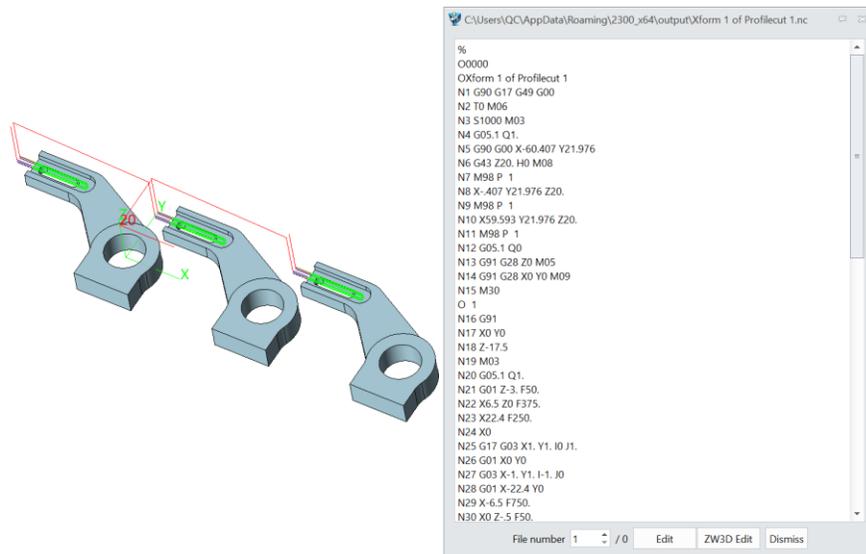
Support 3D arc output in 2X	
Round corners of Ramp	<div style="display: flex; align-items: center;"> <div style="flex: 1;"> <pre> N6Z100. N7Z30.5 N8G01Z28.F50. N9X-50.5F150. N10G17G03X-57.Y-58.I-6.5J0 N11G02X-62.Y-53.Z27.9168I0J5.F250. N12G01Y53.Z26.7926 N13G02X-57.Y58.Z26.7094I5J0 N14G01X57.Z25.5003 N15G02X62.Y53.Z25.4171I0J-5. N16G01Y-53.Z24.2928 N17G02X57.Y-58.Z24.2097I-5J0 N18G01X-57.Z23.0005 N19G02X-62.Y-53.Z22.9174I0J5. N20G01Y53.Z21.7931 N21G02X-57.Y58.Z21.7099I5J0 N22G01X57.Z20.5008 N23G02X62.Y53.Z20.4176I0J-5. </pre> <p>File number 1 / 0 Edit ZW3D Edit Dismiss</p> </div> <div style="flex: 1; text-align: center;"> </div> </div>

<p>Helical Plunge in 2X</p>	 <pre> C:\Users\QC\AppData\Roaming\2300_x64\output\P0001\c:\pir\2cu N6Z100. N7Z48. N8G17G03X.4546Y1.9633Z45.9253I3.9353J.6161F50. N9Y8.8915Z43.6847I-2J3.4641 N10X-5.5454Y5.4274Z41.4441-2J-3.4641 N11X.4546Y1.9633Z39.2033I4.J0 N12Y8.8915Z36.9627I-2J3.4641 N13X-5.5454Y5.4274Z34.722I-2J-3.4641 N14X.4546Y1.9633Z32.4813I4.J0 N15Y8.8915Z30.2407I-2J3.4641 N16X-5.5454Y5.4274Z28.I-2J-3.4641 N17G01Y-.2826F100. N18X3.5865 N19Y5.4274 N20X-5.5454 N21X-11.5454F200. N22Y-6.2826F250. File number 1 / 1 Edit ZW3D Edit Dismiss </pre>
<p>Ramp of the cylinder</p>	 <pre> N6Z100. N7Z60.5 N8G01Z58.F50. N9Y6.5F150. N10G17G03X30.Y0I0I-6.5 N11G02X-15.Y-25.9808Z57.3335I-30.J0F250. N12Y25.9808Z56.667I15.J25.9808 N13X30.Y0Z56.0005I15.J-25.9808 N14X-15.Y-25.9808Z55.334I1-30.J0 N15Y25.9808Z54.667I15.J25.9808 N16X30.Y0Z54.0011I15.J-25.9808 N17X-15.Y-25.9808Z53.3346I-30.J0 N18Y25.9808Z52.668I15.J25.9808 N19X30.Y0Z52.0016I15.J-25.9808 N20X-15.Y-25.9808Z51.3351I-30.J0 N21Y25.9808Z50.668I15.J25.9808 N22X30.Y0Z50.0022I15.J-25.9808 N23X-15.Y-25.9808Z49.3357I-30.J0 N24Y25.9808Z48.669I15.J25.9808 N25X30.Y0Z48.0027I15.J-25.9808 N26X-15.Y-25.9808Z47.3362I-30.J0 N27Y25.9808Z46.669I15.J25.9808 N28X30.Y0Z46.0032I15.J-25.9808 N29X-15.Y-25.9808Z45.3368I-30.J0 N30Y25.9808Z44.6703I15.J25.9808 N31X30.Y0Z44.0038I15.J-25.9808 N32X-15.Y-25.9808Z43.3373I-30.J0 N33Y25.9808Z42.6708I15.J25.9808 N34X30.Y0Z42.0043I15.J-25.9808 N35X-15.Y-25.9808Z41.3378I-30.J0 N36Y25.9808Z40.6713I15.J25.9808 N37X30.Y0Z40.0049I15.J-25.9808 N38X-15.Y-25.9808Z39.3384I-30.J0 N39Y25.9808Z38.6719I15.J25.9808 N40X30.Y0Z38.0054I15.J-25.9808 N41X-15.Y-25.9808Z37.3389I-30.J0 N42Y25.9808Z36.6724I15.J25.9808 </pre>
<p>Support 3D arc output in QM</p>	



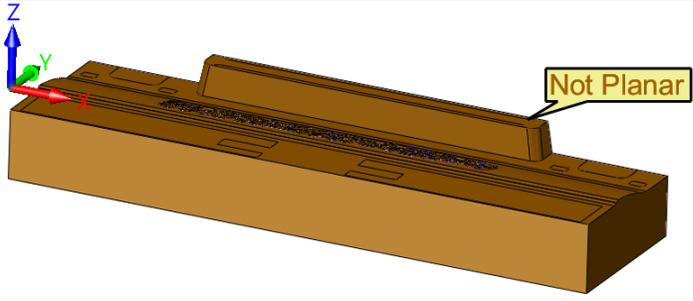
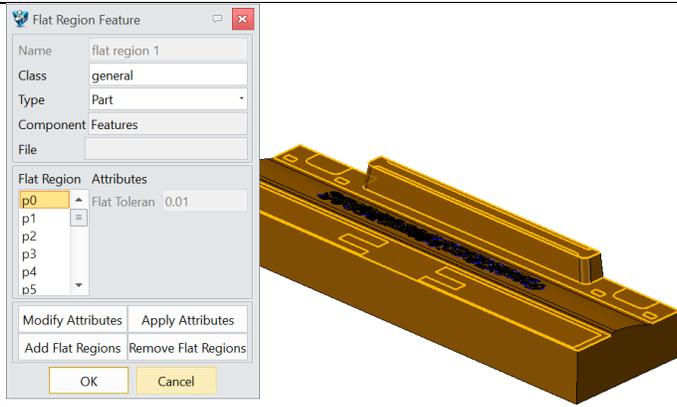
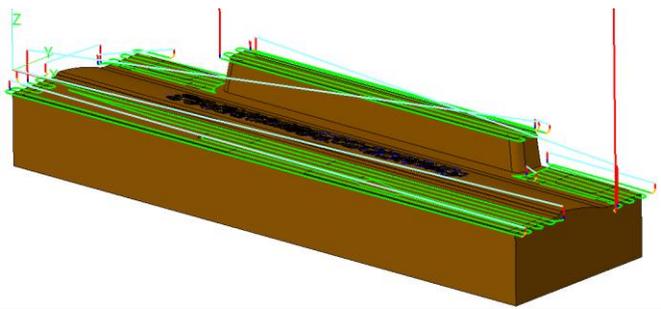
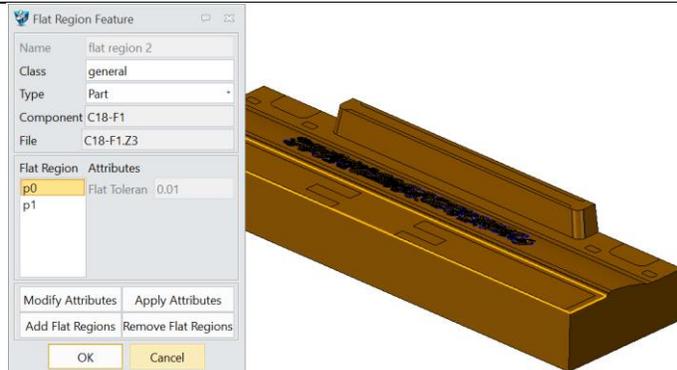
4.9 Linear Array of Tool Path can Output NC Code with Sub-program format

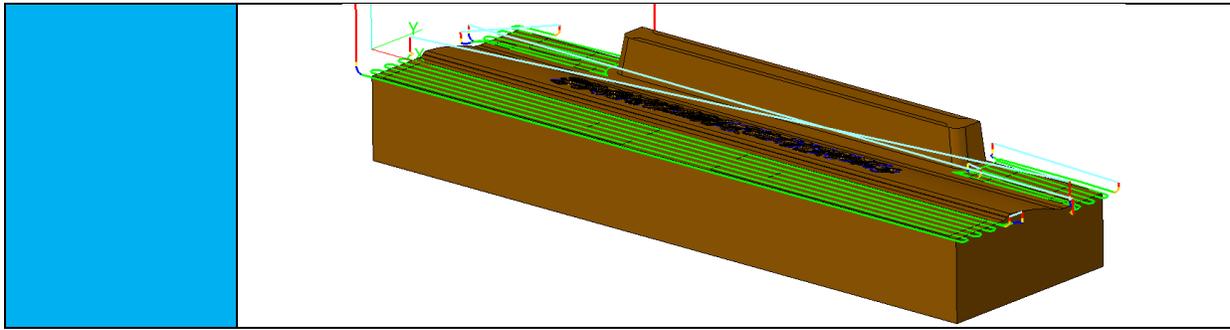
The tool path obtained by the "point-to-point" and "rectangular array" transformation uses the original tool path as the main program, and is converted and outputted in the form of a sub-program, thereby achieving the purpose of streamlining the NC code amount.



4.10 Enhance the Flat Region Recognition

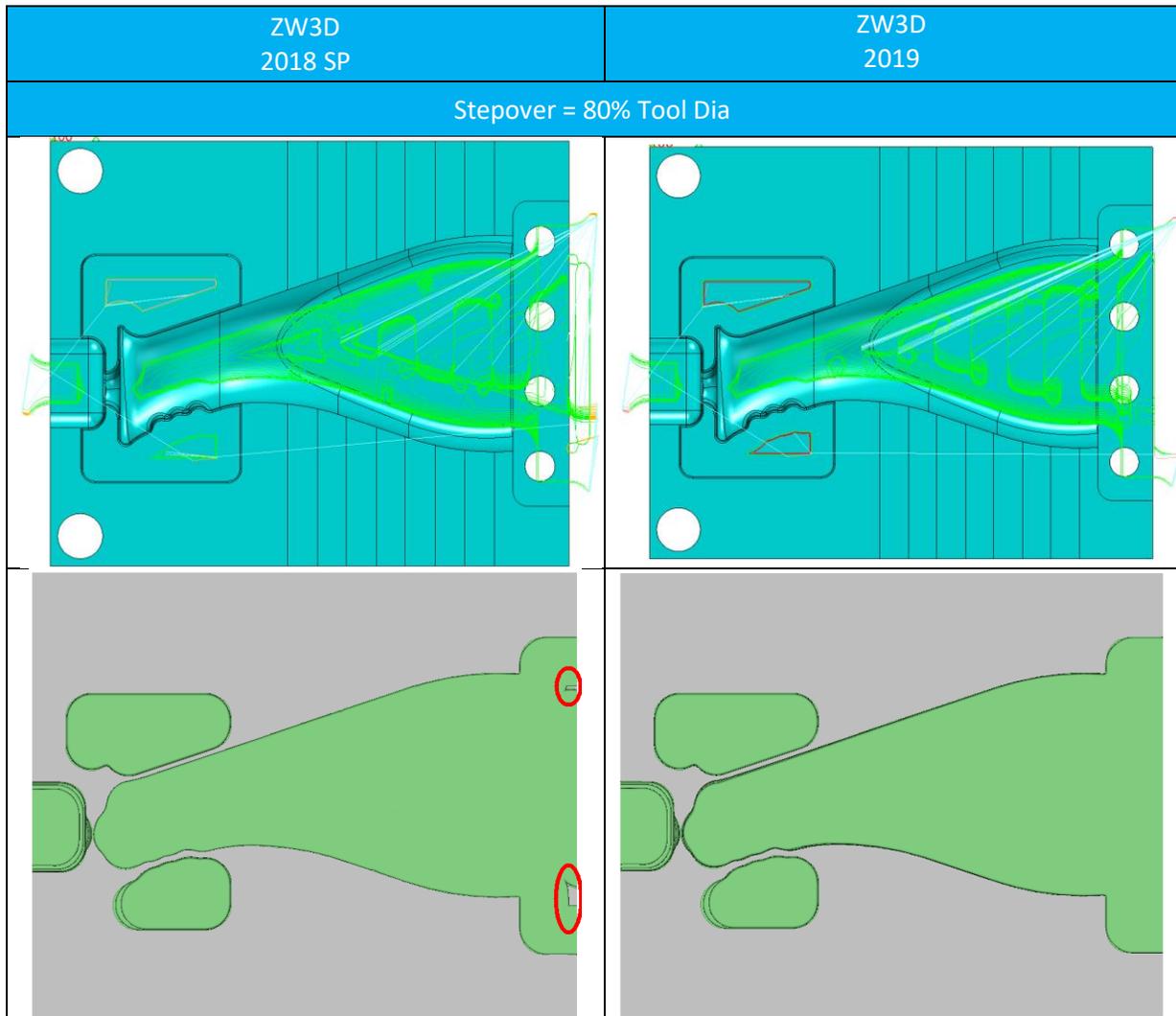
In this release, ZW3D can more accurately recognize the flat region within the flat tolerance, that makes more reliable flat finishing tool path generation.

<p>Flat Tol = 0.01</p>	
<p>ZW3D 2018 SP</p>	
<p>ZW3D 2018 SP</p>	
<p>ZW3D 2019</p>	



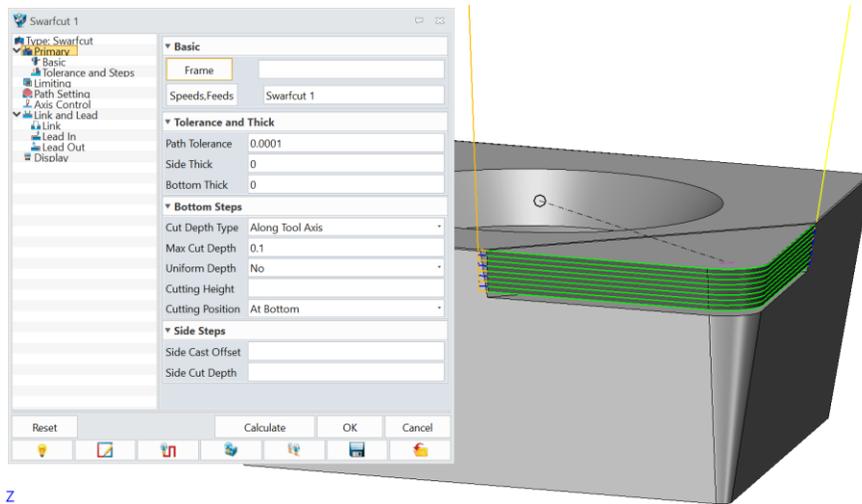
4.11 Enhance the Big Stepper in 3X Roughing

This version enhances the big stepover of 3X roughing, more reliable big stepover tool path will be generated.

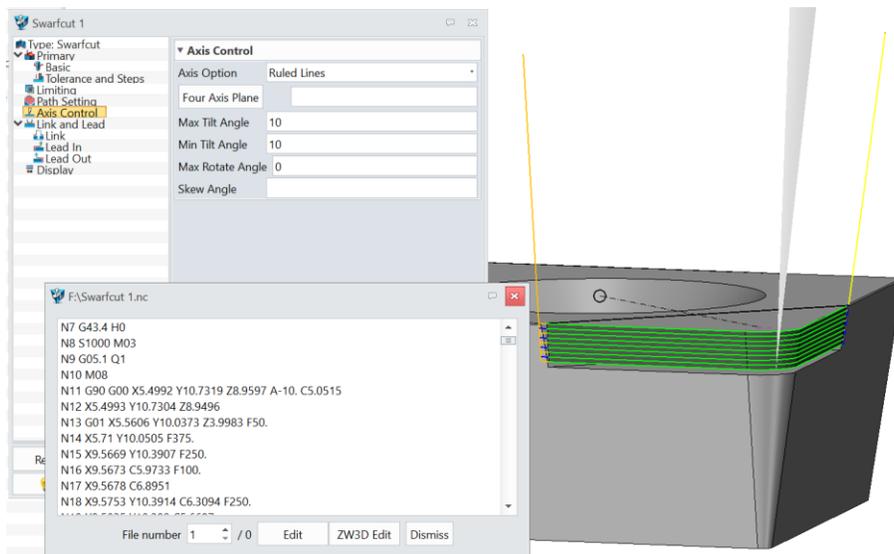


4.12 5X Swarf Cut Enhancement to Support 5X Laser Machine

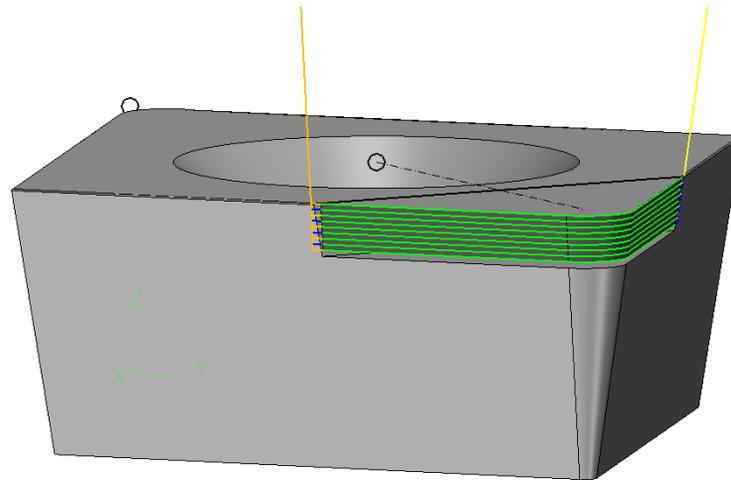
In ZW3D 2019, 5X Swarf cut operation supports more accurate tool path generation.



Add Min Tilt Angle option to control the tool axis min tilt angle, if set the Max Tilt angle same as the Min Tilt Angle, it will output the constant tool axis tilt angle.



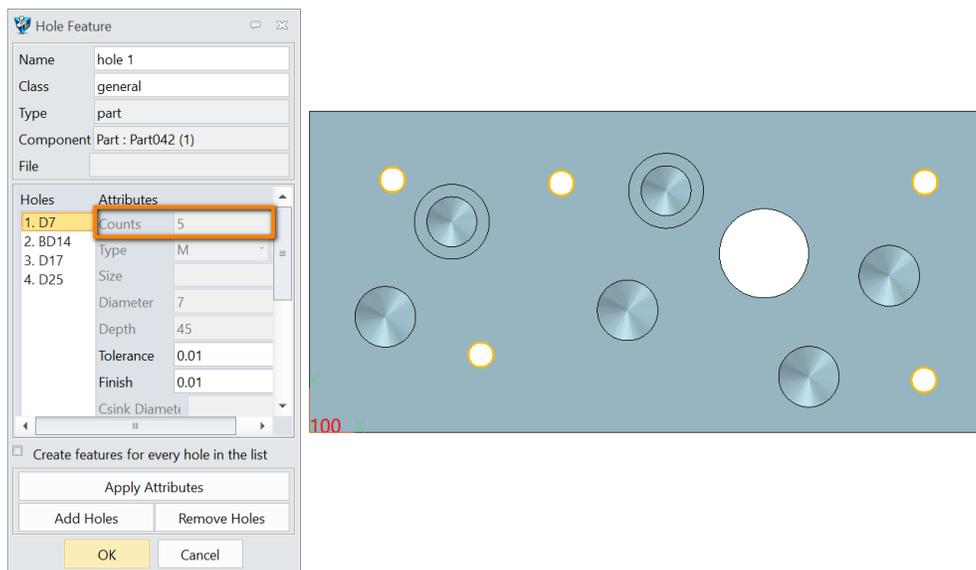
For the Laser machining, the top surface needs to generate a layer of tool path.



4.13 Misc. – some enhancements to users’ work more efficient and more stable

4.13.1 New Quantity Function and Automatic Sort Function in Hole Feature

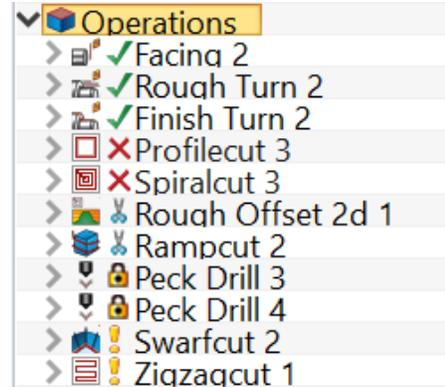
In this release, a new number of entries have been added to the hole feature to help the user count the number of holes of different sizes. Hole features are also automatically sorted by hole diameter from small to large. See the picture below:



4.13.2 New Tool Path Status Icon

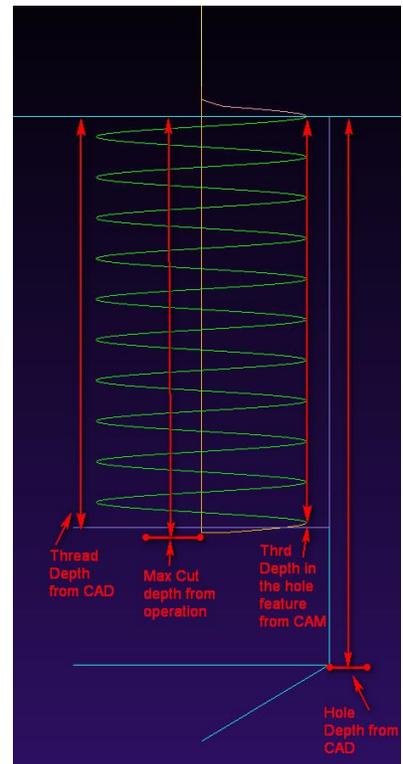
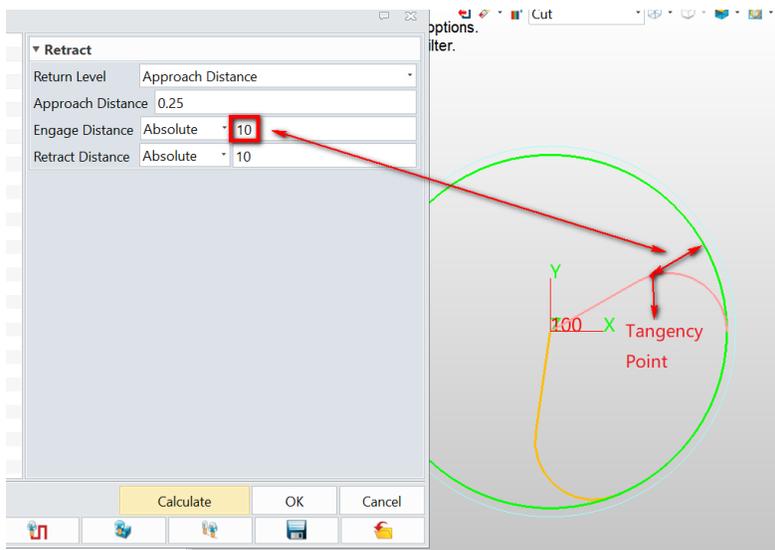
In this version, a toolpath status icon has been added to the Navigator to allow users to view the status of the toolpath.

-  — Tool path generated
-  — No tool path generated
-  — Tool path has been edited
-  — Tool path has been locked
-  — The operation parameters have been modified but the tool path has not been regenerated



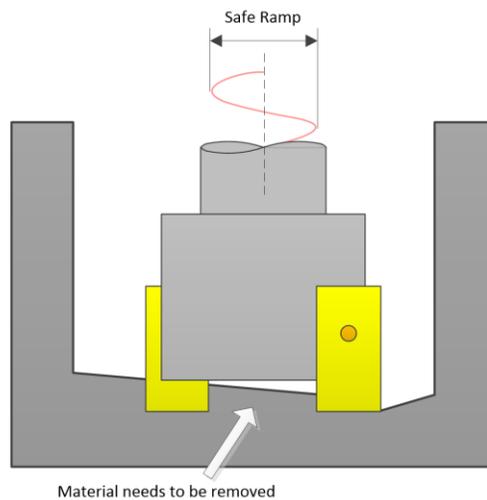
4.13.3 Optimize Definition Process and Redefine the method of Engage and Retract

The depth of the thread is defined by the thread depth in the hole feature. The Max Cut Depth function in the operation defines the thread depth and the depth of the retraction to ensure that the resulting tool path does not hit the bottom of the hole. At the same time, the definition of the Engage Arc and Retract Arc are changed to Engage Distance and Retract Distance, ensure that the threads of tool can be completely withdrawn from the hole thread without damaging the already machined thread.



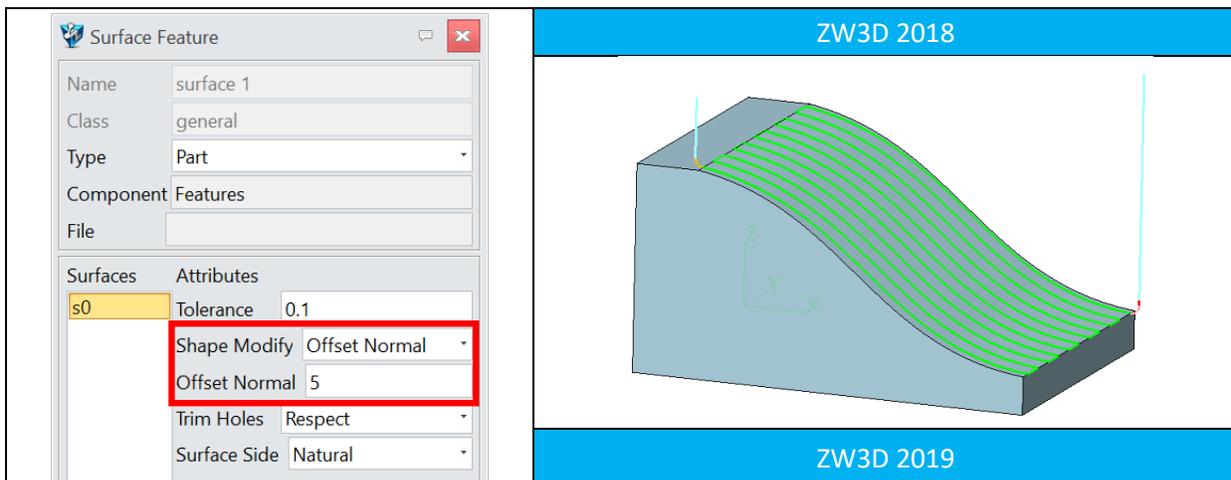
4.13.4 Enhance the Safe Ramp Option

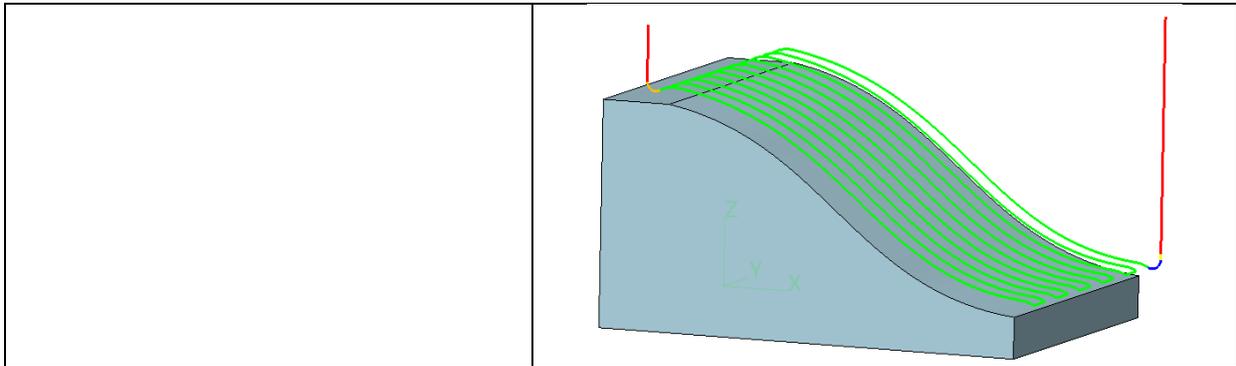
Controlling the minimum distance that the tool must be moved into material during ramping and helical engage. Safe Ramp Length is especially useful for inserting cutters where there must be sufficient overlap between the leading and rear blades to prevent uncut material from colliding with the non-cutting bottom portion of the cutter.



4.13.5 Offset Normal of Surface Feature Supports in QM

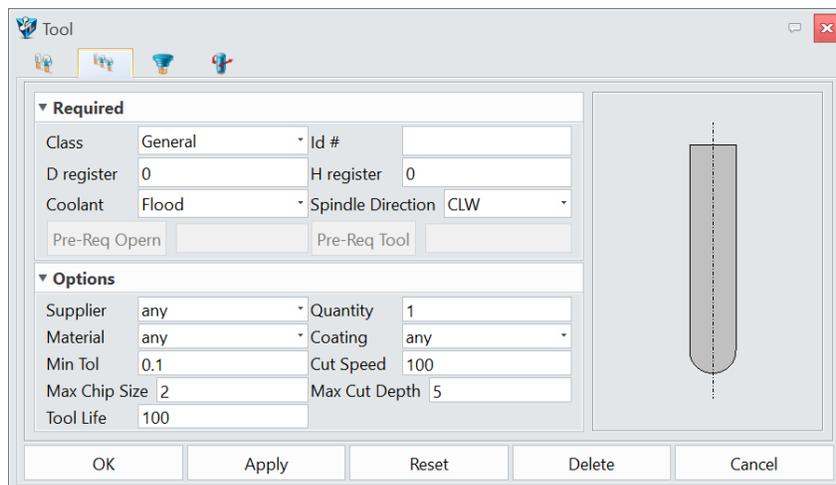
In 2019 version, this option supports in QM, user can through using it to achieve special machining purpose with QM operation, like protecting a particular edge or corner.





4.13.6 Regulate the Tool Manager Parameters Tap

Regulate the parameters places in the tool parameters tap, distinguish between required and options parameters, convenient for user to adjust tool parameters and submit efficiency.



4.13.7 Enhance the 2X Profile Operation and Chamfer Feature

In this version, some modifications make the 2X Profile operation more concise. In Cut Direction function, the “CW” and “CCW” options have been removed. In XY Tool Location function, “On Part, Tan Island” and “Past Part, Tan Island” options have been removed. The Profile Side function have been renamed to Machining Side, and changed its “Left”, “Right” options to “Left, Inside” and “Right, Outside”. For Chamfer feature, the options of Chamfer Side have been changed to “Left, Inside” and “Right, Outside” from “Left” and “Right”, to determine which side of tool to machine on.

4.13.8 2X Operations do not Support the Profile Features of “Contain” Type

Previous versions of the 2X operations have always supported the "Parts" and "Contain" of the two types of profile features to generate the toolpath, and the new version of the 2X operations no longer support the use of the "Contain" type of profile features to generate the toolpath, and when you create a new profile feature, the default type is “Part”.