

ZW3D WHAT'S NEW

V 2021



ZWSOFT CO., LTD.(Guangzhou)





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ZW3D[™] V2021 What's New

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Highlights of ZW3D 2021

Basics:	New "Command Search"
	New "Map Key"
Translator:	Adjustment to "Quick Import"
	Convert Attributes Support
CAD:	New "Equal Curvature"
	New "Cosmetic Sketch"
	Modeling Dimensional Accuracy Update
	Complete CSYS Support
	New Cross-section Type "G2 Blend" in Fillet Command
	"Blend Face" Command New Option "G2 Blend"
	"Fillet Open Faces" command New sub menu "Shape of Fillet"
	New "Zero Radius Flange"
	New "Clearance Check"
	New "Batch Attribute Edit"
Drafting:	New "Point Table" and "Dimension Table"
CAM:	Enhancements of Full Machine Simulation
	The Rough Turning Operation Supports Arbitrary Contour as Stock
	New Arc Fitting Functionality in Surface Engraving Operation and 5X Operations
	New Excel Format of Operation List
	Support the G68.2 Euroctionality
	<u>support the doord runctionancy</u>

Note: Important enhancements in this article are marked with *****



1 Basic

1.1 UI & Workflow

1.1.1 **★**New "Command Search"

Use **"Command Search"** to search by character matching and each search history will be recorded. The search records are ordering by time with the latest ones ahead. The search object includes the command name and the command description. As for search results, they will display in multiple pages with each page of 30 items and provide the functions of the page running and page number. Support dynamic displaying the search result command in current UI. User can directly execute the command search, but only limit to the command in the current environment.





1.1.2 **★**New "Map Key"

"Map Key" supports recording the workflow of a single command or multiple command groups to meet quick completion of some batch design operation. User can input parameters for template command and can add, delete, modify, and save **"Map Key"**.

Definition of "Map Key"

- (1) **"Map Key"** supports all combinations of letters and numbers with 32 maximum characters.
- (2) The widget layer will disable user's input once irregular character input happens (such as control character and special character). All irregular input will be disposed in the method of neglect.

Add, delete, modify, and save "Map Key"

1. Support addition of "Map Key"

(1) Required inputs include "**Map Key**" and Name (**64** maximum characters) while Description is optional input. To select record action is supported.

(2) When **"Map key"** is empty, click OK and the interface will be saved, in case of user losing the recording and input text when the user forgets to input **"Map Key"**.

2. Support deletion of single or multiple "Map Key"

(1) The deletion only targets at the contents of memory. Once user did not click Save, the software would restore the deleted content while it was restarted.

3. Support modification of single "Map Key"

(1) When **"Map Key"** works as unique key, modification is not supported, nor the record option modification.

(2) User can modify name and description of **"Map Key"** as well as re-record workflow of command. However, the command workflow supports re-recording instead of fragment modification.

(3) All modification works only after clicking OK; The button of Cancel can restore the contents prior to modification, and even the re-recording flow can be restored.

🐲 Record Script	\overline{a}	23	ų	Script Record			- X
Мар Кеу			Γ	Key	N	ame	4
				Test	Test		
Name							
Description							
				4			
				Description		•	
Record Options				Description			
Manually finish.							
O Auto finish after all required options	s are s	set.					
OK Cancel				Save		Close	

4. Support save all current "Map Key"

(1) All **"Map Key"** in the current panel will be saved in a configuration file.

(2) Initialize the function of **"Map Key"** when the software restarts, and pre-load all **"Map Key"** in configuration file into memory, aiming to support fast trigger of **"Map Key"**.

(3) Only after clicking Save can all modification be read-in configuration file and actually complete backup. Otherwise, all **"Map Key"** modification will lose when user closes the software.





→Where it is

Menus >> Utilities >> Script Record

1.2 Equation Manager

1.2.1 External Excel Association and Drive Updates Support

In equation manager, newly support external Excel association and drive updates. User can import external Excel to equation manager and associate with it. When user modifies the content in Excel, the

associated content in equation manager will be synchronously updated. The button enables user to break the association between equation and Excel. The associated expressions have link icons as below shown.

💱 Equation Manager				∽ ∞
Expression List				
Filter All		🖾 👗 br/E	esktop\E	quation.xlsx 🗁 🖿 🕼
Name	Expression	Value	Unit	Туре
✓ ♣ Part043				
<u></u> a	100	100	mm	Number
<u>b</u>	150	150	mm	Number
<u></u> c	200	200	mm	Number

→Where it is

Part Environment >> Tools >> Insert >> Equation Manager

1.2.2 Equation Manager Improvement

Equation manager extends the physical quantity types and their unit types as well as 2 physical quantities including pressure temperature and their corresponding units.





Pressure Unit: Ра kPa MPa psi bars atm mmH2O mmHg inH2O inHg mN/mm^2 N/mm^2 N/cm^2 N/dm^2 N/m^2 lbf/in^2 lbf/ft^2 kgf/mm^2 kgf/cm^2 kgf/dm^2 kgf/m^2 at Temperature Unit: Celsius Fahrenheit Kelvin Rankine

→Where it is

Part Environment >> Tools >> Insert >> Equation Manager

1.3 Crash Report Improvement

Adapt to high resolution screen. With the improvement, the error report interface supports high resolution screen as well as normally identifies user's operational system information. Newly add CPU information. The computer information will be sent along with sending error report. The interfaces of application **CrashReport** and **Manager** have been merged to one application **ZW3DCrashReport.exe**.





ZW3D Error Report	×
ZW3D.EXE has encountered a problem and needs to close. We are sorry for the inconvenience.	1
You may try to save your changes, if the application is still responsive.	
We have created an error report that you can send to us. We will treat this report as confidential and anonymous.	
To help us diagnose the cause of this error and improve this software, please describe what you were doing when this error occurred, and send this report to us.	
Describe what you were doing when the error occurred (optional):	
The contents sending now may contain the information of the draw being manipulated.	ing
Click here to select contains to send.	
Don't Send Send Error Report	



2 Converter

2.1 Import Format Update

The following table shows the import format supported by ZW3D, and the newly update 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, CATDrawing, .CGR, .3DXML	V5R8 V5/V6R2020
NX(UG)	.prt	11 - NX 1899
Creo(Pro/E)	.prt, .prt*, .asm, .asm.*	16 – Creo 6.0
SolidWorks	.sldprt, .sldasm	98– 2020(Only 64 bit supported)
SolidWorks_2D	. slddrw	2013-2020(Only 64 bit supported)
SolidEdge	.par, .asm, .psm	V18 – SE2020
Inventor	.ipt, .iam	V6 - V2020
ACIS	.sat, .sab, .asat, .asab	R1 – 2020 1.0
DWG	.dwg	R11 - 2013
DXF	.dxf	R11 - 2013
IGES	.ige, .iges	
STEP	.stp, .step, <mark>.stpz</mark>	203, 214, 242
Parasolid	.x_t, .x_b, .xmt_txt, .xmt_bin	Up to 30.0
VDA	.vda	
Image File	*.bmp, *.gif, *.jpg, *.jpeg, *.tif, *.tiff	
Neutral File	*.z3n, *.v3n	
PartSolutions	*.ps2, *.ps3	
STL	*.stl	
3DXML	.3dxml	4.0 - 4.3
XCGM	.x cgm	R2012-2020 1.0
TL	.jt	6.4-10.4
ОВЈ	.obj	



2.2 **★**Adjustment to "Quick Import"

Optimize the file import efficiency of large 3rd format file. Increase the import efficiency from the following two aspects and optimize user's experience.

- 1) Introduce much more parallel mechanism to speed up the assembly drawing import.
- 2) Process import flow in the platform and optimize import flow.

The modified of import flow are as follows:

2.2.1 Adjustment to "Quick Import" UI

The new **"Quick Import"** cancels the limitation to user setting import mode. User can set some hidden component, suppressed component and free point, etc.

🖉 CATIA5 File Import	\bigtriangledown X						
CATIA5							
▼ Import from							
File C:\Users\Administrator\Desktop_093.CATPart							
▼ Import to	▼ Import to						
Ourrent Object							
O New Object							
O New File							
▼ General							
Repair gap	Auto activate part						
Auto create sub-part	 Import geometry set as layer Associative import Normal 						
Auto save files as full shatter							
Import mode							
▼ Read							
Free curve							
🗹 Free point							
🗹 Hidden component							
🗹 Sheet body							
Hidden entities							
Suppressed component							
PMI							
Default	DK Cancel						

2.2.2 Add Import Feature Node to "Quick Import"

1) Each part created by **"Quick Import"** will generate a history feature node. As for assembly import, multiple history feature nodes will be created. Each history feature node is only associated with display data of the current part. If it is an assembly node, then the display data of its history feature node is that the geometrics of the assembly node.



- 2) Each history feature node records the contents including the imported optional setting, and the 3rd format drawing path info that is corresponding to the current imported part.
- 3) User can redefine the history node created by "Quick Import" and can re-select the path of 3rd format drawing while re-defining. According to the re-selected path, if the path is different as before, the display data associated with the history feature node will be firstly cleaned, and then quickly re-import the display data under the new path. If an assembly drawing is selected, only its geometrics display data will be quickly imported.
- 4) In regeneration, use the current display data and it will not regenerate display data according to the history node generated by **"Quick Import"** but skip the node directly.
- 5) Require adding a feather tag to the history feature node generated by **"Quick Import"**. History regeneration will not be triggered by dragging the lightweight feature node.
- 6) The history node generated by "Quick Import" will increase "Load Model" command. If the corresponding 3rd format file of the history node does not exist, it will delete the display data of the history node accordingly. Meanwhile, load the corresponding geometrics data of the 3rd format and convert the current history node to general static data node. If the 3rd format file does not exist, then give a tip to user that the corresponding file cannot be found and terminate the loading model.

2.2.3 2nd Import Action in "Quick Import" (Import Real Geometric Data)

After **"Quick Import"**, what we get is only the display data not the real geometric data. Thus, after that, we need to import the real geometric data. Such 2nd import action may happen in the following scenarios:

- 1) User requires initiatively to "Load Current Model" or "Load All Models".
- 2) When user activates the corresponding part, prompt the user to load the real geometric data or not. If the user selects Yes, then load the real geometric data.

2.2.4 Remove "Quick View"

Since the new **"Quick Import"** function is basically the same as the current **"Quick View"** function when it is imported for the first time, the **"Quick View"** function is completely abandoned, and the relevant options and command entries will be removed.

→Where it is

File >> Import >> Quick Import



2.3 ★Convert Attributes Support

Support reading-in the user custom attribute of part while importing and reading-in the part attribute as well as hidden components, suppressed components, hidden entities, user custom attributes and material density. Formats and read functions are as below:

Format	Hidden Components	Suppressed Components	Hidden Entities	User Custom Attributes(1)	Material Density(2)
NX	Х	V	٧	√(3)	٧
Creo	Х	V	Х	V	V
SolidWorks	V	V	٧	√(7)	V
CatiaV5	V	V	V	√(4)	٧
Inventor	Х	Х	Х	√(5)	V
Solid Edge	Х	Х	V	√(6)	V

Remark: X means unable to be read; √ means able to be read.

(1). User custom attributes are only captured on the part, assembly, and component instead of the layer of part internal shape.

- (2). At present mater density can only acquire the material name and its corresponding density.
- (3). NX user custom attribute name contains two parts: **Category** and **Title**. After converting **ZW3D** user customer attribute, the attribute name shall be **Category** |**Title**

(4). **Catia V**5 supports acquiring the user custom attributes of assembly or sub-assembly (such as part number and version and so on), but the user custom attributes of component is not supported (such as instance name and so on).

(5). **Inventor** has limits to support the following user custom attributes: Support the attributes in the options of **Custom, Project and Status** but excluding **File Status** of Status.

(6). **SolidEdge** supports user custom attributes started from ST4 while it is not supported in the older versions.

(7). **Solidworks** has limits to support user custom attributes import. The information under **Summary** option cannot be imported to ZW3D.

→Where it is

File >> Import





2.4 Other Adjustment

2.4.1 DWG/DXF Export Add "Explode hatch as single line" Option

In 2D engineering environment, while exporting **DWG/DXF** format, section line is exported as the type of pattern fill block. After checking the option of "**Explode hatch as single line**", the section line will be exported to **DWG/DXF** file as the type of single line.

DWG/DXF File Generation		Π Σ					
DWG/DXF Filter							
▼ Export to							
File C:\Users\Administrator\Documents\ZW3D_0932_2D.dwg							
▼ General							
File type O DWG O DXF(ASCII) O DYF(Binand)							
Version 201	3	•					
▼ Ontional							
Reduce data		0.01					
Trim surface	UV Curve	0.01					
Free surface	Curve	0.01					
Merge arc	Surface	0.01					
]					
Color							
▼ Drawing	-						
Export active sheet only	Export all shee	ts					
Export to model space	Export to layou	ut space					
Export sheet format entities							
Drawing view							
Export as curves	O Export as	block					
O Export as block for each comp	onent						
Export to new layer for each	part						
Scale output 1:1 Default	• 1						
Remove text font info type	None	•					
Explode hatch as single line							
Default	OK Cancel						

2.4.2 DWG/DXF Import Add "Placement" Field

While import DWG/DXF format to current object and current object is a part, add "Placement" field. The drop-down list of "Placement" allows you to select all CSYS under current object. After importing, the origin of the model aligns with the origin of the selected CSYS.

💯 DXF DWG Imp	oort Options				Ę	2	23		
DWG/DXF F	ilter								
Import from	Import from								
▼ Import to	▼ Import to								
Import to	Import to			Target					
 Current ol New object 	bject -+		Part	t					
 New file 			O She	et					
Placement	Default (CSYS 🔻	Sket	tch					
▼ General									
🗹 Auto sew g	geometry		🗹 Auto activate part						
Auto creat	e sub-part		Convert polylines into curves						
Refine poly	 Refine polyline curves Convert polygon mesh to 			Read empty layers					
Convert polyg				STL blocks 🔻					
▼ Unit									
-File units			ZW3D units						
 Use file u Read as 	nits Millimet	ers 🔻	C	onvert to	Millimeter	s	-		
▼ Read									
✓ Model spa ✓ Paper space	✓ Model space✓ Paper space								
•							•		
	Default	OK		Cancel					



3 CAD

3.1 Sketch Design

3.1.1 Improvement of "Conflicting Constraints Check"

In sketch environment, when a conflict occurs to constraint/dimension, all conflicting constraints and dimensions will list in Conflicting Constraints Manager where user can delete constraints/dimensions or convert dimensions to reference dimensions. Newly add the pull-down list to filter display content, including **Show dimensions only**, **Show constraints only** and **Show all**.



→Where it is

Sketch Environment >> Constraint >> Conflicting Constraints

Configuration >> 2D >> Sketch >> Enable constraint conflicting manager

3.1.2 **★**New "Equal Curvature"

In sketch environment, add **"Equal Curvature"** which can be added onto curve and curve, curve and arc, curve, and line. The constraint lines need end to end. After constraint, two lines G2 continuity (curvature equal).



	2 Equal Curvature	23
NR	✓ X	0
	▼ Required	
	Curve	
	Curve	

→Where it is

Sketch Environment >> Constraint >> Equal Curvature

3.1.3 New Character Spacing in ReadySketch Text

User can adjust the characters spacing through ReadySketch Text to better modify the value between characters spacing.

A ReadySketch Text		
✓ X		0
▼ Required		
Origin	4962.04,5846.	64 🛛 🕹 🍷
Text	ZWSOFT	₫
▼ Font		
Font	ZW3D Triplex	Roman 🔹
Style	Regular	•
Size	20	mm 🌲 垫 👻
Spacing	1	‡ 🖢 *
▼ Settings		
Curve		<u>.</u>
Mirror		
20 Z		

→Where it is

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



3.1.4 Support Relocate Sketch Plane, Sketch Direction and Sketch Origin in Sketch Editing

In sketch environment, add the function of **"Relocate"**, by which user can relocate sketch in the process of sketch editing and does not need to exit from sketch at first.

kelocate			23
▼ Required			
Plane	F3		
Up		$\stackrel{\scriptstyle >}{\scriptstyle \sim}$	• 🖢
Origin		$\stackrel{\scriptstyle >}{\scriptstyle \sim}$	• 🖢
Reverse hor	zontal direction		
Use centroid	defined by Plane		

→Where it is

Sketch Environment >> Setting >> Relocate

3.1.5 **★**New Cosmetic Sketch

In 3D sketch environment, newly add the function of **"Cosmetic Sketch"**, by which user can add some symbols or graphic tags in part environment or draw some geometric figures onto other feature or model (such as logo).







Three characteristics of "Cosmetic Sketch":

1) Annotation: The sketch is an annotative sketch, which means it cannot be used in the features of modeling.

2) Hatch pattern: After exiting **"Cosmetic Sketch"**, user can add hatch pattern in closed area in **"Cosmetic Sketch"**.

3) Projected: The cosmetic sketch can be projected to the surface of entities and engineering drawings.

→Where it is

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

3.1.6 Display tinted closed ring

1) Display tinted closed ring. During drawing, the general sketch can hatch closed area in entity (pure color).



2) Closed area check. Prompt the user that exist nonstandard geometries to check closure when he or she exits sketch in creating a sketch with modeling command.







→Where it is

Sketch Environment >> DA Toolbars >> Closed Rings On/Off

3.1.7 New "2D Sketch Offset Constrain"

When the command is executing by sketch, add offset constraint by default. Analyst geometrics like line, circle/arc, polyline, and rectangle and so on are supported. Display offset constraint icon and offset distance dimensions between the original geometry and the offset geometry. Use can modify offset distance through dimensions as well as delete offset constraints and dimensions.



→Where it is

Sketch >> Curve >> Offset

3.1.8 New Option of "Hide used sketch"

Newly add the option of **"Hide used sketch"** to control whether the used sketch will be automatically hidden.

→Where it is

Configuration >> Display >> Toggle settings >> Hide used sketch





3.2 Wireframe Design

3.2.1 Improvement of "Project Curve"

Two mayor improvements of "Project Curve" include:

- 1) Break prior to project to curve and clean overlap after project to curve.
- 2) Solve failed situation of project to curve.
 - a) When the direction is not defined, the circle cannot be projected to sphere.
 - b) Issues related to project to seam edge.
 - c) Incorrect results of 2D sketch reference and offset.

3.2.2 Improvement of "Curve Modify" Align Type

Newly add 6 **"Curve Modify"** align types, including Auto Plan, Plane, WCS, View, Tangent and Normal as follow:

Auto Plane: auto search the least square plane of control point in curve, and the control point moves in the plane.

Plane: use custom defines input plane where the control point can move.

WCS: no limitation to the control point which can move to any point.

View: the control point can move in the current view.

Tangent: the control point moves upwards to the selected tangent line.

Normal: the control point moves upwards to the selected normal line.



🔅 Modify	23
🗸 🗶 🖪	0
▼ Required	
Curve	<u>∲</u>
Point	₫
► Constraints	
Align type	Plane 🔻
Align plane	Auto plane Plane
Use orientation tool	WCS View
▼ Modification	Tangent Normal

→Where it is

Part Environment >> Wireframe >> Edit Curve >> Modify

3.3 Part Design

3.3.1 ★Modeling Dimensional Accuracy Update

The minimum modeling accuracy is supported from **0.0001mm** to **0.00001mm**.

It is suggested that the maximum model accuracy is no more than modeling accuracy*10e+9 while the minimum feature dimension is no less than modeling accuracy *10e-5.

3.3.2 ★Complete CSYS Support

Compose a complete Cartesian coordinate system, including Datum axis, Datum plane, Datum CSYS,

Datum axis 2D and add features in history according to the created timing sequence (sketch axis excluded).

The platform has only one local coordinate system **(LCS)** which is default as modeling reference system with globally invariant world coordinate system **(WCS)** as its default value.

Replace XY, YZ, ZX features on history tree with a value of WCS Datum CSYS feature. It is only a general Datum CSYS feature and does not present WCS or LCS. And it can also be redefined.

3.3.2.1 Datum Axis

Datum axis only contains one orientation and defines origin and length. The datum axis command panel shows as below:

/ Datum Axis		23
🗸 🗶 🖪		0
▼ Required		
😢 🋊 🗠	s 🛃 🛃 🦯	1/2 👄
Geometry		🛬 🔹
Length	9.6	mm 🗘 掛 *
▼ Orientation		
Flip direction		
▼ Datum Attribute	es	
Custom attribut	tes	
Color		
Style		
Width		
Datum format	Arrow	•

Datum axis format includes Arrow and Line which can be set color, style and width.



Datum axis has the following characteristics:

1. Independent Entity



Datum axis is an independent entity and there is a single feature node in history tree. Support common editing operations, including pattern, mirror, move and copy. It can be managed in layer manager and has corresponding filter type DatumAxis.

2. Orientation Input

Execute such as Extrude, Pattern and Project Curves. As for the commands of vector\direction input in the platform, the option of Datum Axis is added to filter with only allowing to select Datum axis object, Datum plane object Z axis and Datum CSYS object has three datum axes. When the filter is Edge/Curve, disallow to select all the above-mentioned objects.

3. Dimension PMI

Support Quick Dimension、Linear、Angular、Symmetry.



- 4. Assembly Constraints
- 1) Insert command: snap axis origin; coincide component origin and axis origin; component Z axis orientation is identical with datum axis orientation.
- 2) Common constraint: parallel, vertical, angle, coincident.

→Where it is

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

3.3.2.2 Datum axis 2D (Sketch)

The command panel is as follows: If "Construction geometry" is checked, create only internal axis to use; otherwise, create external axis to use and work as sketch's revolution axis by default.

1⁄2 Axis	23
🖌 🗙	0 2
Required	
///	<u>∻⊦∠_i</u> ™
1st point	-60,-15 🛛 🗧 🍷 🔹
2nd point	-35,20 😤 🌜 🔹
▼ Settings	
🗹 Display gu	ides
Constructi	on geometry

2D datum axis is an infinite hidden line. The internal axis shares the same display attributes with the general constructive line as grey hidden line. The external axis is brown hidden line. Use can switch the internal and external axis through **Toggle External/Internal**.



2D Datum axis has the following characteristics:

1. Default XY

Sketch contains two XY objects by default, infinitely long, and default XY cannot be modified or deleted.





2. Independent Entity

Support common editing operations, including Pattern, Mirror, Move, Copy and Revolve.

3. Constraint

Support 2D line constraint (excluding equal length and point to center).



4. Dimensions

Support 2D line dimensions.

5. Revolve axis





→Where it is

Sketch Environment >> Sketch >> Drawing >> Axis

3.3.2.3 Datum CSYS

Datum CSYS is composed of origin, three X/Y/Z datum axes, and three datum planes. The origin is something like a dot entity that plays as dot capture reference. The three datum axes can be treated as an independent entity that used as general orientation reference. The three datum planes can be used as a general plane. The coordinate system can be regarded as an independent entity.

The command panel goes as below:

🅌 Datum CSYS		23
🗸 🗙 🖪		0
▼ Required		
	< 🎉 🎉 📣 🗳	F
Geometry		₫
Orientation		
•	K 🕺	
Offset	0 mm ‡	- 💆
Origin	*	<u>⊸</u> .
X point	*	₫ •
X angle	0 deg 🗘	- 🛃
Y angle	0 deg 🗘	- 💆
Z angle	0 deg 🗘	- 💆
▼ Datum Attribute	es	
Custom attribut	tes	
Color		
Style		-
Width		Ψ.
Datum format	Default	*

Three datum axes, three datum planes and one origin are drawn in the drawing area. Each datum axis is marked as its name (X, Y, Z). The coordinate system works as a whole selection by default, and its default color is brown, which shows as below:

Z	
Y	

The datum CSYS has the following characteristics:

1. As an independent entity

Datum CSYS exists as an independent object and corresponds to an independent modeling history tree, which can be executed by common edit commands such as Pattern, Mirror, Move and Copy. There exits corresponding **Datum CSYS** filter object. It can be treated as an independent object as well as managed by layer manager. Besides, it has unique attributes such as visibility, color and line thickness, etc.





2. Component members

It is composed by origin, X/Y/Z datum axes and three datum planes.

3. Constraints

The coordinate system aligns with CSYS.

→Where it is

Part Environment >> Shape >> Datum >> Datum CSYS

Newly add option "CSYS" in Default datums of Configuration and set "CSYS" as default datum. After setting, the default "CSYS" is created when a new part is created. Keep the original "Triad" option, you can still switch to the old "Triad" mode through this option.



→Where it is

Configuration >> Part >> General >> Default datums

3.3.3 New Method to Create Datum Plane

Create a datum plane with 2 entities. Newly add the two following entities to select.

- 1) Select arc surface and datum plane to create plane.
- 2) Select datum plane and line to create plane.

→Where it is

Part Environment >> Shape >> Datum >> Datum Plane >> 2 Entities

3.3.4 Geometries Direct Cross File Copy via "Ctrl+C/Ctrl+V" Support

Newly add hotkey "Ctrl+C/Ctrl+V" to copy geometries (Curves, Surface, Entity). The copied geometries will be pasted as static ones.



3.3.5 ★New Cross-section Type "G2 Blend" in Fillet Command

Rename "Arc Type" label to "**Cross Section Type**" and add "**G2 Blend**" option to the drop down list. When the "**G2 Blend**" option is selected, a pair of "**Start/End Weight Sliders**" will be displayed for further cross section shape control.

🌍 Fillet		Σ3
🗸 🗙 🖪	F	0 2
▼ Required		
	I I I I I I I I I I I I I I I I I I I	
Edges E	*	:
Radius R	5 mm 🗘 堡 🔻	
▼ Variable Radiu	us	
Hold line	*	:
Variable radius	5	-
Add	Modify Delete	
Vertex	<u>4</u>	
Exception	4	
▼ Shape of Fille	:t	
Relief	0 mm 🗘 👲 י	-
Cross-section ty	rpe G2 Blend	•
Start weight	1.000 🗘 —	-
End weight	1.000 🗘 —	-
Rollover Cont	trol	
Settings		-

→Where it is

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

3.4 Free Form Design

3.4.1 New "Cross Trim"

Add a new command Cross Trim in Free Form Ribbon, by which user can decide which side to keep when to select more than two sheets or faces and select whether to automatically sew the remaining or not.

	🐓 Cross Trim 🛛
	▼ Required
	Entities 2 picked 🛛 🕹
	Faces to remove O Faces to keep
	Regions 2 picked 🗧 😤 👻 🕇
	▼ Settings
+	Sew shapes
	▼ Tolerance
	Tolerance 0.01 mm 🗘

→Where it is

Part Environment>> Free Form >> Edit Face >> Cross Trim

3.4.2 **★**"Blend Face" Command New Option "G2 Blend"

Renamed "Arc Type" label to "Cross Section Type" and added "G2 Blend" option to the drop down list. When the "G2 Blend" option is selected, a pair of "Start/End Weight Sliders" will be displayed for further cross section shape control.

🌺 Blend Face		23
🗸 🗙 🖸		0
▼ Required		
		200
From		
То		_ ₫
Through		_ ₫
▼ Orientation		
Spine	Na	itural 🔻
Shape of Bler	nd	
Cross-section ty	/pe	G2 Blend 🔹
Start weight		Circular
End weight		G2 Blend
▼ Sew		
Sew shapes		
Shapes		*
Settings		
Auto Reduce		
► Tolerance		

→Where it is

Part Environment >> Free Form >> Basic Face >> Blend Face

3.4.3 **★**"Fillet Open Faces" command New sub menu "Shape of Fillet"

Added a new sub menu section called "Shape of Fillet", where "Circular", "Conic," and "G2 Blend" cross section types can now be specified. When the "G2 Blend" option is selected, a pair of "Start/End Weight Sliders" will be displayed for further cross section shape control. When the "Conic" option is selected, "Conic Ratio" input is supported.

💰 Fillet Open Fa	ices	23
🗸 🗶 🖪		0 2
▼ Required		
1st face		×
Reverse side		
2nd face		_ ×
Reverse side		
Radius	5	mm 🏮 堡 👻
▼ Shape of Fillet		
Cross-section ty	pe Circular	•
▼ Settings	Circular	
Base faces	No G2 Blend	ī

→Where it is

Part Environment >> Free Form >> Edit Face >> Fillet Open Faces

3.5 Sheet Metal Design

3.5.1 **★**New "Zero Radius Flange"

This function aims at supporting zero radius flange, including its construction, modification, folding and unfolding, and identification import. Add zero radius flange to six commands including **Full Flange**, **Flange** with **Profile**, **Partial Flange**, **Hem Flange**, **Fold by Line** and **Jog**. As for the bend radius less than modeling accuracy, all will convert to zero radius flange.

🌜 Full Flange			23	
🗸 🗶 🖪				
▼ Required				
Edges	1 picked	:	<u>₽</u>	
Flip the flam	iges			
▼ Flange Parar	neters			
Position	i, i		•	
Bend Attribute	utes			
Bend	Simple		•	
Radius R1	0	mm 🗘 💆	-	
Angle	90	deg 🗘 👲	-	
Length type	K_ jK	il 🥂		
Length	40	mm 🗘 垫	•	

3.5.2 Improvement of "Punch"

Once "Punch" is interacted with punch mental plane, punch will be completed. Support "Punch" to select face excluded for forming open.

Support external part model as "Punch". Location is provided to locate related modeling. Related with source is to control whether there are relations and decides to update punch result as external "Punch" model updates. Add "Punch" library to the platform standard library, which allows user to build own "punch" library.

The command panel goes as below:



Required Base F2 Punch P S2 Open O 1 picked Add fillet Radius 2 mm mm multi.Z3PRT S2.200 S2.200 Mathematical Stress Punch/项目\punch/项目test file\22.Z3PRT S2.200 S2.200 Punch S2.200 Mathematical Stress Required Int.Z3PRT 22.23PRT 33.Z3PRT 44.Z3PRT S5.Z3PRT multi.Z3PRT Part001.Z3PRT Part002.Z3PRT		23
Required Base F2 Punch P S2 Open O 1 picked Add fillet ** Radius 2 Mathematical Problematical Problematic	✓ X	0
Base F2 Punch P S2 Open O 1 picked ✓ Fillet ✓ ✓ Add fillet mm \$ < < Radius 2 mm \$ < ✓ Y ✓ Y ✓ Y Ø Y ✓ Y Ø Y <th>▼ Required</th> <th></th>	▼ Required	
Base F2 Punch P S2 Open O 1 picked ✓ Fillet ✓ ✓ Add fillet ////////////////////////////////////		P 5
Punch P S2 Open O 1 picked ▼ Fillet ✓ ✓ Add fillet Radius Radius 2 Mathematical Product State ✓ ✓ Punch ✓ ✓ <t< td=""><th>Base</th><td>F2 💆</td></t<>	Base	F2 💆
Open O 1 picked ▼ Fillet ✓ Add fillet Radius 2 Mathematical Provided File Ørench Ørench Ørench <th>Punch P</th> <td>S2 🔮</td>	Punch P	S2 🔮
▼ Fillet ✓ Add fillet Radius 2 Mm \$ € * Punch ② ♥ Required ● Required ● PunchI页目 test file\22.Z3PRT 11.Z3PRT 2.Z3PRT 3.Z3PRT 44.Z3PRT 55.Z3PRT multi.Z3PRT Part001.Z3PRT Part002.Z3PRT Part002.Z3PRT Part002.Z3PRT Part002.Z3PRT Frame ● ♥ Related with source ♥ Fillet Ø Add fillet Radius 2 mm \$ € * •	Open O	1 picked 🛛 🕹
✓ Add fillet Radius 2 Mathematical Properties of the source ✓ Punchi页目、punchi页目 test file(22.Z3PRT) Base F2 ● ● ● ● 11.Z3PRT 22.Z3PRT 33.Z3PRT 44.Z3PRT 23.Z3PRT 44.Z3PRT Part001.Z3PRT Part002.Z3PRT Part02.Z3PRT Pa	▼ Fillet	
Radius 2 mm \$ ● Punch ② ▼ Required ▼ Required ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ●	🗹 Add fillet	
Punch S3 ▼ Required ● ■ Required ● ■ Required ● ● ● <th>Radius</th> <td>2 mm 🗘 💆 🔻</td>	Radius	2 mm 🗘 💆 🔻
Punch Required		
 Required F2 Punch项目\punch项目test file\22.Z3PRT 11.Z3PRT 22.Z3PRT 33.Z3PRT 44.Z3PRT 55.Z3PRT multi.Z3PRT Part001.Z3PRT Part002.Z3PRT Part002.Z3PRT Vertilet Related with source Fillet Add fillet Radius 2 mm ⁺, ⁽¹⁾/₂ ⁽¹⁾ 	Punch	8
▼ Required Base F2 ▶-Punch顶目\punch顶目test file\22.Z3PRT 11.Z3PRT 22.Z3PRT 33.Z3PRT 44.Z3PRT 55.Z3PRT multi.Z3PRT Part001.Z3PRT Part002.Z3PRT Part02.Z3PRT Part02.Z	✓ X	0 2
Base F2 受 Punch顶目\punch顶目test file\22.Z3PRT (**) 11.Z3PRT 22.Z3PRT 33.Z3PRT 44.Z3PRT 44.Z3PRT 55.Z3PRT multi.Z3PRT Part001.Z3PRT Part002.Z3PRT (*) Part002.Z3PRT (*) Frame (*) Image: Contract of the source (*) Fillet (*) Add fillet Radius 2 mm (*) (*)	▼ Required	
Base F2 -PunchI须目\punchI项目test file\22.Z3PRT 11.Z3PRT 22.Z3PRT 33.Z3PRT 44.Z3PRT 55.Z3PRT multi.Z3PRT Part001.Z3PRT Part002.Z3PRT Location .06784,-22.4128,-30 ✓ Frame ✓ ✓ Fillet ✓ Add fillet Radius 2 mm * *		A 🖡
Punch顶目 \punch顶目 test file\22.Z3PRT 11.Z3PRT 22.Z3PRT 33.Z3PRT 44.Z3PRT 55.Z3PRT multi.Z3PRT Part001.Z3PRT Part002.Z3PRT Part002.Z3PRT Part002.Z3PRT Part002.Z3PRT Vertice ✓ Frame ✓ ✓ Fillet ✓ Add fillet Radius 2 mm *, ④	Base	F2
11.Z3PRT 22.Z3PRT 33.Z3PRT 44.Z3PRT 55.Z3PRT multi.Z3PRT Part001.Z3PRT Part002.Z3PRT Part002.Z3PRT Frame ✓ Related with source ✓ Fillet ✓ Add fillet Radius 2 mm *	-Punch项目\pu	nch项目test file\22.Z3PRT 🔚
Location A.06784,-22.4128,-30 ≥	11.Z3PRT 22.Z3PRT 33.Z3PRT 44.Z3PRT 55.Z3PRT multi.Z3PRT Part001.Z3PRT Part002.Z3PRT	
▼ Fillet ✓ Fillet ✓ Add fillet Radius 2 mm ‡ 5	Location Frame Related with	1.06784,-22.4128,-30
✓ Add fillet Radius 2 mm ⁺ [●] [●] [●]	▼ Fillet	
Radius 2 mm 🗘 🥸 🕇	Add fillet	
	Radius	2 mm 🗘 💆 🔻

→ Where it is

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





3.6 ECAD

Check the option of "One object per file (new files)" in General of Configuration in advance.

→ Where it is

Part Environment >> Configuration >> General >> One object per file (new files)

3.6.1 Import IDF File

3.6.1.1 New "Import IDF File"

The import steps go as below: click "File >Import>Import...", and the following dialogue will pop up after selecting required file to import. After setting the import options, click "OK" to import IDF file.

Import from			
Board file C:\Users	Administrator\[0esktop\01\TEST.emn	
Library file C:\Users	Administrator\[esktop\01\TEST.emp	
▼ Input			
Default board thick	ness	0	4
Default component	height	0	;
Default hole diamet	ter	0	

→Where it is

File >> Import >> Import...



3.6.2 Export IDF File

3.6.2.1 New "Export IDF File in ECAD Environment"

The export steps are as following: click "File>Export>Export...", and then the following dialogue will pop up and click "OK" to export IDF file after setting the board file, library file and export version, etc.

IGES File (*.igs;*.iges) STEP File (*.stp;*.step) DWG/DXF File (*.dwg;*.dxf)	🐲 ECAD Export Options	₽ %
Parasolid Text File (*.x_t) Parasolid Binary File (*.x_b)	ECAD Filter	
VDA File (*.vda) ACIS Files (*.sat;*.sab) STI Bile (* stl)	Export to	
OBJ File (*. obj) VEML File (*. wrl)	Board file C:\Users\Administrator\Documents\ZW3D\Part0	<u> </u>
Neutral File (* vxn) Neutral File (* z3n)	Library file C:\Users\Administrator\Documents\ZW3D\Part0	<u> </u>
HTML File (*.html;*.htm) Bitmap File (*.bmp) CTP Tarra File (*.aif)	▼ Generation	
JPEG Image File (*. jpg;*. jpeg) PNG Image File (*. png)	Export version IDF 3.0	-
TIFF Image File (* tif;* tiff) PDF File (* pdf)		
CATIA V4 File (*.model) CATIA V5 PART Files (*.CATPart)		
CARFILE (* cgr) CGR File (* cgr) VCCM File (* cgr)		
3000ML File (* 3dxml) TT File (* jt)		
IFC File (*.ifc) Intermediate Data Format (*.emn;*.brd)	Default OK Cancel	
Intermediate Data Format (*. emn;*. brd 🗸		

→Where it is

File >> Export >> Export...

3.6.2.2 New "Export IDF in Part Environment"

The export steps are as below: click "File>Export>Export...", and the following dialogue will pop up and click "OK" or "Apply" to export the corresponding file after settings.



TGES File (* igs:* iges)				
STEP File (*. stp;*. step)	Export IDF			23
DWG/DXF File (*.dwg;*.dxf)				
Parasolid Text File (*.x_t)				
Parasolid Binary File (*.x_b)	V 🗛 🗳			
VDA File (*.vda)				
ACIS Files (*.sat;*.sab)	Required			
STL File (*.stl)				_
OBJ File (*.obj)	Coordinate syst	em	Default CSVS	-
VRML File (*.wrl)	coordinate syst		bendan estis	_ <u>~</u>
Neutral File (*.vxn)	-		50	
Neutral File (*. z3n)	lop side		F2	⊻
HTML File (*.html;*.htm)				
Bitmap File (*.bmp)	Bottom side		F1	-
GIF Image File (*.gif)				
JPEG Image File (*.jpg;*.jpeg)	- 0.11			
PNG Image File (*.png)	 Options 			
TIFF Image File (*. tif;*. tiff)				
PDF File (*.pdf)	E 1.01		+-\ 7\//2D\ D-+070	
	Export file	men	TC\//WVSU/Partu/yiemn	
CALLA V4 File (*.model)	Export file	men	ts\ZW3D\Part0/9.emn	
CAILA V4 File (*.model) CAILA V5 PART Files (*.CATPart)	Export file	men	ts\ZW3D\Parto/9.emn	
CAILA V4 File (*.model) CATLA V5 PART Files (*.CATPart) CATLA V5 ASSEMBLY Files (*.CATProduct)	Export file Export version	men	3.0	
CATTA V4 File (*.model) CATTA V5 PART Files (*.CATPart) CATTA V5 ASSEMBLY Files (*.CATProduct) CGR File (*.cgr)	Export file Export version	men IDF :	3.0	·
CATTA V4 File (*.model) CATTA V5 PART Files (*.CATPart) CATTA V5 ASSEMBLY Files (*.CATProduct) CGR File (*.cgr) XCGM File (*.cgr)	Export file Export version Output as	IDF : Boar	3.0	•
CATTA V4 File (*.model) CATTA V5 PART Files (*.CATPart) CATTA V5 ASSEMBLY Files (*.CATProduct) CGR File (*.cgr) XCGM File (*.scgm) 3DXML File (*.3dxml)	Export file Export version Output as	IDF 3 Boar	3.0	•
CATIA V4 File (*.model) CATIA V5 PART Files (*.CATPart) CATIA V5 ASSEMBLY Files (*.CATProduct) CGR File (*.cgr) XCGM File (*.xcgn) 3DXML File (*.3dxml) JT File (*.jt)	Export file Export version Output as	IDF : Boar t log	3.0	•
CATTA V4 File (*.model) CATTA V5 PART Files (*.CATPart) CATTA V5 ASSEMBLY Files (*.CATProduct) CGR File (*.cgr) XCGM File (*.xcgn) 3DXML File (*.sdxml) JT File (*.jt) IFC File (*.ifc)	Export file Export version Output as	IDF : Boar t log	3.0	•
CATTA V4 File (*.model) CATTA V5 PART Files (*.CATPart) CATTA V5 ASSEMBLY Files (*.CATProduct) CGR File (*.cgr) XCGM File (*.scgm) 3DXML File (*.sdxml) JT File (*.jt) IFC File (*.ifc) Intermediate Data Format (*.emn;*.brd)	Export file Export version Output as	IDF : Boar t log	3.0	•

→Where it is

File >> Export >> Export...

3.6.3 New "Create" ECAD Module

3.6.3.1 Create ECAD File

The steps to create an ECAD file are as following: click "File>New", and then select "ECAD" type and click "OK" after naming the file.



💯 Create New File				$\overline{\nabla}$	23
Туре					
a	۲	Ś	1		
Part/Assembly	Drawing Sheet	Standalone Sketch	CAM Plan		
ECAD					
Template	Inform	ation			
[Default]	Unique	Name			
	Part00	6.Z3PRT			
	Descrip	otion			
		Г	OK	C	
		L	UK	Cance	9

→ Where it is

File >> New >> ECAD

3.6.3.2 New/Edit Board



New/Edit

Click **Board** is used to create a new PCB model or edit an existing PCB model.

The steps to create are as following: click "New/Edit board", and then the following dialogue will pop up. Click "OK" after naming the file.



💯 Create a PBC board		$\overline{\nabla}$	23
Part			
BOARD			
Description			
	ОК	Cance	:

→ Where it is

ECAD Environment >> Assembly >> Component >> New/Edit Board

3.6.3.3 Region Set

Use "Region Set" to set cosmetic sketch as a region with specified attributes, where user can define "Routing Region", "Placement Region", "Via Region", "Other Region" and "Placement group region".

🗢 Region Set		23				
✓ X ⊆						
Required						
		\				
Profile P		ا الله الله الله الله الله الله الله ال				
▼ Properties						
3D volume						
Height	0	mm 🏮 塗 👻				
Туре	Keep-in	•				
Owner	MCAD	-				
Color						

- Routing region: set a closed profile cosmetic sketch as a routing region which can be set to allow/disallow routing region.
- Placement region: set a closed profile cosmetic sketch as a region that allow/disallow to place.
- Via Region: set a closed profile cosmetic sketch as Via region.
- Other region: set a closed profile cosmetic sketch as other region which is namely user custom region.





• Placement group region: set closed profile cosmetic sketch as a placement group region.

→ Where it is

ECAD Environment >> ECAD Board >> Engineering Feature >> Region Set

3.6.3.4 Region Cut

Use "Region Cut" to create a cut in PCB module. Notice that the selected sketch must be profile closed.

	Region Co	ut ?		2	
	Profile P			. الله الله الله الله الله الله الله الل	
ager Show M ♣ BOA > ≌ Sc ¥ ⇔ H ♥ ♥	Iost RD abid(1) istory For Default CSYS Tab1 C)Sketch2 Cut1 CONTROL STOP HERE		X		
Repl	ау				

→ Where it is

ECAD Environment >> ECAD Board >> Engineering Feature >Region Cut



3.6.3.5 Add Hole Attribute

Use "Add Hole Attribute" to add hole attributes onto PCB model, such as PTH or NPTH.

Add H	Add Hole Attribute					
	🗸 🗙 🖸	•				
	▼ Required					
¶_ ⊢□	Hole feature	<u>∎</u>				
_	▼ Attribute					
-	Plating style	PTH •				
	Owner	MCAD *				
÷						

→ Where it is

ECAD Environment >> ECAD Board >> Engineering Feature >> Add Hole Attributes

3.7 Assembly Design

3.7.1 New "Dissolve Sub-assembly"

Use command "**Dissolve sub-assembly**" to dissolve sub-assembly and move the component from the lower level to upper one.



3.7.2 New "Group as Sub-assembly"

Use command "Group as sub-assembly" to create a sub-assembly with some serverl components, and also allow to save sub-assembley with new file format or new object.





× 🔏	机	用虎钳。	装配体	\$		
	✓	🍞 (F)机	座			
	✓	🍞 滑动	夬			
	✓) 🗊 🗐	钉			
	✓	👕 活动	Ħロ			
	✓	🇊 (-)螺	70	Auto Regen		1
	✓	🍞 (-)护	1	Configure Component		
	<u>√</u>	🎔 护板		Group as Sub-assembly		
>		Constraii	1			
Group	as o	Sub-asser	mbly		\overline{a}	23
e Name		机用虎钳	.Z3			
t Name	e	Part001				
	Group A Rame	✓ ¾ ৠ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓ ✓	 ✓ ✓ ④ (F)机 ✓ ④ (F)机 ✓ ✓ ● (F)机 ✓ ✓ ● (-)螺 ✓ ✓ ● (-)螺 ✓ ● (-) (-) (-) (-) (-) (-) (-) (-) (-) (-)	 ✓ ✓ ✓ ● (F)机座 ✓ ● (F)机座 ✓ ✓ ● (F)机座 ✓ ● (F)机座 ✓ ● (F)机座 ✓ ● (F)根距 ✓ ● (F)根距 ✓ ● (F)根距 ● (F) ● (F)	 ✓ ✓ ✓ ● (F)机座 ✓ ● 滑动块 ✓ ✓ ● (-)螺钉 ✓ ✓ ● (-)螺钉 ○ Auto Regen ✓ ✓ ● (-)螺 ● Auto Regen ✓ ✓ ● (-)螺 ● Auto Regen ✓ ✓ ○ (-)螺 ○ Auto Regen ✓ ✓ ○ (-)螺 ○ Auto Regen ✓ ✓ 	 ✓ ▲ 机用皮钳 美容体 ④ (F)机座 ④ 滑动块 ④ (一螺钉) ④ (一螺钉) ④ (一螺钉) ④ (一螺钉) ④ (一螺管) ▲ Auto Regen ④ (一螺管) ▲ Auto Regen ④ (一螺管) ▲ Auto Regen ④ (一螺管) ① Configure Component ④ 护板 ● Constrain ♥ Group as Sub-assembly ♥ Group as Sub-assembly ♥ Rame 机用皮钳.Z3 t Name ♥ Part001

3.7.3 "Edit Constraint" Support Multi-select Components and Search Mutual Constraints

Add a new "Edit Constraint" to support multiple components selected and search their mutual constraints.

Edit Constraint		23
🗸 🗶 🖪		0
▼ Required		
Component(s)	2 picked	\approx
▼ Constraints		
Constraints		
Coincident 8 Pl	ane Plane	*
Coincident 9 Pl	ane Plane	<u> </u>
Concentric 4 C	ylinder Cylinder	Ŧ

→ Where it is

Part Environment >> Assembly >> Constraint >> Edit Constrains



3.7.4 New "Filter" in Assembly Tree

Add a filter button in search box of manager to control the show result.

Mana	ger		= X3
9 _	Show Most	•	Y

3.7.5 **★**New "Clearance Check"

Provide new assembly search command "Clearance Check" with the following characteristics:

- 1) Support for searching clearance value between components;
- 2) Support for searching clearance value between component and shape;
- 3) "Clearance Check" result displays in a table;
- 4) To export "Clearance Check" result in xlsx file is supported.



→Where it is

Part Environment >> Assembly >> Inquire >> Clearance Check



3.7.6 **★**New "Batch Attribute Edit"

Add a new command **"Batch Attribute Edit"** to provide user to batch edit part attributes. The command panel goes as below; its left list shows the attributes and values while its right list shows the objects to edit. User can modify multiple attributes of multiple components at one time.

The left list supports to import via external template, and the command will read the template's attribute and value. User can edit template file. Support select a component through **Input Format** button and take the component's attribute value as default value. Support Synchronous modification of attributes like material, density, part size and stock size, etc.

😨 Batch Attribute	e Edit						Ţ	53
Load Config	D:\Trunk\2500x64\	supp\Template.z	1			Pick objects to edit		
Include	Attribute	Value		ID		Name		
	Number	ZWSoft123	-	1		prt0072		
	Class	Part	-					
	Designer	Mark	•					
	Cost	12.99	-					
	Supplier	ZWSoft	-					
	Manager	Jason	-					
	CustomAttribut1	User Attrib1	-					
Material			-	🗹 Remain	current	list		
Density		kg/m^3	•	Mass	* =	1612188062	Target	Q
Part size	LxWxH					Rename picked shapes		
	Auto calculate			Clear list	ed obje	ects after update		
Stock size	198.37x291.64x136	5.74(mm)		Hide list	ed obje	cts after update		
	Auto calculate		٢					
		ОК	Car	ncel A	pply			

The right component list supports user to select part and filter to select parts.

→Where it is

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

3.7.7 New "Include unplaced component"

The assembly adds the "**include unplaced component**" command, which is used to insert an unplaced component in the assembly tree. The command form is as follows:

🔏 Include Unpl	laced Component	t	23
🗸 🗶 🖪			0
▼ Required			
File/Part			
Part079.Z3			Q
		All	•
Preview	Off		•
Part config		~	S
▼ Settings			
Insert to layer	Active layer		-

Unplaced components will be displayed on the assembly tree with the unique icon of unplaced components (as shown in Part079 in the following figure). Because the components are not placed, the unplaced components will not be displayed in the graphics window. Unplaced components can be included or excluded when creating a bill of material and are not accounted for in physical properties calculations. You can select the unplaced components on the assembly tree and convert them to general



components through the right-click menu "**Insert Component**". After the components is converted to general components, it cannot be converted back to unplaced components.

Mana	iger			8 23
;,	Show All		•	Y
⊢ □	🗠 🔏 Part079			
_	🗸 😹 (–)Oi	ilCea	I_IN	
.	✓ 1 (−)	6	Open Part	
		\$	Insert Component	
7		2	Change Component	

→Where it is

Part Environment >> Assembly >> Component >> Include Unplaced Component

3.8 Drafting Design

3.8.1 Variable Input in Engineering Drawing Show Label Support

Add a new variable input in engineering drawing show label.



🖗 View Attributes	
Style <from standard=""></from>	▼
General Label Lines Text Com	q
	ً≱
😔 🚫 🛵 [
Inherit PMI	
Show scale	
Scale type Use custom scale	
◎ X/Y 1 ‡ : 2	2 \$
○ X.X 0.5	\$
Synchronize sheet scale	
Show label	
Label TOP	1
	Get variable
All Off Toggle A	ll On
Layer Layer0000	•
OK Cancel	Apply

→Where

Drawing Sheet Environment >> View Attributes >> Label >> Get Variable

3.8.2 **★**New "Point Table" and "Dimension Table"

Point Table: a table that user can extract point coordinate. It is used to tag the selected coordinate value through table, like BOM or hole table. Bubble tag, table split, and table are supported.

The point list has the following characteristics:

1) The input objects in point list are 3D coordinate points and non-sketch points in engineering drawing.





- 2) Extracting points in both 2D and 3D coordinates is supported.
- 3) When the selected point contains point of view, default 3D coordinate origin as reference point. When the selected point does not contain point of view, default 2D drawing origin as reference point.
- 4) When the selected point contains point of view, the reference point and point of view will move as view moves; otherwise, it will not.
- 5) When the point list contains different point of view, the default reference point will generate order according to view in priority.
- 6) Be similar with BOM or hole table. Bubble tag, split table and table style are supported.



Annotation			23
✓ X			0
Required			
	6% .)]	
Point list			\approx
Base point		\approx	٠ 🖢
Name			
▼ Item Numb	ers		
Starting ID	1	* *	<u>⊸</u> .
Starting lab	el A		
Regenerate	IDs after so	ort	
Show origin	n indicator		
▼ Table Forma	t		
Columns	Label		
Availab	le	Selected	
Point 3D		ID Deint 2D	
		Remarks	
Attribut	es	Default	
Sort by	ID	verant	41
Sort when	regeneratin	g	21
System	defined	User defined	ł

Dimension table: mark number and list table to the dimensions in drawing according to order.

The dimension table has the following characteristics:

- 1) Dimensions in table contain all dimension content, including symbols such as radius **R**, angle °, thread **M** and depth, etc. It is not only a number.
- 2) Dimension id will move as view and dimension move.
- 3) Dimension is associative with table. When the dimensions change, the table will update as along.



4) When one dimension is lost, the dimensions will change in red and the corresponding row in the table will remain.

Annotation	23
✓ X	0
▼ Required	
Dimension list	*
Name	
▼ Item Numbe	rs
Starting ID	1 🌲 🖞 🔹
O Starting labe	A
🔲 Regenerate I	Ds after sort
▼ Table Format	
Columns	Label
Available	Selected
	ID Value Remarks
Attributes	Default
Sort by	ID ~ 2↓
Sort when re	egenerating efined User defined

→Where it is

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





3.8.3 New Insert "OLE Object"

Add a new "OLE Object" to drafting environment and support insert "OLE" object to drafting.

→Where it is

Drawing Sheet Environment >> Dimension >> Symbol >> OLE Object

3.8.4 New 2D BOM Default Template Setting in Configuration

Add a 2D BOM default template setting in configuration and configure 2D BOM template through this filed. Default create this template when to create 2D BOM.

💯 Configuration		∑ ⊐
General	Internet browser	iexplore.exe
Part	ZW3D file folder	C:\Users\Administrator\Documents\ZW3D
2D	Working folder	C:\Users\Administrator\Documents\ZW3D
Color	ZW3D backup folder	C:\Users\Administrator\Documents\ZW3D_
Declarated	Save backup files in the same lo	ocation as the original
Background	Standard parts folder	
Display	🗹 Create instance in working file	C:\Users\Administrator\Documents\ZW3D\S
Files	Punch library	E:\2500_x64
CAM	Symbol library	
User	Clipboard library	
PDM	🗹 Session journal	ninistrator\AppData\Roaming\ZWSOFT\ZW3 \equiv
ECAD.	Export folder	
ECAD	Quick rollback cache folder	C:\Users\Administrator\Documents\ZW3D_
	Drafting	
	Paper size definitions	DEF_SHEET_MM
	BOM template	
	4	•
Reset Default		OK Cancel Apply

→Where it is

Configuration >> Files >> Drafting >> BOM template



3.9 ZWMold

3.9.1 "Trim Pin" Support to Select Multiple Components as Trim Body

Add a new "Trim Pin" to support select multiple parts as trim body.

→Where it is

Part Environment >> Mold >> Library >> Trim Pin



4 CAM

4.1 ★Enhancements of Full Machine Simulation

1. Support Machine Builder

The Machine Builder is used to construct the machine tool movement chain and components, edit the chain properties and other machine tool parameters. The entrance of the machine builder is in the FMS toolbar.



2. New FMS Setting Functionality

The FMS Setting dialog is used to set the Part, Stock, Machine and CNC controller what will be used for simulation.

💱 FMS Setting				23
Work_Assembly				
Stock	Part001_Stock.1			
Target Part	Part001			
Machine				
Machine	5x_table_C_on_A			
CNC Controller				
Controller	ZW_Fanuc_Mill			
	ОК	Cancel		

3. New Fanuc CNC Controller

New the Fanuc CNC controller to interpret the NC file which output from Fanuc post processor. The current version only supports Fanuc controller.

4. Support RTCP Functionality

In ZW3D 2021, we support the RTCP functionality in FMS, if the real machine tool has RTCP function, please turn on the RTCP option in FMS during simulation. If the real machine tool does not have RTCP function, please turn off the RTCP option and configure the .znc file for machine simulation.

E Setting	×
Simulation options	
Stop on clash	
Stop on first clash	
Stop on gouge	
Stop on axis limit	
Stop on tool change	
Analysis Setting	
-1.0	
-0.1	
0.0	
0.1	
1.0	
Background	
✓ Dark Purple □ Light Purple □ Light Blue □ Light Grey	7
5X Control	
✓ RTCP	
OK Cancel	





5. New Machine Register Functionality

Machine Register is used to set the coordinates of G54~G59 and the parameters of tool compensation.

ID		H	I Value		D Valu	e
1	0	.000		5		
2	0	.000		10		
3	1	0		15		
	Add			[Remove	
Fork Coordinate	Add e System			[Remove	
Fork Coordinate	Add e System	X		[Ү	Remove	Z
Fork Coordinate Name G54	Add e System 0.000	X	0.000	[Ү	Remove 62.1	Z
Fork Coordinate Name G54 G55	Add e System 0.000 0.000	Х	0.000	[Ү	Remove 62.1 0.000	Z
Work Coordinate Name G54 G55 G56	Add e System 0.000 0.000 0.000	X	0.000	[Ү	Remove 62.1 0.000 0.000	Z
Vork Coordinate Name G54 G55 G56 G57	Add e System 0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000	X	0.000 0.000 0.000 0.000	Υ	Remove 62.1 0.000 0.000 0.000	Z
Tork Coordinate Name 654 655 656 657 658	Add e System 0.000	X	0.000 0.000 0.000 0.000 0.000	[Y	Remove 62.1 0.000 0.000 0.000 0.000	Z

6. New Measurement tool

New Measure Distance, Show Coordinate and Inquire Arc Data these tools, easy to measure the model size and location in FMS.



Measure Distance –	
1st Point 1090.063,271.11 2nd Point 1035.000,129.56 Distance 1001.848 [mm] Dist-X 55.063 [mm]	9, 1373. 790 9, 383. 523
Dist-Y [141.550 [mm] Dist-Z 990.268 [mm]	1st Point 1035.000, 147.859, 1237.617 2nd Point 1035.000, 266.741, 413.932 3rd Point 1035.000, -261.831, 325.557 Radius 500.049 [mm] Diameter 1000.098 [mm] Angle 177.442 [deg] Center (1035.000, -67.168, 786.161) Normal [-1.000, -00.00, 000)
	Show Coordinate - X
	Coordinate (919.700, -207.798, 1199.403)

7. Support the Tool Compensation Function

FMS can simulate the tool path with tool compensation, user needs to set related parameters in the Machine Register.

	🗢 Previ	ous Page	1 🗘 / 1 👄 Next	Page	Machine Tee	Decistor			×		
	Status	Line	NC	^	Machine Toc	rregister			^		
			%		Tool Compens	ation					
		1	N1 G40 G17 G94 G49 G90 G21 G54		ID		H Value	D Value			1
		2	N2 G91 G28 Z0.0		1	0.000)	5.000			
		3	N3 G28 X0.0 Y0.0								
		4	N4 G90 A0.0 C0.0							Y	
X		5	N5 T1 M06								
		6	N6 \$1000 M03		A	dd		Remove			
		7	N7 M08		Work Coordin	ate System					
		8	N8 G90 G00 X-22.5 Y-40.5		Name	х	Y	Z	^		
		9	N9 G43 Z100. H1		G54	0.000	0.000	-67.900			
		10	N10 Z22.5		G55	0.000	0.000	0.000			l
		11	N11 G01 Z20. F50.		G56	0.000	0.000	0.000			l
	Þ.,	12	N12 G41 Y-29. F150. D1		G57	0.000	0.000	0.000			
		13	N13 X-29.	-	G58	0.000	0.000	0.000	~		
		14	N14 Y29. F250.		OF	Consel					
		15	N15 X29		UK	Cancel					

8. New the Simulation Speed Functionality

Simulation Speed functionality is used to control the simulation speed, the right side of the control bar speeds up the simulation.







9. Support the Display of NC Code and External NC Code Simulation

The newest FMS can display the NC code directly, but not show the .tp code as ZW3D 2020. And user can also input the NC code to simulation.

Prev:	ious Page	• 1 🗘 / 2	t Pa
Status	Line	NC	
		%	
	1	N1 G40 G17 G94 G49 G90 G21 G54	
	2	N2 G91 G28 Z0.0	
	3	N3 G28 X0.0 Y0.0	
	4	N4 G90 A0.0 C0.0	
	5	N5 T22 M06	
	6	N6 \$1000 M03	
	7	N7 M07	
	8	N8 G43 G90 G00 X-26.9713 Y-165.9481 Z51.2056 A-91.5248 C165.548 H22	
	9	N9 X-2.0234 Y-69.1467 Z53.8667	
	10	N10 G01 X776 Y-64.3066 Z53.9998 F50.	
	11	N11 X-3.6782 Y-63.5555 Z53.8853	
	12	N12 X-5.4191 Y-63.1049 Z53.8166 A-90.6394 C165.5139 F250.	
	13	N13 X-7.4672 Y-62.5749 Z53.7356 A-89.5972 C165.4736	
	14	N14 X-8.7791 Y-62.2353 Z53.6823 A-88.9297 C165.4482	
	15	N15 X-8.7793 Y-62.2352	
	16	N16 X-9.2073 Y-62.1214 Z53.6649 A-88.7228 C165.2671	
	17	N17 X-9.6785 Y-61.9962 Z53.6429 A-88.4952 C165.0688	
	18	N18 X-10.7443 Y-61.7128 Z53.5932 A-87.9805 C164.6203	

10. Encrypted the .Job, .Mch, .Ctl, .Wkp, .Tls files.

4.2 ★The Rough Turning Operation Supports Arbitrary Contour as Stock

1. Support the Stock Type of Profile as the Stock to generate tool path

2020	2021



 Operations 		Operations	
Y 😹 🗸 Rough Turn 1		 Test Test 1 	
Tool : Tool 1			
✓		• eatures	
A profile 1		Protile 1	· · · · · · · · · · · · · · · · · · ·
Store to the store		Stock : profile 2	
Stock : profile 2		Machine (undefined)	the second s
Machine (undefined)		Uutput	
Output			
	Y		Y States and the second s
1			

2. Support the Stock Type of Workpiece to generate tool path



4.3 New Arc Fitting Functionality in 2X Operation

2X operations added the Fit Arc option; the generated tool path will be as close to the arc as possible according to the parameters set.







Chord Tolerance -- Determines the accuracy of the arc

Max Radius -- Determine the max radius of the output G02/G03, if the radius of the fitted arc is greater than Max Radius, G01 will be output.

4.4 ★New Arc Fitting Functionality in Surface Engraving Operation and 5X Operations

In the past, the Surface Engraving and 5X operations can only output linear tool path, after added this Fit Arc option, the generated tool path of surface engraving operation will be as close to the arc as possible. And the 5x operations will output the arc in the XY, XZ, YZ planes of the coordinate of the operation.



4.5 New Regular Pentagon, Hexagon and Octagon Turning Tools

New Regular Pentagon, Hexagon and Octagon Turning Tools support rough, finishing and facing operations.

Pentagon Tool	Hexagon Tool	Octagon Tool





4.6 ★New Excel Format of Operation List

1. Support for output operation list in Excel format

In ZW3D 2021, the operation list can be output in Excel format, user can customize the Excel template as needed.

🦉 CAM Ope	ration List S	ettings	⊽ ⊠
▼ Settings			
Select Temp	late		
OperationL	ist Template	e OpListExcel.xsls	
Custom Var	iables		
Keyword	Name	Value	
	ОК		Cancel



								omenane Norme										ONC
								ongany name										Operation List
Sender:			Ref.#:				-		Receiver:					Ref.#:				
Machined Part: 5X	Demo Case_01						Programmer: Ac	iministrator					Date: 2020/4/1					
Material: None Set							Checked by:						TIME:9:30:38					
Quantity:							Estimated Machi	ning time:12MIN 495E0	c				Required Finished	Date:				
Archived Path: e:																		
Serial #	Program Name	Operation Name		Tool Parameters		Feed	1	Thickness	9	tep Value	Tool Diameter	<u> </u>	Tool Length	1		Tool Path Attribut	e	Comment
	Reflector .	Dealling & A	Tool #	Name	Speed(RPM)	2500	Side	Bottom	XY	2	10	Flute	Min.	Overhang	Z max	Zmin	Time	
2	Bottom	Profecut 4	T 1 H1	D10R0	4000	2500	0.15	0.15	7	1	10	20	0	40	-1	-21.05	1MIN 15SEC	
1	Bottom	Professor 5	T 1 HI	01080	4000	2500	0.15	0.15	7	1	10	20	0	40	-23	-27.85	52500	
4	Bottom	Sotalat 1	T180	D1080	4000	1000	0	-0.1	7	6	10	20	0	40	-0.1	-0.1	395FC	
5	Bottom	Profilecut 3	T1H1	D10R0	4000	600	0	0	0.08	1	10	20	0	40	-27.85	-27.85	1MIN 15SEC	
6	Bottom	Proffecut 6	T 1 H1	D10R0	4000	600	0	0	6	1	10	20	0	40	-22	-22	1MIN 38SEC	
7	Bottom	Profilecut 2	T 1 H1	D10R0	4000	600	0	0	7	1	10	20	0	40	-10	-10	1MIN SSEC	
		2	Rock Size: Length 100	.000 Width 100.000	0 Height 82.000									Z Nin -27.850 Z	Max -0.100			
236.508 mm										234.508 mm		-		K				
236.508 mm			2004000		Sgned by Program	amer:		Signed by Operator:	2		Sgned by Tea	m Leader:		Delivered to:				Date:
	Verified Result	[]Qualified						-gring by openator.	-									
		[]Unqualified																
	Operation	.ist Received?Date			Stock Received?Date			Machine #		Sta	ted Date			Complet	ted Date		Act	ual Nachining Time
								Machine 1										
-																		

2. Support for show or hide the default coordinate

User can show or hide the default coordinate as needed; this determines whether it will be displayed on the view of operation list.



3. Support the output view of the specified coordinate system





User can specify the coordinate system by Set/Cancel Active option of frame, then operation list will output the view according to the active coordinate system.



4. Increased output speed

Output the Excel format is faster than the Html format about half the time.

4.7 Independent the Web Editor Program of the Zw3D Post

In ZW3D 2021, user can web edit the .znc file on the independent the web editor, no longer dependent on Internet explorer and the port 8088.





4.8 New Tool Path Verification Progress Bar

New the progress bar in tool path verification, user can realize the simulation of drag and drop mode, which is easy to find the problem of tool path.



4.9 Support the Left to Right Cutting Direction in Groove operation

In ZW3D 2021, The Groove operation can cut from left to right.

📩 Type: Turn Groove	▼ Cutting Control				
Primary Frame	Cut Regions	OD		-	
Seatures	Cut Direction	Left to I	Right	+	
Tool and SpeedFeed	Cut Strategy	Zigzag		Ŧ	
🖬 Limiting	Relief Amount	0.2			
Dath Setting	Stock Height	0			
Fail Setting	Dwell Time (s)	0.			
📥 Lead In	Bough Clearance	1			
🚍 Display	Safe Distance	2			
	Output Type	Custom	1	-	
	Station Point		-		
	▼ Finish Cut				
	Finish Groove	Yes		-	
	Finish Thick	0			
	Retract Position	0			
	Overlap	0.1			
	Back Off Distance	1			•
Reset	Calculate		OK	Cance	

4.10 ★Support the G68.2 Functionality

In ZW3D 2021, we support the G68.2 functionality; user can call corresponding function in ZW_FANUC_5X.znc, ZW_SINUMERIK_5X.znc and ZW_HEIDENHANIN_5X.znc files.

4.11 New Limit Stepover Option in QM Roughing Operations

In QM Roughing Operations, we added a Limit Stepover option to restrict whether user can set big stepover. The default setting of Limit Stepover is "Yes", which means that the user cannot set the stepover more than 90% tool flat diameter. If set to "No", that user can set big stepover.



💱 Rough Lace 1			\bigtriangledown X
Type: Rough Lace	Advanced Setting		
Frame	Safe RAPID Traversal	No	-
Seatures	Limit Stepover	Yes	-
Tool and SpeedFeed	▼ Enable Arcs		
🗸 📠 Limiting		7	V7
💜 Boundaries			XZ
Reference Tool	Link A	-C	
Check	Radius Range	0.1	10000
I Filters			
Tolerance and Steps	+ Analysis Accuracy		
> 🔷 Path Setting	User Value		
✓ ≚ Link and Lead	Fine 0.00 🗘		Coarse
🐴 Link	Reduce Surface Mesh	No	•
📥 Lead In	Reduce Surface Mesh	110	
_ 🚣 Lead Out			
Tisplay			
✓			
🙊 Path Pattern			
Feed Control			
King More Setting			
Reset	Batch Calculate Calcul	late OK	Cancel
👳 📝 🖞	1 😺 🖡		6

4.12 New Stepdown Type and Stepdown items in Spreadsheet

In the Spreadsheet Interface – Operation View, we added Stepdown Type and Stepdown these two items to show stepdown parameter in spreadsheet. User needs to configure them in the Item Config.

💱 Spreadsheet Inter	face - Operatio	on View															∇	23
Operation	Туре	Class	Tool	То	D reg	H reg	Tool Min	Frame	Toler	Step	Step	Stepdown Type	Stepdown	Surface T	Z Sur	Speed	Feed	
Profilecut 4	Profilecut	Finish	D10R0	1	1	1	unavailable	Frame	0.005	% Too	70	Uniform Depth	1	0.15	0.15	4000	2500	=
Profilecut 1	Profilecut	Finish	D10R0	1	1	1	unavailable	Frame	0.005	% Too	70	Uniform Depth	1	0.15	0.15	4000	2500	
Profilecut 5	Profilecut	Finish	D10R0	1	1	1	unavailable	Frame	0.005	% Тоо	70	Uniform Depth	1	0.15	0.15	4000	2500	
Spiralcut 1	Spiralcut	Finish	D10R0	1	1	1	unavailable	Frame	0.01	% Тоо	70	Uniform Depth	6	0	-0.1	4000	1000	
Profilecut 3	Profilecut	Finish	D10R0	1	1	1	unavailable	Frame	0.005	Absol	0.08	Uniform Depth	1	0	0	4000	600	
Profilecut 6	Profilecut	Finish	D10R0	1	1	1	unavailable	Frame	0.005	Absol	6	Uniform Depth	1	D	0	4000	600	
Profilecut 2	Profilecut	Finish	D10R0	1	1	1	unavailable	Frame	0.005	% Тоо	70	Uniform Depth	1	D	0	4000	600	
Rough SmoothFlo	Rough Sm	Rough	D10R0	1	1	1	Not calcul		0.05	% Тоо	45.0	Absolute	0.5	0.5	0.2	2400	2500	-
4								I				-		•				•
Feature View	Reset Va	alues	Ca	culate	All	ltem	Config	Export	Spreadsh	eet Imp	oort Spre	eadsheet Tactio	Interface	Can	cel		Save All	



4.13 Distinguish the Type of Input Value for the Speed Feed Table

In ZW3D 2021, user can select the type of input value in the speed feed table. It can be selected to Percent or Numeric in Speeds table, and selected Percent, Rapid or Numeric in Feed rates table.

Name						Load Spo
Speeds			Feed rates			
Units	RPM	-	Units	MMPM	•	
Speeds	1000.0		Feeds	250.0		
Rapid	Percent 🔹	100.0	Rapid	Rapid		
Step-over	Numeric 🔻	100.0	Step-over	Percent 🔹	100.0	
Plunge	Percent 🔹	100.0	Plunge	Rapid	-	
Engage	Percent 🔹	100.0	Engage	Numeric 💌	60.0	
Retract	Percent 🔹	100.0	Retract	Percent 🔹	300.0	
Traversal	Percent 🔹	100.0	Traversal	Percent 🔹	100.0	
Slotcut	Percent 🔹	100.0	Slotcut	Percent 🔹	40.0	
Slowdown	Percent -	100.0	Slowdown	Percent -	60.0	\rightarrow