User Manual CADbro Version 2024

March 2023

Copyright (c) 2023 by ZWSOFT CO., LTD.(Guangzhou)



1	Syst	em requirements	1		
	1.1	CADbro System Requirements	1		
	1.2	Network License Manager System Requirements	2		
2	Soft	Softkey activate3			
	2.1	Stand-alone Version	3		
		2.1.1 Activation	3		
		2.1.2 Returning a License	7		
3	Basi	c knowledge	12		
	3.1	Start CADbro	12		
	3.2	Interface introduction	14		
	3.3	Open a new file	15		
	3.4	File Browser	15		
4	Man	ager	17		
	4.1	Part Browser	17		
	4.2	Assembly manager	17		
	4.3	View manager	18		
	4.4	Visual manager	19		
	4.5	Export manager	19		
5	Con	vert files	20		
6	Viev	View			
	6.1	View Angle	24		
	6.2	View	25		
	6.3	Visibility	25		
	6.4	Display Mode	25		
	6.5	Screen	26		
	6.6	Windows	26		
	6.7	Rotation	26		
7	Sele	ction	27		
8	Tool	S	28		
	8.1	Dimension	28		
	8.2	Curve creating	28		
	8.3	Attribute	29		
	8.4	Section	31		
	8.5	Capture	33		
	8.6	Assist	34		
	8.7	Mode	35		
9	Ana	уzе	36		
	9.1	Measure	36		
	9.2	Inspect Entity	38		
	9.3	Inspect Model	40		
10	ΡΜΙ		41		
10	10.1	Manager	41		

	10.2	Dimension	
	10.3	Annotation	
	10.4	Text	
	10.5	Tools	
11	Healir	ng	44
	11.1	Checking	44
	11.2	Healing	
12	Edit		47
	12.1	Direct Edit	
	12.2	Shape transform	50
	12.3	Part transform	50
13	Mark	(up	51
14	Pro		55
	14.1	Assembly	55
	14.2	Mold	68
15	HelpD	Desk	71
16	2D Dr	rawing	73
17	Cloud	d service	80
	17.1	Create a cloud service account	80
	17.2	Personal space management	80
	17.3	Team space management	
	17	7.3.1 Create a team	
	17	7.3.2 Invite members to a team	
	17	7.3.3 Create a project	83
	17	7.3.4 Add members to a project	83
	17	7.3.5 User roles	
	17	7.3.6 File management	
	17.4	Collaboration	
	17	7.4.1 Login CADbro	
	17	7.4.2 Upload models	
	17	7.4.3 Start collaboration	88
	17	7.4.4 Operations in collaboration	89
	17	7.4.5 Playback	
	17	7.4.6 View 3D models in browser	
18	Filter	·	92
19	Interf	face customization	93
20	Chang	ging language	96
21	Online	ne update	97
22	Set Fi	ile associations	98

1 System requirements

1.1 CADbro System Requirements

Operating System	CADbro Microsoft® Windows 10 Recommended system: Windows 10 Version 1909 or Windows 10 Version 20H2 Tip: The 32-bit Windows operating system is not supported by CADbro since version CADbro 2023.
Processor	Intel Pentium [®] 4(@2GHz or above), or equivalent AMD [®] processor , or equivalent AMD [®] processor
GPU	Support Microsoft DirectX [®] 9 and above or OpenGL 3.1 and above graphics cards Recommended configuration: NVIDIA GeForce GTX1050 Ti or NVIDIA GeFore GTX 1650
RAM	Minimum configuration: 2G Recommended configuration: 8G or above
Virtual memory	Minimum configuration: 1G Recommended configuration: 2G or above
GB	Minimum configuration: 4G free disc space Recommended configuration: 8G or above free disc space
Video	Minimum configuration: 1024 x 768 VGA with True Color Recommended configuration: 1280×1024,1680×1050,1920×1080 VGA with True Color or above
Pointing device	Three-button mouse, trackall or other device.
Required for licensing	Soft Key
Required for Floating License	
CD-ROM	Any speed (for installation only)



1.2 Network License Manager System Requirements

	Minimum System Requirements:
	Microsoft [®] Windows Server 2008R2 and above
	Microsoft® Windows 7
	Microsoft® Windows 8
	Microsoft® Windows 10
	Microsoft® Windows 11
Operating System	Recommended System Requirements:
	Microsoft® Windows Server 2012
	Microsoft® Windows Server 2016
	Microsoft® Windows 7
	Microsoft® Windows 10
	Microsoft® Windows 11
	Minimum configuration: Intel® Pentium™ 4 1.5 GHz or equivalent AMD® processor
Processor	Recommended configuration: Intel® Core™2 Duo or AMD® Athlon™ X2 CPU or higher
	Minimum configuration: 1G
RAM	Recommended configuration: 2G or above
	Minimum configuration: 128M
Display card	Recommended configuration: 1 GB or more
	Minimum configuration: 2 GB free space in OS disk, 1 GB free space in installation disk
Hard disk	Recommended configuration: 4 GB free space in OS disk, 2 GB free space in installation disk
	Minimum configuration: 1024*768 VGA with True Color
Resolution	Recommended configuration: 1440*900 with NVIDIA 8 series or AMD Radeon HD series or higher
Pointing device	Mouse, trackball, or other devices
DVD-ROM	Any speed (for installation only)



2 Softkey activate

2.1 Stand-alone Version

2.1.1 Activation

If you wish to operate CADbro with full functions, it must be activated. Activation needs a soft key, if you do not have a Soft Key, please contact the local dealer from whom you purchased CADbro.

Please follow these steps to activate CADbro online:

1. On the desktop, double-click the **CADbro 20XX** icon.



2. You can see the activate interface.

ᄊ CADbro License Manager			 \times
Detailed	Information of License Status		
	No authorized product detected, click "Activate License" button below to activate.		
All licens	5es (0)		
Activate	License		
		ок	

3. Select *Offline Activation*, Enter the Activation ID (Soft Key) in the text box and Click the *Active* button. The activation will be successful after a few seconds.

∢∧> Us	ser Manual
------------------	------------

< License Manager	? - ×
	Online Activation Soft Key
ි Home	Activation ID:
Standalone	
Online Activation	
☆ Offline Activation	Reinstalling system or changing hardware may cause license lost. Please return the license before such operation, and reactivate it afterwards.
Network	
◎ Request a Floating License	
Borrow a License	
	Activate

4. Click *Done* to close the successful activation message, and back to the license manager.

< License Manager	? — ×	
	Online Activation Soft Key	
l Home		
Standalone		
Online Activation		
☞ Offline Activation		
	Thank you!	
	CADbro 2024 has been successfully	
Network	activated.	
🖒 Borrow a License		
	Done	

5. Click *Ok* to close the license manager dialog box, and *CADbro* will be restarted to get the license.



൜ CADbro License Manager	? - ×
Detailed Information of License Status Product: CADbro 2024 Activation ID:PVNX-J8VM-JX77-ZML6-HNUU-****	Validity:2023-11-15 to 2024-11-10
Fulfillment ID:FID_2ce9154a_18bcb69bd655d6a Type: Standalone Tech: Soft Key Status: Normal All licenses (2)	Major Modules: CADBRO_BASE_MODULES Minor Modules: CADBRO_PRE_MODULES CADBRO_ADV_MODULES

In the case of no Internet connection on the local computer, you can activate CADbro through procedures illustrated as follows:

1. On the desktop, double-click the **CADbro 20XX** icon.



2. On *License Manager* home page, click *Activate License* button and switch to *Activation Page*, Select *Offline Activation*, and enter "Activation ID", and click *Generate* button, and specify save path in prompt Save As dialog.



< License Manager	? - ×		
	Offline Activation Soft Key		
l Home	 Generate request file Enter Activation ID, and click [Generate] to get request file. 		
Standalone	V9DG S43U D7E9 Generate		
Online Activation	 Get response file 		
Comparison Comparison Login ZWSOFT License Management Platform(http://support.zwcad.n Softline Activation flexnet/operationsportal/) by Activation ID, and generate activation refiles by activation request file. Refer to <u>Help document</u> for more detail			
	 Activate Click [Browse] to select the response file. 		
Network	Select response file Browse		
◎ Request a Floating License			
සී Borrow a License	 Reinstalling system or changing hardware may cause license lost. Please return the license before such operation, and reactivate it afterwards. 		
	Activate		

3. Login ZWSoft Product self-service activation

platformhttp://support.zwcad.net:8888/flexnet/operationsportal/ to get response files.

The steps are as follows:

- a) Select Login with Activation ID, and enter the activation ID of product to be activated and select language, and click Login buttton.
- b) Select Manager License, check a product to activate, and then click Manual Activate button.
- c) Click Select files, and select the activated request files generated from Step 2, click Submit button.
- d) On the new page, click Save to File, and select the path to save the response files.
- e) After saving successfully, exit ZWSoft Product self-service activation platform.
- 4. Click *Browse* button, and select the response files generated from Step 3, click *Activate*

^>	User	Manual
--------------	------	--------

< License Manager	? - ×
	Offline Activation Soft Key
 Home	• Generate request file Enter Activation ID, and click [Generate] to get request file.
Standalone	Y9DG S43U D7E9 2TF7 2KTJ SKH6 Generate
Online Activation	Get response file
☆ Offline Activation	Login ZWSOFT License Management Platform <u>(http://support.zwcad.net/8888/</u> flexnet/operationsportal/) by Activation ID, and generate activation response files by activation request file. Refer to <u>Help document</u> for more details.
	Activate Click [Browse] to select the response file.
Network	C:/Users/Administrator/Desktop/responseXML.xml
◎ Request a Floating License	<u>س</u>
u Borrow a License	 Reinstalling system or changing hardware may cause license lost. Please return the license before such operation, and reactivate it afterwards.
	Activate

 If activated successfully, prompt tip "You have successfully activated" and *License Manager* home page displays the activated licensing product information. If failed to activate, please check license number according to feedback tip or contact ZWSoft client service.

2.1.2 Returning a License

Please follow these steps to return CADbro License online:

- 1. Launch License Manager as follows
 - a. On the Windows taskbar, click the **Start** button.
 - b. Choose All Programs (or Programs). (Operation system is win7 or win 8 need to choose ZWSOFT folder first).
 - c. Choose CADbro20XX and then click the License Manager.
- 2. Select the license to return, click Return.

User Manual

Chapter 2 Softkey activate



3. Choose Online Return and Click Return.

< License Manager		? — X
	Online Return	
合 Home	Activation ID:	
Return License	PVNX J8VM JX77 Z	ZML6 HNUU ****
le Online Return		
☞ Offline Return	License can be activated again af	fter successfully returned.
		Return
		Return

4. The license will be successfully returned in a few seconds.

</>
</>
</>
</>
</>
</>





Please follow these steps to return CADbro License offline:

- 1. Launch License Manager as follows
 - a. On the Windows taskbar, click the **Start** button.
 - b. Choose **All Programs** (or **Programs**). (Operation system is win7 or win 8 need to choose **ZWSOFT** folder first).
 - c. Choose CADbro20XX, and then click License Manager.



- 2. Select the license to return, click Return.
- 3. Select *Offline Return*, click *Generate* button in the step of requesting to generate files. Specify the save path in the Save As dialog box that pops up..

< License Manager	? - ×
	Offline Return
l Home	 Generate request file Click [Generate] to get request file. Fulfillment ID:
Return License	FID_6c66dc4c_1844106bbc2_2683 Generate
@ Online Return	Cot response file
☞ Offline Return	Login ZWSOFT License Management Platform <u>(http://support.zwcad.net:8888/</u> <u>flexnet/operationsportal/)</u> by Activation ID, and generate return response files by return request file.Refer to <u>Help document</u> for more details.
	Image: Select response file Browse Browse
	License can be activated again after successfully returned.
	Return

- Logon ZWSoft product self-service platform http://support.zwcad.net:8888/flexnet/operationsportal/ to get the response files. The steps are as follows:
 - a. Select Login with Activation ID, and enter the activation ID of product to be activated and select language, and click Login buttton.
 - b. Select Manager License, check a product to return, and then click Manual Repair button.
 - c. Click Select files, and select the returning request files generated from Step 2, click Submit button.
 - d. On the new page, click Save to File, and select the path to save the response files.
 - e. After saving successfully, exit ZWSoft Product self-service activation platform.
- 5. Click *Browse* button, select the response files generated in Step 4, and click *Return*.



6. If returned successfully, prompt tip "Return successfully", and the licensed product information does not display on *License Manager* page.

3 Basic knowledge

Jser Manual

3.1 Start CADbro

Click the CADbro icon from the desktop



Or Start-> Program->CADbro

The interface below when you first start CADbro





3.2 Interface introduction

- 1. Function area
- 2. File list
- 3. Quick toolbar
- 4. Manager area
- 5. Filter toolbar
- 6. File Browser
- 7. ViewCube





3.3 Open a new file

🕶 Open						₽ XX
n n 🔻 🛉	C: > Users > Administra	tor 🕨 Documents	▶ CADbro ▶	\sim	0	Search 🔚 🐏
My Document	Name	Туре	Date Modified	Size		
₩ Working Folder ■ Desktop ■ My Computer > ₹ Network	 cache draw2 0-FlexibleAssembly-Tracks.Z. 002 52-C3 (1).Z3D 002 52-C3.Z3 002 52-C3.Z3D 	File Folder File Folder Z3D File Z3D File Z3 File Z3D File	2021/5/15 16:10 2020/6/19 15:38 2022/6/9 14:42 2020/7/28 11:19 2021/3/12 9:29 2020/7/20 11:17		5.1 ME 343 KE 291 KE 343 KE	≣ 8 8 3 8 8
Local co	ору				(Quick filter: 🤳 🥸 🚳 🐻 👿
File name:				Search	~	CADbro / Z3 File (*.23p; *.23; *.1 *) CADbro / Z3 File (*.23p; *.23; *.VX; *.Z3PRT; *.Z3DRW; *.Z3ASIM) CADbro File (*.23D) Part (:.23PRT) Drawing (*.Z3PRT) Drawing (*.Z3PRT) DWG File (*.dwg) DXF File (*.dwg) DXF File (*.dwg) DXF File (*.dwg) IGES File (*.dwg) Parasolid File (*.xmt, bin; *.x_t*.x_b) STEP File (*.stp; *.step) STL File (*.stp; *.step) STL File (*.stp; *.step) STL File (*.stp; *.step) STL File (*.stg, *.stg; *.sst; *.sasb) CATIA V4 Files (*.model; *.exp; *.session) CATIA V4 Files (*.model; *.exp; *.session) CATIA V4 Files (*.nodel; *.exp; *.session) CATIA V4 Files (*.nodel; *.exp; *.session) CATIA V4 Files (*.stp; *.asmt; *.asm*) Solidd V5 File (*.sldprt; *.sldsasm) Graphic format File (*.stp; *.sldsm) STEP242 Compress File (*.stpz) JT File (*.jtp) CATIA V5 Drawing File (*.CATDrawing) SolidWorks Drawing File (*.cATDrawing) SolidWorks Drawing File (*.cATDrawing)

3.4 File Browser

Using the File Browser can quickly locate the directory and add your frequently-used folder to as Favorites.

List windows can list different file formats and quick search the specified file by name.

1-> Favorites and Recent files

2-> Disk directory

3-> File list windows







4 Manager

4.1 Part Browser

Part Browser used to list all the entities belong to the part, like solid, surface, wireframe and so on, you can uncheck the checkbox to hide the entity or right-click the folder to erase, hide or show all.





4.2 Assembly manager

Assembly manager used to manage the whole assembled structure, all the subassembly, components can be list here, then right-click on the specified component, more functions will be support.



Manager	🗉 🔀 🕞 Free_M
The show All	🍜 🗲 🔮 🎸 🖉
Show All Show A	 Suppress Blank Show Only Zoom To Pack Rename Part Cut Copy Paste
	 Face Attributes Toggle Entity Transparency Entity Info Activate Layer Layer On/Off Display Parent Configurations

4.3 View manager

View manager used to manage all the view angle, section view, custom view and also the PMI.





4.4 Visual manager

Visual manager used to customize the display status, like open the shadow, switch on/off the coordinate or datum plane, setting up the open edge on/off, even the color or thickness.



4.5 Export manager

Export manager support to predefine your exporting configuration, like exporting the file format to a specified folder, it can be saved as configuration, after that, you can put the current 3d model from graphic window to run this task.

Mana	iger				o 23
\$ _	Config				
Fo	D		/	\$	4
.	ExportTo	_IGS_Conf	ig_0		
	ExportTo	_STP_Cont	ig_\		
	ExportTo	CATPART	_Config_2		
~					
<u>}</u>					
	Running T	ask End	Task		
				20%	×
	🚽 🖗 Free		ise_& (Expo	rtTo_C	A
	🛛 🔏 Free	e_Machine_V	ise_& (Expo	rtTo_S	TP
	👗 Free	_Machine_V	ise_& (Expo	rtTo_l(5S



5 Convert files



Open: Open all the supported files in CADbro, including the Z3D file.

Import: Import all other files into current Z3D. It will not create a new Z3D file.

Quick Import: Quick import supports importing a new Z3D file and only showing its display data, which is faster than the normal type of import. However, there is no Nurbs data, which means that editing this kinds of imported model is limited.

~ 🔏	🐧 AS	SY	camshaft
	🖌 🧊	9	–)LDS-S M6
	🖌 🇊	٩	 DIN912-M6x10-12.9
	🖌 🇊	٩	–)LDS-S M6
	🖌 🧊	٩	 DIN912-M6x10-12.9
	🖌 🧊	9	–)LDS-S M6
	🖌 🧊	٩	 DIN912-M6x10-12.9
	🖌 🇊	٩	 DIN912-M6x10-12.9
	🖌 🇊	٩	–)LDS-S M6
	🖌 🧊	9	-)E030501B
	🗸 🇊		-)E031001B
		_	

If you want to load the Nurbs data, just click "Load current model" or "Load all model" in the right-click menu of the components or the top-assembly.

Mana	ager		= 23			Mana	ager	- 23
☞_	Show All		T			\$	Show All 🔹	Y
ΗD	✓ ▲ ASSY camshaft					FO	🗸 💒 🎙 ASSY camshaft	
	🗹 🇊 🎙 (–)LDS-S M6	4	Face Attributes			-	🔽 🇊 🎙 (–)LDS-S M6	
.	🗹 🧊 🎙 (–)DIN912-M6x10-12.9	1	Toggle Entity Transparen	ncv		÷.	🗹 🇊 🎙 (–)DIN912-M6x10-12.9	
	🗹 🧊 🎙 (–)LDS-S M6	9	Entity Info	,			🔽 🌍 🎙 (–)LDS-S M6	
	🗹 🧊 🎙 (–)DIN912-M6x10-12.9	R	Entity Into				🗹 🇊 🎙 (–)DIN912-M6x10-12.9	
	🗹 🧊 🎙 (–)LDS-S M6		Activate Layer			9	🗹 🌍 🎙 (–)LDS-S M6	
	🗹 🧊 🎙 (–)DIN912-M6x10-12.9	1	Layer On/Off				🗹 🌍 🎙 (–)DIN912-M6x10-12.9	
	🗹 🧊 🎙 (–)DIN912-M6x10-12.9	2	Load current model		→		🗹 🌍 🎙 (–)DIN912-M6x10-12.9	
	🗹 🧊 🎙 (–)LDS-S M6 🛛 🚽	2	Load current model				🗹 🌍 🎙 (–)LDS-S M6	
0	🔽 🧊 🎙 (–)E030501B		Display Parent	•			🔽 🌍 🎙 (–)E030501B	
1	🔽 🧊 🎙 (–)E031001B		Configurations	+		_	🗹 🌍 (–)E031001B	

Chapter 5 Convert files



Mana	iger		Mana	ger	- 23
\$	Show All	Y	\$	Show All	Y
	🗸 🌋 🖣 ASSY camshaft		Fo	ASSY camshaft	
	🔽 🍞 🎙 (–)LDS-S 🧐 Edit Part			🗹 🧊 (–)LDS-S M6	
÷.	🗹 🇊 🎙 (-)DIN91 👝 Inquire		- B	🗹 🇊 (–)DIN912-M6x10-12.9	
	🗹 🇊 🎙 (–)LDS-S 💆 Inquire	-	L	🗹 🍞 (–)LDS-S M6	
	🗹 🇊 🎙 (–)DIN91 🚰 Load all models			🗹 🇊 (–)DIN912-M6x10-12.9	
\square	🗹 🧊 🎙 (–)LDS-S Мб	_		🗹 🇊 (–)LDS-S M6	
	🗹 🧊 🎙 (–)DIN912-M6x10-12.9			🗹 🧊 (–)DIN912-M6x10-12.9	
	🗹 🇊 🎙 (–)DIN912-M6x10-12.9			🗹 🇊 (–)DIN912-M6x10-12.9	
	🗹 🧊 🎙 (–)LDS-S M6			🗹 🇊 (–)LDS-S M6	
\sim	🗹 🧊 🎙 (–)E030501B			🗹 🇊 (–)E030501B	
1	🗹 🇊 🎙 (–)E031001B			🗹 🌍 (–)E031001B	

Multi-Import:

Multi-Import the third-party format files (ACIS, Catia V4, Catia V5, UG/NX, Proe/Creo, Inventor, SolidEdge and Solidworks) and autosave those files to .Z3D files.

🗠 Multi-Import	Ģ	23
Input format	All Files(*.*)	•
Import setting	All Files(*.*) AutoCAD(*.dwg:*.dxf)	
Include files from sub	IGES(*.iges;*.igs) STEP(*.step:*.stp)	
Write into input folde	r Parasolid(*.xmt_txt;*.xmt_bin;*.x_t;*.x_b) VDA(*.vda)	
Output file C:\Users\A	STL(*.stl) Vrml(*.wrl)	
File name	ACIS(*.sat;*.sab;*.asat;*.asab)	
	Catia V4(.ntode; .exp; .session) Catia V5(.CATPart, CATProduct) Inventor(*.ipt;*.iam) ProE/Cero(*.prt;*.pt.;*.asm;*.asm.*) Solid&Orks(*.sldptt;*.sldasm) UG/NX(*.prt) Graphic format File(*.cgr;*.3dxml) STEP242 Compress File(*.stpz) Rhino(*.3dm) Catia V5 Drawing(*.CATDrawing) SolidWorks Drawing(*.slddrw)	
Empty list	Run Add files Add folde	r

Save: Direct save the current Z3D file.

Save as: Except for saving the current Z3D file, you can save the whole file to any other files.



Export: Compare to Save As, Export command support to select specify shape or component



from the current file to export. Even can make a filter to include or exclude by Layer, Geometry type and Attribute.

V IGES File Generation				Ģ	X
IGES Filter					
▼ Select					
O All Objects					
O Select From Screen			Pick Objects		
▼ Exclude layer					٦
Layer0000					I
▼ Exclude Geometry					
Curve Su	urface	Solid	Plane		
X-hatch Di	imension	Text	✓ Blanked		
▼ Exclude Attribute					

Multi-Export: Using the default exporting setting to export files to multiple file formats at one time. Users can select the files and objects to export on the **Select Object** page.

Sele	ect File	es		
	•			
6			Na	me
8		:\Users\Admir	nistrator\Docu	iments\CADbro\002 S2-C3
ele	ect Ob	jects		
ele Filt	ect Ob	jects Show All		•
Sele Filt	ect Ob er	jects Show All Name	Туре	File
Sele Filt	er Ø	Show All Name S2-C3	Type Part	 File C:\Users\Administrator\
Sele Filt 1	er Ø	Show All Name S2-C3 S2-C3_2D	Type Part Drawing	 File C:\Users\Administrator\ C:\Users\Administrator\
Sele Filt 1 2	ect Ob er Ø	Show All Name S2-C3 S2-C3_2D	Type Part Drawing	 File C:\Users\Administrator\ C:\Users\Administrator\
Sele Filt 1 2	er Ø	Show All Name S2-C3 S2-C3_2D	Type Part Drawing	 File C:\Users\Administrator\ C:\Users\Administrator\
Sele Filt 1 2	ect Ob er V	Show All Name S2-C3 S2-C3_2D	Type Part Drawing	 File C:\Users\Administrator\ C:\Users\Administrator\
Sele Filt 1 2	er	Show All Name S2-C3 S2-C3_2D	Type Part Drawing	 File C:\Users\Administrator\ C:\Users\Administrator\
Sele Filt 1 2	ect Ob er V	Show All Name S2-C3 S2-C3_2D	Type Part Drawing	 File C:\Users\Administrator\ C:\Users\Administrator\
Sele Filt 1 2	ect Ob er Ø	Show All Name S2-C3 S2-C3_2D	Type Part Drawing	▼ File C:\Users\Administrator\ C:\Users\Administrator\

On the **Export Settings** page, you can specify the destination and select formats to export.

Chapter 5 Convert files



				₽ 33	
Select Object	Export Settings				
-Destination -					
Folder C:\U	sers\Administrator\Docur	ments\CADb	ro		
Re	eplace if file exists 🛛 🔽	Export all sh	neets		
Formats					
Parasolid	Text (*.x_t)		Set options		
🔲 IGES (*.ig	s)		Set options		
STEP (*.st	p)		Set options		
STL (*.stl)			Set options		
ACIS (*.sa	et)		Set options		
PDF (*.pd	if)		Set options		
VDA (*.vd	la)		Set options		
🔲 VRML (*.)	wrl)		Set options		
DWG / D	Set options				
TIFF Image (*.tif)					
🔲 Neutral (*	'.vxn)				
🔲 Neutral (*	'.z3n)				
Parasolid	Binary (*.x_b)		Set options		
CATIA V4	(*.model)		Set options		
CATIA V5	Part (*.CATPart)		Set options		
CATIA V5	CATIA V5 Assembly (*.CATProduct)				
🔲 JT (*.jt)			Set options		
🔲 Bitmap (*	.bmp)				
GIF Image	e (*.gif)				
JPEG Ima	ge (*.jpg)				
PNG Image	ge (*.png)				
		ОК	Ca	ncel	



6 View

6.1 View Angle

Click different View icon to change the view angle

									٢
Isometric	Тор	Bottom	Left	Right	Back	Front	Auxiliary	Align with Plane	Align with Direction
View Angle									

Align with Plane:

Click this command then select one plane, the view will change to auto-align this plane

Align with Direction:

Define the origin point, Up direction and Right direction to customize the view angle

▼ Required				
Point	×	• 🛓		
Up	*	• 🛓		
Right	*	• 🛓		

How to save the View angle and reuse it later?

Click the **View manager**, then right-click the **Custom View -> New**, then you can save current view angle





6.2 <u>View</u>



6.3 Visibility



6.4 Display Mode



Analyze: Display the previous face analyze status

Combination: This display mode is used for assemblies when you want to display components in different display modes. Like the assembly model below, each component can show in different display modes.



Note: To set the different components to different display modes, you can right-click the component >Display>then select your needed display mode.







6.5 <u>Screen</u>

Display a multi-window screen, each window size can be dragged to change the size.



6.6 Windows

The list method for all open files.



6.7 Rotation

Rotate the 3D model around the X/Y/Z axis. Users can also difine a specific axis by **Lock any axis** for rotation





7 Selection



Pick control: Control the input field, picking or unpicking. You can also call the pick control by a right mouse button click on the graphic window.



Pick Rule: Control the picking behavior

>Normal Pick is the default picking behavior, which means when you Left mouse click the entity, it will auto be selected but the previous picking entity will be removed, you can hold Ctrl key to achieve Add picking or unpicking.

>Add Pick means always add the selected entity to the input field by clicking the left mouse button.

>Del Pick means always remove the selected entity by clicking the left mouse button.

Pick Scope: The window pick behavior, for example, it means all the entity in the windows frame will be selected.

Pick from list: When there are multiple legitimate objects under the current mouse position, you can switch different object choices through "Pick from list" option from right-click menu. Or use the shortcut keys Alt (cannot be released) + right clicking (continuously): traverse and prehighlight (pre-selection) the legal objects under the current mouse location and left click the mouse when pre-highlight some entity to confirm the current selection.



8 Tools

8.1 Dimension

New style dimensions can be created by the following tools, which are always parallel to the view window.



The final effect is shown below.



8.2 Curve creating

CADbro offers curve creating tools to create the wireframe that includes the Line, Polyline, Circle, Rectangle and so on.





8.3 <u>Attribute</u>



Line attribute: Use to change the color, style, and width of next creating curve

Face attribute: Use to change the face color, or you can select face set or shape body first then click this command

For face attribute, the most often used is the Face color and Transparency

🖗 Face Attribute	s		⊽ ⊠
Face			
Back			
Ambient	20	¢]
Diffuse	90	÷]
Reflectance	0	÷] 🛛 ———————————————————————————————————
Transparency	0	÷] []
Refraction	1.15	÷	-0
Specular			_
Reflection	25	÷	
Exponent	64	÷] ——[] ——[]
Color			
🥩 C	Ж		Cancel

You can also find the command from the Quick toolbar as below.



Material attribute: Use to change the shape or part materials, which allow you to select the existed material from the list or add your new material.

Chapter 8 Tools



Require						
Туре	Shape Material	-				
Shapes	1 picked	\approx				
Name	Name					
HDPE-High I	Density Polyethylene					
Density (kg/m	13)					
950						
-File/Bundle-						
Bundles.Z3	•	<u> </u>				
HDPE-High [Density Polyethyle 🔹	X				
Copper Glass Gold						
Graphite HDPE-High I	Graphite HDPE-High Density Polyethylene					
Iron-pig		_				
Iron-pure	+					
LDPE-Low D	ensity Polyethylene	-				

Part attribute: Use this form to define attributes assigned to the active part. Use this dialog box to define properties for active parts, supporting standard attributes, user-defined attributes, and physical attributes.

For Import/Export/Edit in Excel.

Name	00 Assieme		
Number		Designer	
Manager		Supplier	
Cost		Class	
(eywords	Each keyword should be separa	ted by a "," comma	
Description			
Description Create time	9/14/2007		Short date 🔻
Description Create time .ast modified	9/14/2007 11/20/2019		Short date +
Description Create time .ast modified iheet object	9/14/2007 11/20/2019	Sheet co	Short date +



8.4 Section



Planar Section:

Use this command to dynamically section the active part or assembly. Completing this command will create section curves in the active part and the active component in an assembly. It can also be used as a visual aid to dynamically clip away portions of the part to view features that might normally be hidden from view.

The required inputs include the section plane and offset distance. Optional inputs include multiple sections (copies at distance intervals) and using the section plane as the first copy.

Section:

Required

This command allows you to set specific characteristics of the section view that is enabled when Section View Display Mode is activated.

▼ Required						
View name	section1					
	Insert view					
▼ Section plane						
Align plane	<u>⊸</u>					
▼ Section Plane Placement						
Offset	5.9 mm 🗘 🍜 👻					
Thickness	11.8 mm 🗘 🕸 🔹					
	Reset position					
	Reverse direction					



Display control

Sectioned shape - support two buttons, one is the default option to hide the sectioned part, another button to show the sectioned part with a wireframe.

Interference - support to specify the interference color.

Hatch color - allow to use part color or specify a color for the hatch.

Hatch style - support three buttons here, the first one is the switch off, the second and third one is the different hatch style.

Spacing -is offset value of the hatch style

Opaque -can drag the slider to change the transparency percent of the hatch

Align with section plane- can align the current model to the section plane

Display Interference -Switch on/off the interference checking for the section view

Display section planes -Switch of/off the section plane

Display hatch for non-closed shapes - Whether creating the hatch for the open surface, that's not perfect solid with a gap or missing face

Move handle only - Only moving the dynamic moving handle when dragging, that will not move the section plane position.





Section Curve

CADbro supports to display the section curves with extra window, working on the extra window, you can rotate the section profile even export it to DWG/DXF/IGES and STEPS.

<u>Insert</u>

CADbro supports to insert datum and curves into the current model.

Component

CADbro supports to exclude and include some specified components on the section curves window.

▼ Section Curve		· · · · · · · · · · · · · · · · · · ·	
Display section curves		Export	
Curve color	=	🔯 Zoom All	
▼ Insert	1	💽 Rotate Right 90 Degrees	
Incast datum		🤣 Rotate Left 90 Degrees	
insert datum insert curves		😮 Rotate 180 Degrees 🛛 🔟	
▼ Component			()
Exclude *			
Component ×		107.008 mm	

8.5 Capture



Capture:

Use this command to capture the active display to a file. It automatically turns off the view extents readout, the world axis triad, and the default datum display. You can save to bmp, gif, jpg, and tiff graphics file formats. It also allows up to the maximum pixel resolution supported by your graphics card

Screenshot:

This command will use window pick to select the area to capture, which can be stored at the clipboard so that you can paste to any other application like Word or Excel, what's more, you can save to file or choose a different background color.
Chapter 8 Tools



✓ Screensł	not	\triangleright		₽ 33
Save to				
Clipb	oard			
🗌 File	C:\Users\	Administrat	or\Docum	
Backgrou	ind			
O Curre	nt			
O Custo	mize			
Preview				
Pick Area	а	ОК	Ca	ncel

8.6 <u>Assist</u>



Datum Axis

Use this command to insert a new datum axis. The attributes of the datum axis include one orientation, an origin and the length.

Datum

Use this command to insert a new datum plane, which can be used to create a 3D curve or used to be a section plane.

Datum CSYS



Use this command to insert a new CSYS, which consists of three axes, three datum planes and one origin.

Frame

Frame can be used to create a local frame, which can be used to recalculate the coordinate dimension.

Stock

Use this command to create an extruded stock feature that completely encloses a single face, an entire shape or a block. The required inputs include the shape(s) to enclose and a reference plane to orient the stock. By default, the size of the stock equals the extents of the shape.

Hole

Use this command to create a simple hole in CADbro.

8.7 <u>Mode</u>



2D Sheet

Use this command to go to 2D Sheet environment.



9 Analyze

9.1 Measure



This command can base on the selected entity to generate the measurement result, which can be also recorded and attached in the model and minimize the label.



Distance Distance:

This command measures 3D distances at the Part Level. You can measure points, geometry, and planes. Remember that you can select more input options. Refer to each method and options below.

(A) User Manual	Chapter 9 Analyze
➡ Distance E3 ✔ X ☑ ✔ Required ● ✔ Required ● ↓ X ☑ ● ↓ X ☑ ● ↓ X ☑ ● ↓ X ☑ ● ✓ Result Control ✓ ● ✓ X distance ✓ ✓ ✓ Y distance ✓ ✓ ✓ Z distance ✓ ✓ ✓ Direction Magnitude only ✓ Projection None ✓ Methods Between bounds ✓	Measure Distance S Distance 16.1201 [mm] Dist-X 10.8 [mm] Dist-Y 0.02057 [mm] Dist-Z 11.96734 [mm] OK Cancel



Angle Angle:

This command measures 3D angles at the Part Level. You can measure points, geometry, and planes. You can select from a variety of different geometry types to measure from. Using the RMB (right mouse button) provides even more options to choose from.



Arc Arc:

Use this command to select points and display arc data such as radius, included angle, center, and normal. You can use this command to check the radius of a 3D arc, curve or edge as shown in the figure or to check the theoretical arc data for three random points.



Length Length:

Use this command to inquire about the length of a line, arc, curve, or edge. You can also inquire about a length summation by selecting multiple items to measure. In the sketch, the external curves can be captured to inquire.

- 1) Select a line, arc, curve, or edge to measure. The length appears in the Current Curve field.
- Select another line, arc, curve, or edge or move. Move the cursor over additional items. The new length and total length for each is displayed in the Last Curve and Summation fields.
- 3) Click **OK** to close the form.



Area Area:

Use this command to inquire about the area of an enclosed planar 3D or 2D region. The following information is displayed in the **Inquire Area Properties Form** (shown below). Refer to the 3D and 2D procedures below.

Inquire area properties	₽ %			
Ref point (11.17818, 70, 11.17818)				
Centroid (11.17818, 0, 0)				
Area 15393.8 [mm^2]				
Perimeter 439.823 [mm]				
Density 1				
Area mass 15393.8				
Inertia about reference point				
Moment of inertia lx 9.428705e+07 [mm^4] ly 2.078089e+07 [mm^4] lxy -1.204523e+07 [mm^4] lr 1.150679e+08 [mm^4]				
Moment of inertia lx 1.885741e+07 [mm^4] ly 1.885741e+07 [mm^4] lxy 0 [mm^4] lr 3.771482e+07 [mm^4]				
To file				
ОК				



Use this command to inquire about an arbitrary 3D planar area enclosed by boundary curves. The boundary curves may consist of 3D/2D curves, partial or whole sketch and/or trace profiles, edges or curve lists. All boundary geometry must lie on the same plane.

9.2 Inspect Entity





? + Coordinate:

Use this command to display the location of a point measured in specific X, Y, Z coordinates.

🖑 Curve Info

Use this command to inquire curve information on any line, arc, curve, or edge.

NURBS data

Use this command to inquire the NURBS data of a line, arc, curve, or edge.

Control Polygon

Use this command to display the control polygon of a curve. This refers to the line segments that connect adjacent control points of a curve. The control points are the set of points that are saved by the system and used mathematically to define the curve. You can apply this command to the active part to another component in an assembly or sub-assembly from the active part.



Curvature Plot

Use this command to display the curvature plot of curves or edges. The plot is displayed as line segments projecting out perpendicular from the curves. The length of each line segment illustrates curvature at that point on the curve. This command helps to visualize overall curvature across a curve or edge.

Param Space Partition

Use this command to display the parameter space of a curve. Point markers at each point on a curve associated with a knot value are displayed along with the normalized parameter and arc length values at each point.

This helps you determine how close the curve is to arc-length parameterization. This echo relates to the existing curve. Use the Cubic Refit Curves command to echo this information for a resulting refitted curve. You can apply this command from the active part to another component in an assembly or sub-assembly.

Analyze Faces

Use this command to analyze faces for surface imperfections, draft angles, curvature, and other

surface characteristics. The analysis results are presented in a variety of plots superimposed on the active part. This command switches the Graphics Window to Analyze display mode and leaves it in that mode when the command terminates. Use the View menu or the Toolbar's display mode button to switch to another display mode when the analysis results are no longer needed.

Surface curvature

Use this command to inquire about the curvature of a face where it passes through a specified point. By default, the point lies on the face. You can use the **Face** option to select a different face to analyze. By default, a direction that is normal to the face through the specified point is used to "slice" the face for curvature analysis. You can also select a different direction if desired.



Continuity Error

Use this command to analyze the tangent continuity across two mating faces at a selected edge. The edge must be shared by both faces. The results displayed are the maximum and average error measured in degrees. You can also check for CO continuity (gaps) along the edge by selecting **Position** from the **Continuity** option.

9.3 Inspect Model





Mass Properties

Use this command to calculate and display mass properties information (e.g., volume, area, mass, etc.) for a selected shape (open or closed), component, or the entire assembly. Blanked and/or suppressed components in an assembly will be ignored during these calculations. (Surface) The area is the only value that can reasonably be relied on for an Open Shape. A warning message is prominently displayed letting you know that you have selected an Open Shape.



Part Interference Check

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



10 PMI



10.1 Manager

After creating PMI notes, you can go to the View manager as below to activate different views to create PMI, move or copy PMI from one view to another view, what's more, you can control the PMI visibility by using the manager.



10.2 Dimension

Use to create different 3d dimension types as below.





10.3 Annotation

Use to add different 3D annotations on the model.



10.4 Text

Simple text and the text with balloon



10.5 Tools

PMI Dimension

This command is used to preset all PMI attributes, including the dimension style, text style. etc.

Chapter 10 PMI

User Manual

孙 PMI Dimensio	on Attribute	es		₽ %	
General Lin	es Text	Misc			
	- > ×× ->	xxx -	 +	×x ,	
Unit/Tolerance	e Altern	1 kate Units			
	xxxx 🔻			™©n	
x.xx ^{±,01}	x.xx 👻	<mark>0.20</mark> ▼	x.	•	
1.20 <u>,xx</u>	x.xx 👻	0.20 ₩	x ^{+0.01} -0.00	•	
			x.	•	
All Off	Т	oggle		All On	
1		C	K	Cancel	



Resize PMI

When creating a PMI note, it will generate a different note sizes base on a different view, this command can reset the global PMI size to match all notes.



This command uses to do regeneration for PMI, if you want the PMI to be refresh or update, you can use this command to do that



11 Healing

11.1 Checking





Heal/Analyze Heal/Analyze

This command analyzes and attempts to repair abnormal conditions relating to the geometry of entities (e.g., vertex points, edge curves, UV curves, face boundaries - refer to the figures above) that defines the topology of the active part. Such conditions include gaps between face edges and coincident vertices.



Show Open Edges

Use this command can quick check the open edges one by one. Use the arrow button you can jump to previous or next one, use Zoom to the window will zoom to the current open edge to display.

🐼 Show Open Edg	ges 🖓 🏵
Number Open	0
Edge	
•	
Id #	
Length	0.000000
	Zoom To
Re	eset OK



Check Edge

This command checks and/or delete tiny edges whose curve lengths are smaller than a specified



tolerance and that are not required to define the topology of the active part. This command will attempt to bring the encountered conditions within the current <u>Geometry Tolerance</u> setting. Select the edges to check/delete or middle-click for all edges.



Fit Plane Fit Plane

Use this command to check and fix planar faces.

11.2 Healing



Sew Sew

Sew the open edge face to be merge one.



Explode Explode

Explode the selected face set to be multiple faces.



Gap

Sew Edge's Gap

This command attempts to sew a gap that exists between two sets of face edges. After selecting the two sets of edges, CADbro will extend the two faces to try to determine curves of intersection. If intersections are found, the faces are trimmed to the new intersection curve which then becomes the new shared edges. If multiple edges are selected for each set, it is best that they form a closed loop.

If the faces will not intersect when extended, then a medial curve located between the edges is projected onto each face and then each face is trimmed. This will not close the gap but will make the two edges more closely aligned with each other. This operation is limited to working on open faces. Therefore, you must first explore the faces and then sew them. Refer to the optional **Edge Cv** input below.

You can also select two sets of edge curves. A checkbox is provided to enable this option.





This command is designed mainly to handle imported geometry to attempt to sew gaps that exist between shared face edges. After selecting the edges, **CADbro** will extend the faces to try to determine curves of intersection. If intersections are found, the faces are trimmed to the new intersection curves which then become the new shared edges. If multiple edges are selected, it is best that they form one or more closed loops.

If the faces will not intersect when extended, then a medial curve located between the edges is projected onto each face and then each face is trimmed. This will not close the gap but will make the two edges more closely aligned with each other. This operation is limited to working on imported open faces. Therefore, do not sew the faces before using this command.



This command fills open gaps between faces. The gaps must be bordered by a closed loop of open edges. If the edges are not coplanar, a ruled face is created. If the edges are coplanar, a planar face with trim loops is created.



Reverse Direction

Use this command to reverse the normal direction of faces or shapes. Direction arrows will appear indicating the current direction of the face or shape.



12 Edit

12.1 Direct Edit



Use these commands to move face. Various methods are supported including directions, points, and frames.



Use this command to align move face by using 3D constraints. The constraints include coplanar, concentric, tangent, parallel, perpendicular, at angle and symmetric.

Motion face

Select a face to move.

• Stationary

Select a stationary object. The motion face will keep the specified constraint with this object.

• Angle

This field is available if the constraint is **At angle.** Set the angle between the motion face and the stationary object.

• Symmetry plane

This field is available if the constraint is **Symmetric**. Specify the symmetry plane between the motion face and the stationary object.





Dim Move Face

Move face by dimension, which supports linear and angular dimensions.



Use this command to offset one or more faces of a shell. This command extends or trims faces as needed during the offset operation to close gaps and resolve intersections.



This command creates a draft feature about selected faces.



Use these commands to copy face. Various methods are supported including directions, points, and frames.



Use this command to mirror the selected faces.



Use this command to pattern the selected face.





Modify Fillets

Use this command to modify the selected fillet.



Modify Radius

Use this command to modify the radius of cylinder and sphere.



Modify Holes

Use this command to modify the hole without re-generation history.



Use this command to replace one or more faces of a solid or shape with another face or shape or datum plane.



Simplify

Use this command to simplify a part by removing selected faces. The command will attempt to extend and reconnect faces to close the resulting gaps in the part. If the part cannot be closed properly an error message will be returned. Select the faces to remove and then middle-click to remove them.

Alternately, you can also **delete** or blank the features that contain the faces.





12.2 Shape transform



Shape transform commands support to move, copy mirror and scale the selected shapes.

12.3 Part transform



Part transform commands support to move, drag, mirror and rotate the selected component. Compare to shape transform, the target entity type is different. One for shapes, one for components.



13 Mark up



Create new comment

Create a new comment will list on the manager as below



Right-click the comment you will see the list below, you can change the setting, save the comment to Z3DM file, rename or erase it.





With setting command, you can add more information for the comment, even change the font style like text font, line thickness, and color.

Cha	pter	13	Pro

()	∧> User №	/lanual				
	📣 Setting			ç	23	
	Name:	Administra	itor			
	Phone number:					
	Email address:					
	Font Name:	@Arial Unicode MS 🔹				
	Font Height:	24			÷	
	Line Thickness:	þ			÷	
	Color					
	Apply changes	to:				
	O All com	nment in this	environment			
	 Current comment 					
	New comments					
		ОК	Cancel			

Apply changes to:

All comment in this environment: Will change all comments

Current comment: Only affect the current comment, this option only available when you rightclick a comment and use the setting.

New comments: Will affect the new creating comment, but not the existed comments.



Open markup

Open the saving comment file, which is Z3DM file.



Save markup

Save all comments or select one of them to save.



Text with leader

Create the label text with a leader.







Only create the text without a leader



Image

Insert the extra image.



Linear dimension

Create the linear dimension as below





Angular dimension

Create the angular dimension as below.



Chapter 13 Pro





Insert the stamp to the screen





14 Pro

14.1 Assembly

CADbro provides many tools to work with components within an assembly. You can check for interference, blank a component or inquire component properties. You can also set the regen status of each component individually and clone parts. Refer to the topics below.



Interference Check

Interference Check

Use this command to check the interference between components. Please note that the suppressed components in an assembly will be ignored during these calculations. The following steps show how to use this command.

Step 1: Select the component to be checked. If you leave it blank, the whole assembly will be checked.

Step 2: Define the check scope and other options according to the needs.

Step 3: Click "Check" button to calculate, and the results will be listed in the dialog.



Below are the explanations of the options of this command. **Scope Options**



Scope Only among the picked	Settings		
	Scope	Only among the picked	•
Include shapes Only among the picked	Include shapes	Only among the picked	
With other components	Check among	With other components	

> Only among the picked

Only check the interference among the selected components.

> With other components

Check the interference among the seleted components, and meanwhile, check the interference among the selected components and other components.



Other Options

Include shapes within the assembly

Check this option, the interference between the picked components and shapes will be checked.

Check among shapes

Check this option, the interference among all the shapes will be checked.

> Treat subassembly as a whole

Check this option, the interference between the subassemblies will not be checked.

Note: This option is only available when the selected components contain subassemblies.

Ignore hidden shapes and components

Check this option, the hidden parts and components will be ingnored.

Save interference geometry

Check this option, you can save the interference shapes in the history list.

Non-interfering components

This option is to define the display mode of the *non-associated components*, including "Hidden", "Transparent", "Shade" and "Wireframe" modes.





Clearance Check

Use this command to check the clearance among components. The following steps show how to use this command.

Step1: Select the components to be checked. If you leave it blank, the whole assembly will be checked.

Step2: Set the clearance value.

Step3: Define the scope of clearance check.

Step4: Click "Check" button to get the results.

	Clearance Check Required Components 5 picked Clearance(<) 2 mm : Check
	▶ Settings
	▼ Result
	Unrelated components Hidden
	Part name 1 Part name 2 Value (mm)
	V 031_Lamina 071_Rolam Contact
	🔲 032_Lamina 033_Lamina Interference 🗮
	✓ 032_Lamina 071_Rolam 1.244748
Ma.	Output check results

There will be 3 possible results:

- ♦ Contact: The distance between the components is 0mm.
- ♦ Interference: There is interference between the selected components.
- ♦ Distance value: The distance between the components is more than 0mm but less than the pre-set clearance value.

Below are the explanations of the options of this command.

Check clearance between components and shapes

Check this option, and the clearance between the picked components and shapes will be checked.

Check sub-assembly internal parts clearance

Check this option, and the clearance between the parts in the sub-assembly will also be checked.

Ignore hidden shapes and components

Check this option, and the hidden parts and components will be ignored.

Unrelated components

This option is the same as the "Non-interfering components" option in the "Interference Check" command.

> Output check results

Click this button to export all clearance results to an Excel sheet.







Use this command to compare two parts and show the differences between them. CADbro provides two methods for comparison, solid compare and face compare

This command support to select the Z3D file from local disk without opening in CADbro.

Solid Compare

Select the part file as below, then click the Calculate button.

🕶 Compare Parts	₽ 🛛
Solid compare	○ Face compare
Base	Compared
Connector_V2.Z3D 🔹	Connector_V1.Z3D 🔹 🔄
Connector_V2	Connector_V1
C FIEVIEW	
	Ĩ
C C	
	Calculate

Below is the compare result, which shows the different regions, you can also change the color and the transparency and hide the regions.

Chapter 15 HelpDesk



Reset: Reset all the settings for display, color, and transparency.Save file: Save the comparison result as Z3D or other neutral formats.Save report: Generate a comparison report.

User Manual



Face compare

Select the part file as below, then click the Calculate button.



Below is the calculating result, which shows the unchanged faces, changed faces and unique faces, you can also change the transparency and hide the faces.

Manager	• 23	Bottom_part_V1.Z3D - [Part_V1] ×	Compare Result.Z3D - [Assembly] 🗙 🔹 🗧 🤤
▼ Unchanged faces	6	3	• 🏘 • 🔯 •
Color			
Transparency 0 🛟			
✓ Base: 16			
Compared: 16		0	
▼ Changed faces		•	
Color		X	
Transparency 0 🗘		Base	Compared
✓ Base: 19			
Compared: 19			
▼ Unique faces			
Color			
Transparency 0 🗘		(Da	
Base: 5		Z _V	
Compared: 14		×	
	> 🔒	117.431 mm	*

Click the Save button to generate the compare report.





Exploded View

Use this command to create exploded views for each assembly configuration. This command provides a list to record every explosion step, in which you can re-order the existing steps by dragdropping the picked step.



Add Exploded Step

CADbro added "**Move exploded**" "**Rotate exploded**" and "**Radial exploded**" in "**Add Exploded Step**" command to meet user's requirement of more complex exploded view including complicated path, display path and entirely exploded of sub-assembly.

Meanwhile, CADbro added "**Record moving turn point**" and "**Add trails**" to enrich the exploded path. Besides, the option of "**Select subassembly part**" has achieved different exploded needs.



😹 Add Exploded Step 🗸 🗙 🖸 0 Required ANSIB18234M HXFGSW M8L2 ☆ ANSIB18234M_HXFGSW_M8L2 ANSIB18234M_HXFGSW_M8L2 ANSIB18234M_HXFGSW_M8L2 \times ۱ 🕹 4 Direction 0.27564,0,0.96126 ¥ 200 mm 🇘 0.27564,0,0.96126 ∻ - 📥 י 315 deg 🗘 垫 🔻 mm 🗘 垫 🔻 Auto-space 0 Settings Record moving turn point Select subassembly part Add trails

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



Chapter 15 HelpDesk



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





Chapter 15 HelpDesk



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



Explode Auto Explode

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









Add Exploded Trail

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



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

🗳 Add Exploded Trail	23
✓ X	0
▼ Required	
1st entity	-= 👲
2nd entity	-=1 👲
▼ Settings	
🗹 Along XYZ	





Auto Exploded Trail

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

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





Reuse Exploded View

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

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







Exploded View Video

Use this command to convert the picked exploded view from the lists to generate an AVI video.



¹ 3D BOM

Use this command to generate BOM table of all components of the assembly parts.

Level Display Settings
 Indented

Top-level only: Select this item to display only the top-level components.

Parts only: Select this item to display only parts.

Indented: Select this item and indent it by assembly level.

Shape only: Select this item to display only the shapes.

• Include shape & Update calculated attributes

Include shape: Click on the "Include shape" button, and the 3D BOM will display the shape information.

Update calculated attributes: Click on this button, update all the calculated attributes in the 3D BOM table.

Ø

Search Function	•		畿	Y	
-----------------	---	--	---	---	--

Select an attribute and enter the required value to search the corresponding content in the 3D BOM table.

Exact search: When using exact search, only the component information which coincides with t he search content will be displayed as the search results. After entering the content to be search ed but not clicking the button to start, the component information containing the search content t will be highlighted.

Filter search result: Display the search results according to the preset filtering conditions.





Export format: The current form of 3D BOM is output as a template, and the template is saved as .Z3DBOMTT format.

Export data: Export the current 3D BOM content to EXCEL.

Column

	Column	
•	Column	

All the optional attributes of the 3D BOM are displayed in the column tab, including system attributes and user attributes.

Attribute: Attribute is the corresponding name in the system and can not be modified.

Descriptive Name: The descriptive name is the corresponding name in 3D BOM and can be modified.

Include: Check the checkbox and the selected attributes will be displayed in the 3D BOM.

Total: Check the checkbox and the selected attributes will display total row in 3D BOM.

Add user attribute: Users can add user attributes whose types include string, Boolean, integer, numbers, and date.

ŝ	🛛 3D BOM 🖓 🖓														
Ind	ented	- 📬 🗇	ID	•				K			🗽 🔹 🕨 🤇	Column			
	ID	Name	Material	Cost	Quantity	Number	Manager	Mass (kg)	Volume (mm^3)	Stock Size		Attribute	Descriptive Name	Include	Tot 🔺
1	1	E020501B	Steel	10	1	ZW-01	lvy	7.44	953363.53	410.00x130.00x73.77(mm)		ndex	ID		
2	2	BEARING	Steel	5.9	1	ZW-02	lvy	0.18	22858.07	13.00x80.00x80.00(mm)	P	Name	Name		
3	3	E023001B	Steel	1.5	1	ZW-03	lvy	0.28	35984.15	55.00x56.00x56.00(mm)	N	Number	Number		_
4	4	E052002A	Aluminum	3.2	12	ZW-04	lvy	0.01	2192.45	20.00x20.00x7.00(mm)	C	Class	Class		
5	5	E051502B	Aluminum	13	1	ZW-05	Leo	0.06	23088.58	26.00x132.01x132.00(mm)	C	Designer	Designer		_
6	6	E052501A	Steel	22	1	ZW-06	Leo	0.97	124341.03	62.50x138.00x138.00(mm)	C	Cost	Cost		
7	7	E051001A	Steel	80	1	ZW-07	Sam	0.66	84135.22	8.00x245.00x244.98(mm)	S	Supplier	Supplier		_
8	8	E052502A	Steel	55	1	ZW-08	Leo	0.14	17718.83	134.32x24.00x134.32(mm)	h	Material	Material		
9	9	DIN912-M6x25-8.8-YP	Steel	1.2	8	ZW-09	lvy	0.01	940.66	10.00x10.00x31.00(mm)	C	Quantity	Quantity		
10	10	NORDLOCK-M6	Steel	0.3	8	ZW-10	lvy	0.00	110.64	11.00x11.00x1.80(mm)	h	Manager	Manager		
11	11	ASSY piston+conrod	Aluminum	100	4	ZW-11	Leo	1.16	269962.80		C	Description	Description		_
12	11.1	conrod	Aluminum	55	4	ZW-12	Leo	0.47	174198.78	106.00x106.00x59.70(mm)	ĸ	Keywords	Keywords		
13	11.2	E024501A	Steel	25	4	ZW-13	Leo	0.06	7148.33	105.60x105.60x15.25(mm)	S	Start Date	Start Date		
14	11.3	E025002A	Steel	0.5	8	ZW-14	lvy	0.00	185.39	1.50x25.40x25.40(mm)	E	End Date	End Date		
15	11.4	E025001A	Steel	16	4	ZW-15	Sam	0.13	16274.94	76.00x23.00x23.00(mm)	N	Mass	Mass		
16	11.5	E023505A	Steel	20	4	ZW-16	Sam	0.49	62768.38	23.00x74.00x179.50(mm)	V	/olume	Volume		
17	11.6	E022004A	Aluminum	9.5	8	ZW-17	Leo	0.01	2213.26	19.00x51.79x25.53(mm)	Т	Total Mass	Total Mass		
18	11.7	DIN912-M8x1x35-12.9	Steel AISI 41	2.5	8	ZW-18	lvy	0.02	2387.53	13.00x13.00x43.00(mm)	C	Density	Density		
											F	File Path	Source file path		•
									(Columns from templ	ate	-			
	Type String *						•	¥	-						
		Name													
													Add user attrib	ute	
								Reset	OK Can	el Apply					



14.2 Mold



This feature is used to realign geometry after data is imported. Click to enter the interface.



This command is used to define a product's shrinkage.



The command according to the regularity of mold structure can quickly review all the surface properties of the model. In Auto Define, selecting a product model and determine the opening mold direction (default is z-axis). System will automatically analyze and get different colors according to different parts of a mold, which distinguishes between core and cavity easily. System goes into Manual Define when completing automatically. Users can view the analysis results under regional options. In this interface, users can change their mold structure to satisfy the design needs so as to provide basis for future mold design. The core and cavity areas have been defined when the Undefined Faces show zero.



Project Silhouette to Plane

Use this command to project the shape silhouette to a plane. You can control to display the outer loop of silhouette only or all curves of it.







Project Shape to Plane

Use this command to project the shape silhouette to a plane, and the result is face. You can control to display the outer loop of silhouette only or all loops of it.





Use this command to analyze faces for surface imperfections, draft angles, curvature, and other


surface characteristics. (See Chapter 5.2 Inspect Entity-> Analyze Faces)



Thickness Analysis Thickness Analysis

Use this command to analyze the thickness of parts and help users to estimate whether the design is correct.



Undercut Analysis

Use this command to analyze whether there are any undercut faces in the model. It can assist users to estimate where to set sliders or other systems. Besides, it can also help users to select appropriate operation and clamping method, in order to create a complete tool path for machining. After defining the detect direction (parting direction or cutter axis direction), CADbro will automatically color the whole part.

Tips:

- For mold analysis, it should check on the "include negative direction".
- For CAM analysis, it should not check on the "include negative direction".



Height Analysis

Use this command to detect all the plane parallel to a specified datum, and list them in the table. It is easy for the user to find out the important plane and calculates the plane numbers. Tips:

Height analysis cannot be used in assembly.



Height Analysis Result

This command is used to record the result of the last time Height analysis. In this mode, the user can view the part with colors, and add PMI or annotation in those faces.

Tips:

Before Height Analysis, the command will be gray out.



15 HelpDesk





User Folder Manager

Use this command to manager the user folder. In this command, user can clear "auto output files", "custom configurations", "template files" or "user folder". If user want to save/load configuration, please click "Save current configurations" or "Load backup configurations".

? About About

Use this command to check the information about CADbro. In this page, you can view the Version, VERNUM, OS Version, Graphics Card, OpenGL Version, User Folder, Program Folder and license information.







Learning Video Learning

Use this command to link to the CADbro video learning website.



Mail Mail

Use this command to sent a email to CADbro service team.



Bug Report Bug

Use this command to report errors to CADbro service team. When using this function to send reports, the system automatically sends files and computer system information to the CADbro service team.



16 2D Drawing

Using the 2D Sheet command from tool ribbon



CADbro can automatically generate a 2D view as below. Location, projection angle and different layouts can be changed.



Working with 2D Drawing, you can create a simple dimension and draw 2D entities manually.





Standard Standard View

Use Standard View to create a standard orthographic layout view from a 3D part. Steps are as below:



Step1: Pick the Standard View command.

Step2: Select the object from a file.

Step3: Pick a view type.

Step4: Set the detailed parameters like scale and label.

Step5: Place the standard view on the work area.







Projection View

Use Projection View to create a view projected from another existing 3D layout view. Steps are as below:

Step1: Pick the Projection View command.

Step2: Select an existing view to generate the projection view.

Step3: Set the detailed parameters.

Step4: Place the projection view on the work area.







Auxiliary View

Use Auxiliary View to create an auxiliary view that is projected perpendicular from an edge of another layout view (referred to as the base view). Steps are as below:

Step1: Pick the Auxiliary view command.

Step2: Select an existing view to generate and choose a line for alignment.

Step3: Place the Auxiliary view on the work area.







Section Full Section View

Use Full Section View to create a fully sectioned views base on a 3D layout view. Steps are as below:

Step1: Pick the Full Section view command.

Step2: Select an existing view to generate the section view.

Step3: Define the section line.

Step4: Place the section view on the work area.







Detail View

Use Detail View to create circular and rectangular detail views from another 3D layout view. **Step1:** Pick the Detail View command.

Step2: Select an existing view to generate a detail view.

Step3: Define the area you want to zoom in on.

Step4: Place the detail view on the work area.

Step5: Set the detailed parameters.







Crop View

Use Crop View to create a partial view by trimming a drawing view with a defined boundary.

Step1: Pick the Crop View command.

Step2: Select the crop type and pick an existing view to generate the crop view and define the boundary.





17 Cloud service

17.1 Create a cloud service account

Before using cloud service, you need to sign up an account for CADbro cloud service. Please follow the below steps to create a CADbro could service account.

- 1) Go to CADbro Cloud website (https://cloud.cadbrother.com/v3/login).
- 2) Click "Sign up" in the upper-right corner.
- 3) Input your country/region, email address, verification code and password.
- 4) Click "Sign up" and then the system will send you a verification mail.
- 5) Login your mailbox to finish the account verification.



17.2 Personal space management

CADbro Cloud will give each user an online storage space of 100 Mb capacity for free to experience the cloud service. And users can check the basic information (capacity and personal setting) and manage the uploaded files.



CADbro files of member × +				- 🗆 ×
← → C	sonal/files			0, ☆
CADbro Perso	nal Space		📮 7.5MB//100.0MB	En~ 🔮
Teams Create Create	Files Shares Recycle Bi	n	Capacity	
I Personal space	All files		Search file	Q
	File name	Type Size	Owner	Creation
	🔲 👵 ERT912201 - Mk2 - [ERT912201 - Mk2].z3ds 😡	z3ds 2.2MB	gofyyeung@gmail.com	2010-12-04
	🔲 💀 S2-C3 - [S2-C3].z3ds 📢	z3ds 47.7KB	gofyyeung@gmail.com	2019-12-04
	🗌 👵 UAV - [Middle Frame_Front Support].z3ds v1	z3ds 93.1KB	gofyyeung@gmail.com	2019-12-04
	🗌 👼 Electrode - [Core_V2].z3ds 📢 🧠 😤 💼	z3ds 3.0MB	gofyyeung@gmail.com	2019-12-03
	Open the model Check the versions Create share link	Version history	ete	

17.3 Team space management

After creating a CADbro cloud service account, we can create a team with cloud service license to manage the files by projects and start project collaboration.

17.3.1 Create a team

To create teams, you need to purchase a cloud service license. Creation procedure is as below:

- 1) Click "Create" in the upper-left corner.
- 2) Input your team name, cloud service license, and team description.
- 3) Click "Submit" to finish.



CADbro files of member × +			- 0	×
\leftrightarrow \rightarrow C $($ cloud.cadbrother.com/v3/person	nal/files		07	☆
<pre> CADbro Person </pre>	al Space	🚍 7.5MB//100.0MB	En~ 🄇	
Teams Teams Create	Files Shares Recycle Bin			
Personal space	All files	Search file	Q,	
	File same	Owner	Creation	
	Create Team ×	gofyyeung@gmail.com	2010-12-04	
	Team name	gofyyeung@gmail.com	2019-12-04	
	ZWSOFT INTERNAL	gofyyeung@gmail.com	2019-12-04	
	License 4133051C3E8D8804E82A3A1CF42D3FBF	gofyyeung@gmail.com	2019-12-03	
	Team description			
	ZWSOFT internal testing team			
	Cancel			

Notice: When the team is created successfully, the account creating it will become the super manager of the team. The super manager will have the permission to:

- change team settings such as the team name, description and reset the team
- invite members to join the team or remove members from the team
- create projects
- assign roles (manager, member and visitor) to members
- manage all the files in the projects

17.3.2 Invite members to a team

There are two methods for super manager to invite members to join the team.

Email: Send an invitation email to members. The members who are invited need to accept the invitation via email.

URL: Generate invitation urls in CADbro cloud and send to the members to be invited. The invited members need to load the URL for acceptance.



Teams × +	- 🗆 X
\leftrightarrow \rightarrow C $(=$ cloud.cadbrother.com/v3/tear	n/1575449224267542696 📴 🖈
∢∧≻CADbro	א 1/100 ☐ 0.0B/1000.0GB En√ 🔮
Teams Create A ZWSOFT INTERNAL	General Members Projects Create project
rersonal space	Add member
	panjiahui@zwsoft.com Invite Create Invite url
<	•
1	

17.3.3 Create a project

The super manager can create projects, assign roles to members, distribute storage capacity and share links of projects. In team collaboration, model files can only be stored in projects.

₩ Teams	× +		– 🗆 ×
← → C 🔒 cl	loud.cadbrother.com/v3/team/15754492242	67542696	ም ጵ
<a>CADb	oro		ಞ 0/200 🖾 0.0B/1000.0GB En∨ 🔮
La Teams	Create	General Members Project name Uxx respect Project name Uxx respect Project description Ux vespect Project scale General Solo *Max t00 Members Solo Manger Solo Dan Manger Solo Solo Solo Solo Manger Solo Solo Solo Solo Manger Solo Solo Solo Solo	Create project

17.3.4 Add members to a project

Super manager or manager can add members to a project.



CADbro file management	× +		- 🗆 X
\leftarrow \rightarrow C \cong cloud.cadb	orother.com/v3/team/15754492242675426	96/project/1575451567017542033/files	~ ☆
<a>CADbro	ZWSOFT INTERNAL >	UAV PROJECT	& 2/50 □ 0.0B/100.0GB En∨ 💿
🔊 Teams	Create	Files Shares Recycle Bin Members 2	Settings
R ZWSOFT INTERNAL		Project members	
🔁 UAV PROJECT		Add new member Add 3	
🗈 Personal Space		dengweibin@zwsoft.com	
		Goofy (Member) Set Role	Remove member
		Add new member $4 \times$	
		Leo Leo.zhang@zwsoft.com Jayson fengjiexiong@zwcad.com	

17.3.5 User roles

Super manager or manager can assign different user roles to project members.

Manager has the permission to change project settings, and manage members and files in projects, including uploading/updating models, creating share links for models, and viewing models.

Member has the permission to upload/update models, create share links for models, and view models.

Visitor is only allowed to view models.

	Files	Shares	Recycle Bin	Members	1	
Projec	t membe	ers				
P+	Add ne	w member	Add			
P	Terry dengwei	Manager bin@zwsoft.co	m			
6	Goofy (panjiahu	Visitor ii@zwsoft.com		2	Set Role	Remove member
-	Leo Su leo.zhan	per Manager) g@zwsoft.com			Member	Remove member
	Jayson fengjiexi	Member ong@zwcad.co	m		Set Role	Remove member

17.3.6 File management

After creating projects and assigning user roles, super manager, project manager and member can view, share, delete and restore files via web or client. Note: Uploading is only supported in desktop client currently.





<a>CADbro z	WSOFT Test Team 👌 Proj 2			桑 2/100 📋 5.0MB/10	0.0GB En~
R Teams Create	Files	Shares Recycle Bin Members			Settings
& ZWSOFT TEST TEAM	All files			Search file	Q
🕞 Project 2	File name	ype	Size Owr	her	Creation Time
🔁 Project 1	🗌 👵 Camera_V1 - [ASM].z3ds 🔽	๙空盲 z3ds	1.2MB den;	gweibin@zwsoft.com	2020-07-03
🗈 Personal Space	.0 Assembly - [.0 Assembly].z3ds v1	z3ds	1.6MB 180.7	zhang@zwsoft.com	2020-02-20
	🗌 🐱 engine - [Engine ssembly].z3ds 📢	3ds	N9MB den;	gweibin@zwsoft.com	2020-02-17
	🗌 💀 Cabinet - [Ass].z3ds 📢	z3 ls	391.5KB leo.2	zhang@zw.coft.com	2019-12-10
	View share details Share files	View version history Delete	file dela	lanage M sted files me	lanage embers

File share

Steps for creating a share link.

- 1) Click icon near the file to be shared.
- 2) Set the sharing options, such as expiry date, password, view times and so on.

Sharing:01 Import igs1 - [01 Import igs].z3ds $ imes$							
Title	01 Ir	nport igs1 - [01 Imp	ort igs].	z3ds			
Expiry period	• Pe	Permanent Pick the date					
Encrypt	© Er ● Pl	Encrypted accessPublic access					
View Times	• Ur	nlimited \bigcirc 10 \bigcirc 1	50 🔍 10	00			
Advance	d 🗸	Assembly tree	✓	explode			
Setup	✓	section	✓	PMI			
	✓	Measure	✓	Drag			
	\checkmark	Animation					
l	Submit			Cancel			

3) Click "Submit" to finish.

Management of shared files

In Shares tab, users can copy share link and password (if there is), modify the share options, cancel sharing, and view the current usage of the shared files.



CADbro zwso	FT Test Team > Proj 2	& 2/100 🖾 5.0MB/100.0GB En√ 🗲
Teams Create	Files Charge Recycle Rin Members	Settings
& ZWSOFT TEST TEAM	Thes shares Recycle bin Weinbers	Searge
🔁 Project 2		
🔁 Project 1	TitleOwner	Hits Creation time Expiry date
🗈 Personal Space		vsoft.com 0 2020-07-07 -
<	Check sharing link Modify sharing details	Cancel sharing

Sharing methods

You can share files via links or QR codes.



> File history

Click icon to invoke the History list which records all the operations on the current files, e.g.

create, edit and update to a newer version.

Hist	ory					×
	Operation	File name	Versior	n Time	Size	
	update	01 Import igs1.z3ds	v2	2019-11-19	562.4KB	
	edit	01 Import igs1.z3ds	v1	2019-11-19	561.8KB	
	edit	01 Import igs1.z3ds	v1	2019-11-19	561.8KB	
+	create	01 Import igs1.z3ds	v1	2019-11-18	561.8KB	

Delete/Restore files

Click icon to delete any files in the project. The deleted files will be stored in the Recycle Bin temporarily. Users can restore the file or delete them permanently.

	Files	Shares	Recycle Bin	Members				Sett	tings
All files									
File name					Туре	Path	Size	Creation Time	е
🗌 👵 Cabinet - [Ass].z3ds				t c	z3ds	/	391.5KB	2019-12-10	6
			Perma dele	nently ete	Restore				



17.4 Collaboration

Members in the same project can use collaboration function in the CADbro to share the views, annotations and chat in real time. Starting the collaboration session requires you to login CADbro Cloud and upload a file first.

17.4.1 Login CADbro

Steps for logging in CADbro Cloud in desktop client

- 1) Launch CADbro.
- 2) Go to the "Start" toolbar, and click Login command.
- 3) Input your account and password to login.



17.4.2 Upload models

Open the file in CADbro, click Upload File command and select a project it will be uploaded to. Note: CADbro will convert the model into light-weight format (.z3ds) and upload it to the Cloud server.



Chapter 17 Cloud Service



17.4.3 Start collaboration

Steps for starting collaboration:

- 1) Choose a model from a project and right-click it.
- 2) Select "Collaboration".



- 3) Input a name for the collaboration.
- 4) Select members to join the collaboration
- 5) Describe the collaboration if needed.
- 6) Click "OK" to start the collaboration



Chapter 17 Cloud Service

Create Collaboration 3	₽ X					
Collaboration Name UAV Design Optimization						
Please Select Collaborators	Select All					
Name	Status					
🗷 👶 dengweibin@zwsoft.com	Available					
🗷 🕹 Goofy 🛛 🗳	Available					
🗌 🌡 Jayson	Collaborating					
Collaboration Description 5 This collaboration is created for design optimization, including: 1. Appearence design confirmation 2. Stock size and interference check 3. Centerpoint position check 4. Local details check 6						
Ok Cancel						

17.4.4 Operations in collaboration

In collaboration, users are allowed to share views, annotations and chat in real time.



1) Add Members

Click this icon to add new members to the current collaboration.

2) Sync Switch 🔽 & 🧱

Click these icons to pause/continue the Sync.



User Manual

Click this icon to show the current state of the part to newly-joined members.

4) Leave Collaboration 📥

Click this icon to leave the current collaboration.

Note: The controller must hand over the controller right to another member before successfully leaving the current collaboration.

17.4.5 Playback

System will record all the operations during the collaboration and save them to the collaboration folder. A playback record will be helpful to the ones who didn't attend the collaboration or the ones who want to review the collaboration process. Below are steps to review the playback record.

Step1: Find the target collaboration in Space > Collaboration.

Step2: Right-click the collaboration and click "Entry" to enter the playback environment.

Step3: Review the playback record. There are many options for users' better experience.



17.4.6 View 3D models in browser

By loading the share links, users can open models in a web browser. They can:

- View 3D models
- Check assembly structures
- Show or hide components
- Check explosive view
- View by sections



- Measure
- Move components



And if there is pre-defined exploded view in ZW3D or CADbro desktop version, you can view it in the browser as below:







18 Filter

Filter is important to control the select entity from the graphic window.

The pick filter list shows the entity types that the active command prompt is looking for. The list is displayed when the pick filter icon is selected (see above) of when (Shift) key and the right mouse button (RMB) are selected together. When you select an entity type from the list, only those entities can be selected or highlighted. You can pick the Attributes button (when displayed) to filter by additional attributes.



Holding the Shift key + Right mouse button, the filter list will display on your mouse pointer position.





19 Interface customization

You can right mouse button click on the ribbon empty area, then you can see interface you customization command. Or can directly click the Customize... command from the Ribbon tab.

	Ribbon Appearance 🔹 🕨			
	Styles 🔸	Classic Black	1	
	Customize	Blue		
		Silver		
	/ 🛧 🛥	Black		
	🎸 🄰 🧊 *	✓ Flat Silver	1	
File Start View Selection To	ools Analyze PMI Healing Edit Pro	0		
🗲 🚰 🎒 📑	🎽 👫 📙 🚔	🗠 🗠 🐯 🖡	📑 💊 👔	
Open Import Quick Save E View As	xport Multi-Export Close Close Print/Plot All	Undo Redo Configuratio C	Customize License Applic Manager Mana	ation Start ager Another
	File	Control	Utilities	

What you can customize?

You can add your own command.

൜ Customi	ze						₽ %
Command	s Transfer	Hotkey	Mouse				
Comman	d List				Property		
Туре:	Part		•		name:	My command	
Group:	Start		-		ribbon text:	new1	
Search:					menu text:	new1	
? Abo	ut			11	icon:		
Clos	se α ΔΠ				hint:		
Con 🎯	figuration				description:		
Cust	tomize nse Manager						
Nar	nual						
new	1						
Conii Cone	ine Learning •n						
Prin	t/Plot						
👝 Red	0						
🔚 Save	e						
nd The	lo						
		Add	or Remove				
		_	\frown				
Defau	ılt	Add/Delete C	ommand: 🕂 😑				Change Image
			OK Ca	ncel	Apply		

Transfer the command between different ribbon tab, you can also create a new ribbon tab.

📣 Customi	ze		∇
Command	s Transfer Hotkey Mouse		
Comman	d List		Environment
Туре:	Part	-	Name: Part/Assembly *
Group:	Start	•	File: Controls.zcui 🔹 🖶 🚍 Default
Search:			Menus 4 Pibbon
? Abo Clos Clos Cos Cust Mar Onli Prin Prin Redu Save Vnd	out se se All figuration tomize ense Manager nual ine Learning en t/Plot o e		 Nobon Start View Selection Tools Tools Analyze PMI P Healing Edit P ro ToolBars Quick Access Toolbar Document Aware Toolbars

Customize the hotkey for CADbro.

User Manual

 \checkmark

Customiz	re 5 Transfer Hotkey Mouse		φ Σ
_			
Category:	All	*	Default
Search:			Just show items with hotkey
Menus			
Quick /	Access Toolbar		
5	Open	Ctrl+O	=
	Save	Ctrl+S	
~	Undo	Ctrl+Z	
~	Redo	Ctrl+Y	
A Docum	ient Aware Toolbar		
	Erase		
- 2	Turn "Direct Edit" On/Off		
⊿ 💟	Removed hidden lines		
	Removed hidden lines		
	🖤 Display Edges in Shaded Mode		
▲ 🛞	Auxiliary View		
	😚 Auxiliary View	Ctrl+U	
	💕 Top View	Ctrl+Up	
	Isometric View	Ctrl+I	
	🧈 Left View	Ctrl+Left	
	👎 Front View	Ctrl+Down	
	🌮 Right View	Ctrl+Right	
	🥵 Back View		
	🧩 Bottom View		
	🕒 Auto Align	Ctrl+Home	
	💕 Align with Direction		
⊿ 🔘	Zoom All		
	700m ΔII	Ctrl+∆	•
Descriptio	n		
	OK	Cancel Apply	

Customize your mouse action, with this function below, you can customize CADbro mouse

actions to be the same as other 3D systems, this will make you easy to get used to working with CADbro.

📣 Customi	ize				₽ 3	23
Command	ls Transfer Hotkey Mouse					
-Mouse A	ctions					
	Function Key			Mouse Button		
Pan	Null	•	+	Middle Mouse Button	•	
Rotation	Null	•	+	Right Mouse Button	•	
Zoom	Ctrl	•	+	Middle Mouse Button	•	
Revers	e mouse wheel zoom direction				Default	
		-	ali			~



User Manual

Click the Windows button, find the ZWSOFT folder and click the button Change Language.



Then you can choose one to change the installation language

Choose a language to install

English	~
中文(简体) 中文(繁體) 日本語 한국어	
English	
Deutsch polski čeština Português(Brasil)	





21 Online update

Click Windows button, find the ZWSOFT folder and click the button Check For updates.







User Manual

Click Windows button, find the ZWSOFT folder and click the button Set File Associations.



Use this command you can set the formats to link with CADbro. When you double click on the format then it will default to open in CADbro.

