User Manual

CADbro Version 2020

Dec.2019

Copyright 2020 ZWSOFT Co., Ltd. (Guangzhou)

Contents

1. System requirements....1
2. Softkey activate....2
2.1 Stand-alone Version....2
2.1.1 Activation....2
2.1.2 Returning a License....10
2.2 Network Version....18
2.2.1 Activation....18
2.2.2 Using Network License....26
2.2.3 Returning Network License....33
3. Basic knowledge....40
3.1 Start CADbro....40
3.2 Interface introduction....41
3.3 Open a new file....42
3.4 File Browser....42
4. Manager....44
4.1 Part Browser....44
4.2 Assembly manager....44
4.3 View manager....45
4.4 Visual manager....46
4.5 Export manager....46
5. Covert files....48
6. View....52
6.1 View Angle....52
6.2 View....53
6.3 Visibility....53
6.4 Display Mode....53
6.5 Screen....54
6.6 Windows....54
6.7 Rotation....54
7. Selection....55
8. Tools....56
8.1 Dimension....56
8.2 Curve creating....56
8.3 Attribute....56
8.4 Section....59
8.5 Capture....62
8.6 Assist....63
9. Analyze....65
9.1 Measure....65
9.2 Inspect Entity....67

9.3 Inspect Model 69
10. PMI 70
10.1 Manager 70
10.2 Dimension 70
10.3 Annotation 71
10.4 Text 71
10.5 Tools 71
11. Healing 73
11.1 Checking 73
11.2 Healing 74
12. Edit 76
12.1 Direct Edit 76
12.2 Shape transform 78
12.3 Part transform 79
13. Mark up 80
14. Pro 84
14.1 Assembly 84
14.2 Mold 89
15. 2D Drawing 92
16. Cloud service 99
16.1 Create a cloud service account 99
16.2 Personal space management 99
16.3 Team space management 100
16.3.1 Create a team 100
16.3.2 Invite members to a team 101
16.3.3 Create a project 102
16.3.4 Add members to a project 102
16.3.5 User roles 103
16.3.6 File management 104
16.4 Collaboration 106
16.4.1 Login CADbro 106
16.4.2 Upload models 107
16.4.3 Start collaboration 107
16.4.4 Operations in collaboration 109
16.4.5 Playback 110
16.4.6 View 3D models in browser 111
17. Filter 112
18. Interface customization 113
19. Changing language 116
20. Online update 117
21. Set File associations 118

1. System requirements

CADbro System Requirements

Operating System	CADbro Microsoft® Windows 7 SP1 Microsoft® Windows 8.1 Microsoft® Windows 10 Note: Windows 7 SP1 & Windows 8.1 may not install the KB2999226, resulting in the incomplete installation of the VC2015 plug-in. When Starting CADbro, it will alter "api-ms-win-crt-runtime-l1-1-0 DLL". The solution is to install KB2999226 package, please refer to the following Microsoft's official website link: https://support.microsoft.com/en-us/help/2999226/update-for-universal-c-runtime-in-windows CADbro Network License Manager: Microsoft® Windows Server 2008 R2 SP1 Microsoft® Windows Server 2012 Microsoft® Windows Server Microsoft® Windows 8.1 Microsoft® Windows 10
Processor	Intel Core 2 Duo @2GHz or above, or equivalent AMD® processor
RAM	Recommended configuration: 4G or above
Video	OpenGL 3.1 or above nVIDIA Quadro FX 580 @ 512MB or above, or equivalent AMD Graphic card Minimum configuration: 1024 x 768 VGA with True Color Recommended configuration: 1280 x 1024 VGA with True Color or above
Required for licensing	Soft Key
Required for Floating License	Installation of TCP/IP Network Protocol

2. Softkey activate

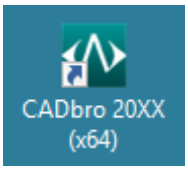
2.1 Stand-alone Version

2.1.1 Activation

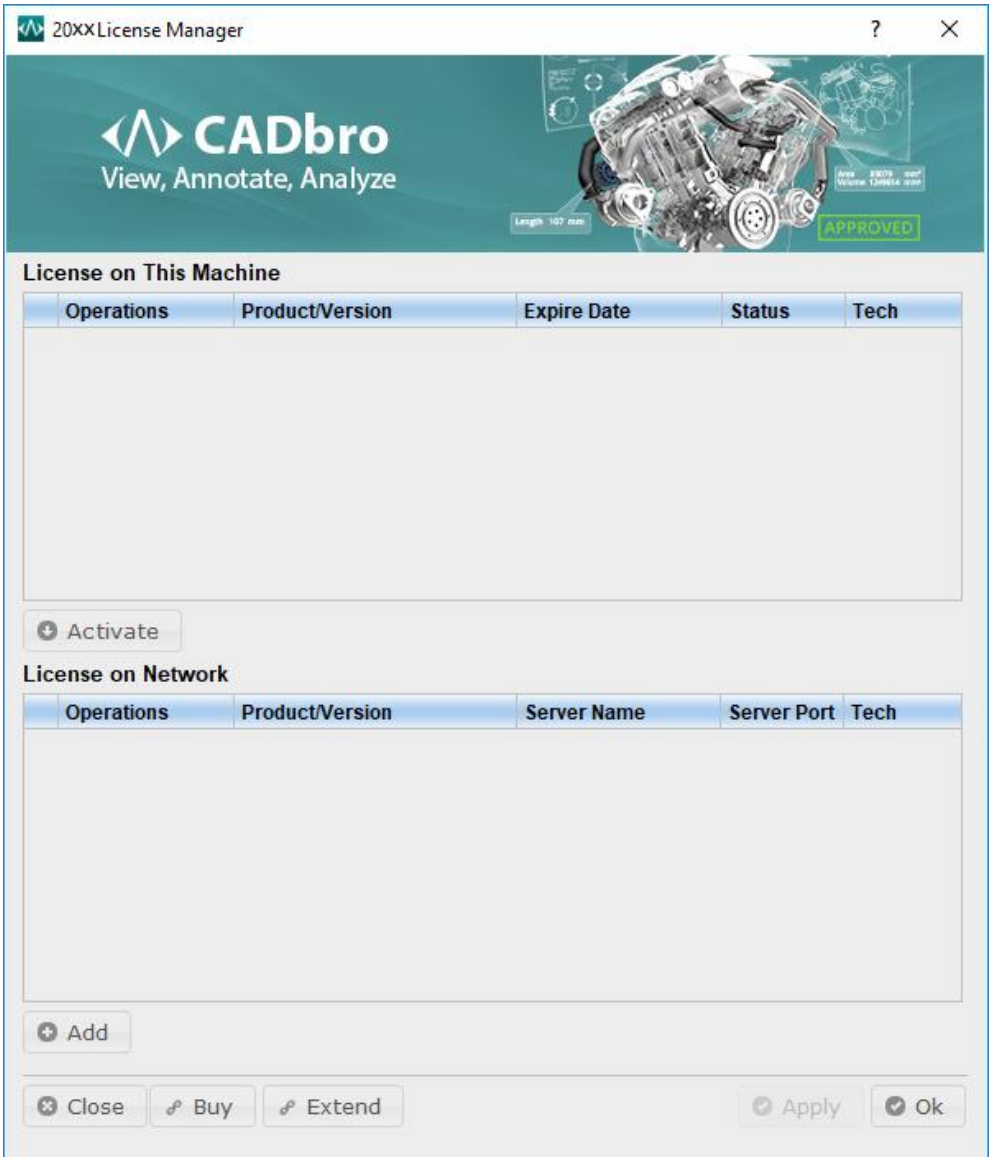
If you wish to operate CADbro with full functions, it must be activated. Activation needs a soft key, if you do not have a Soft Key, please contact the local dealer from whom you purchased CADbro.

Please follow these steps to activate CADbro online:

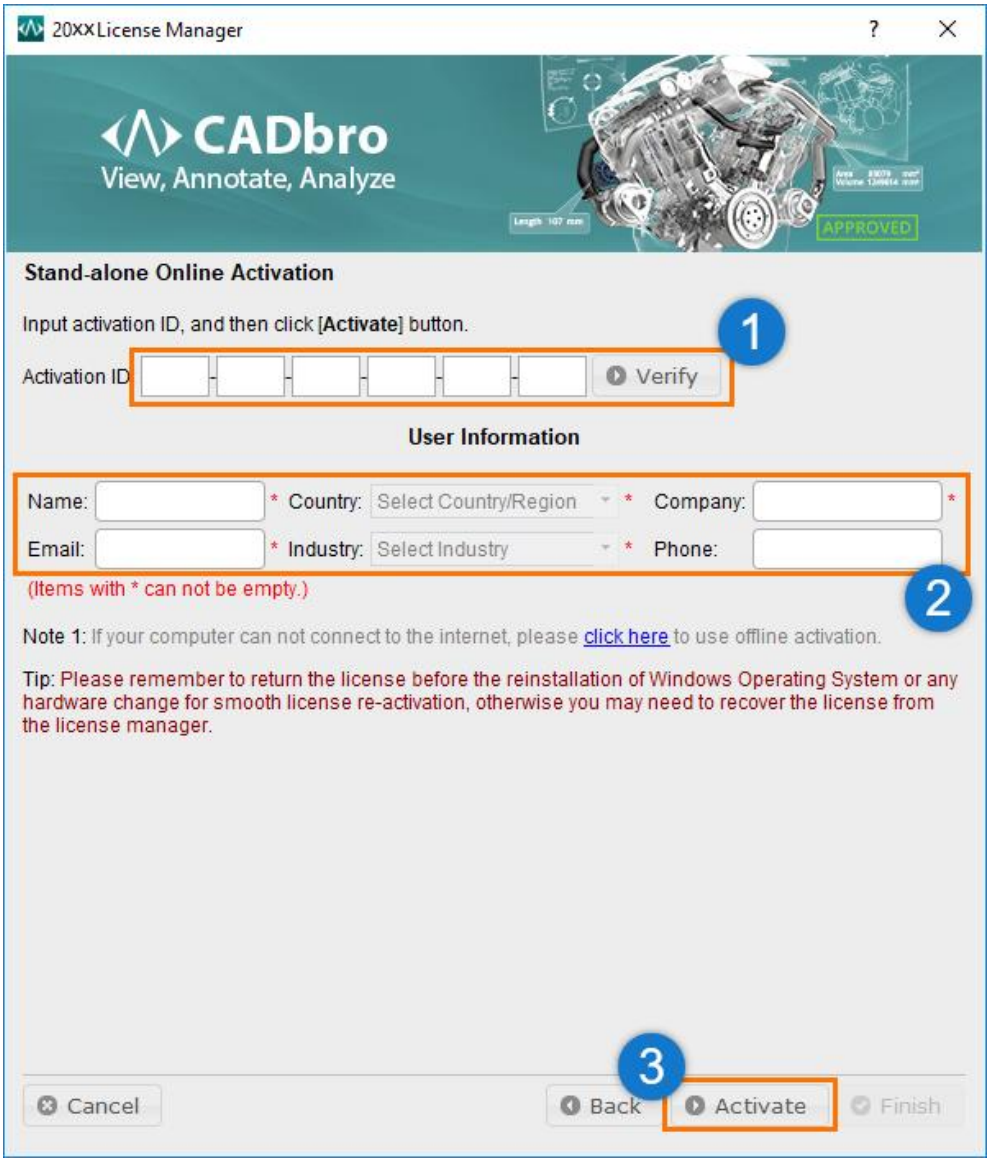
1. On the desktop, double-click the **CADbro 20XX** icon.



2. You can see the activate interface.

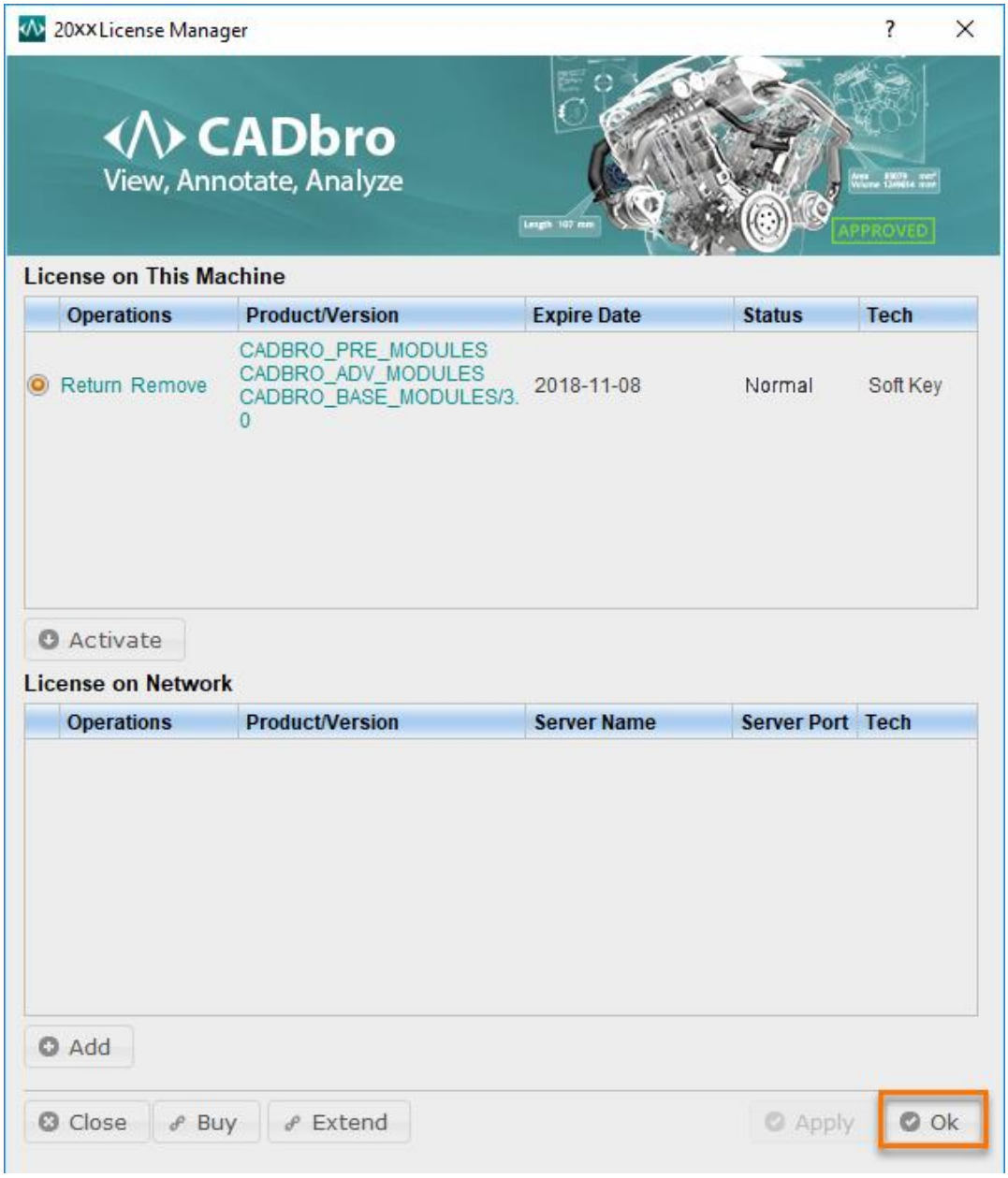


3. Enter the Activation ID (Soft Key) in the text box and press the ***Verify*** button. If the Activation ID is used the first time, the user needs to enter your information. Click ***Activate***. The activation will be successful after a few seconds.



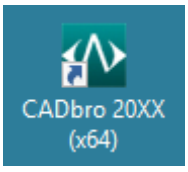
4. Click ***OK*** to close the successful activation message, and back to the license manager.

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

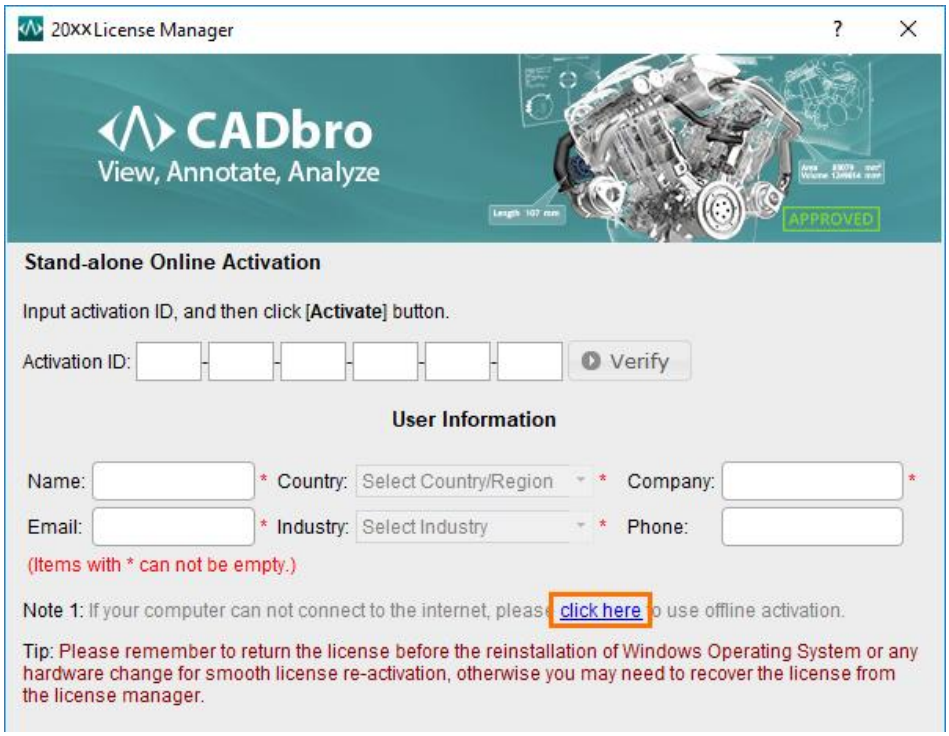


In the case of no Internet connection on the local computer, you can activate CADbro through procedures illustrated as follows:

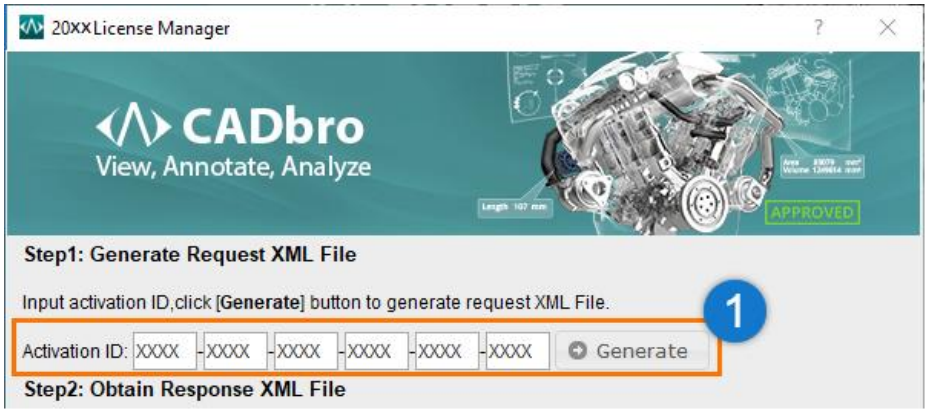
1. On the desktop, double-click the **CADbro 20XX** icon.

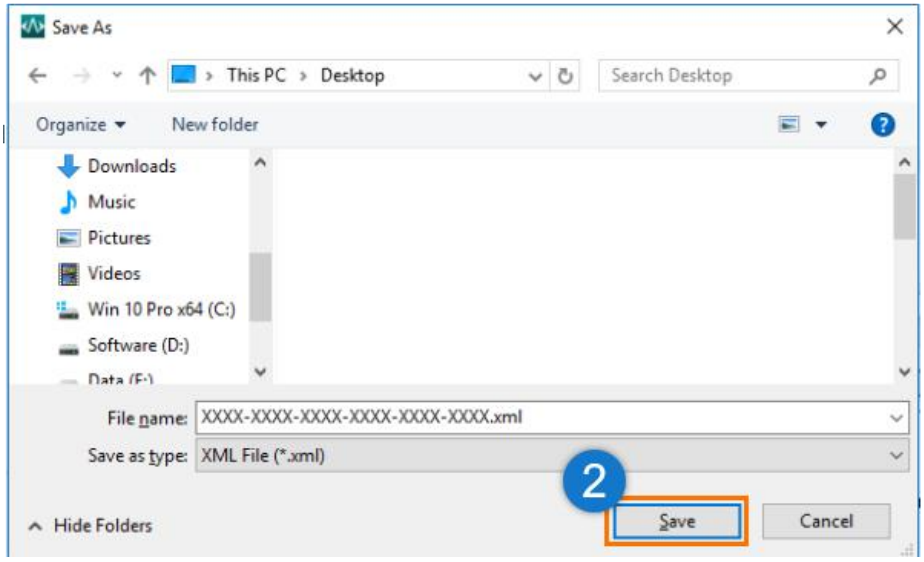


2. In the Online Activation dialog box, click the **click here** to go to offline activation dialog.



3. In the Offline Activation dialog box, fill in the Activation ID (Soft Key) and **Generate** a Request XML File. Save the file onto the local computer.



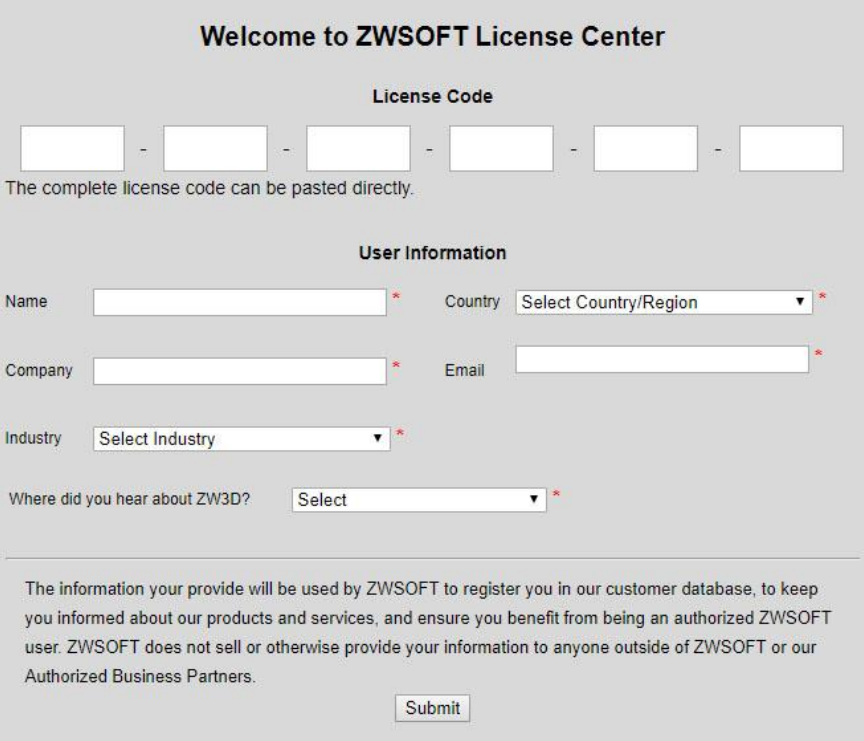


4. Go to another computer with an Internet connection and use the Request XML File to obtain a corresponding Response XML File via E-mail or Online Self-Service

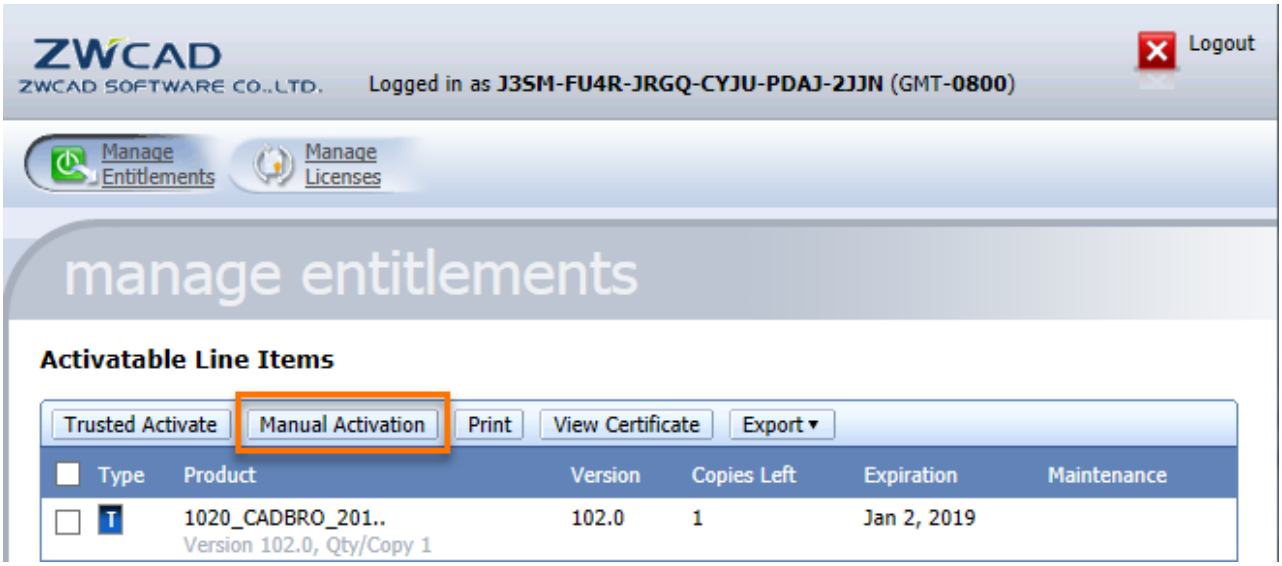
a) Option 1: E-mail
You may send the generated Request XML File to the ZWSOFT License Center at register@zwcad.com. The ZWSOFT License Center will send you the corresponding Response XML File as soon as possible.

b) Option 2: Online Self-Service:

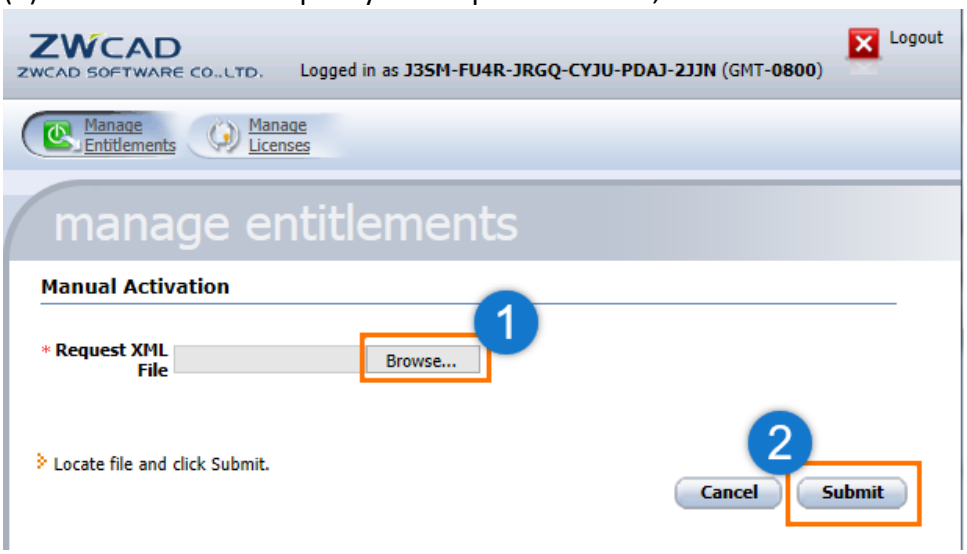
(1) Go to the ZWSOFT License Center at http://license.zwsoft.com/zw3dCustomer.aspx and ***Login*** with the Activation ID (Soft Key).



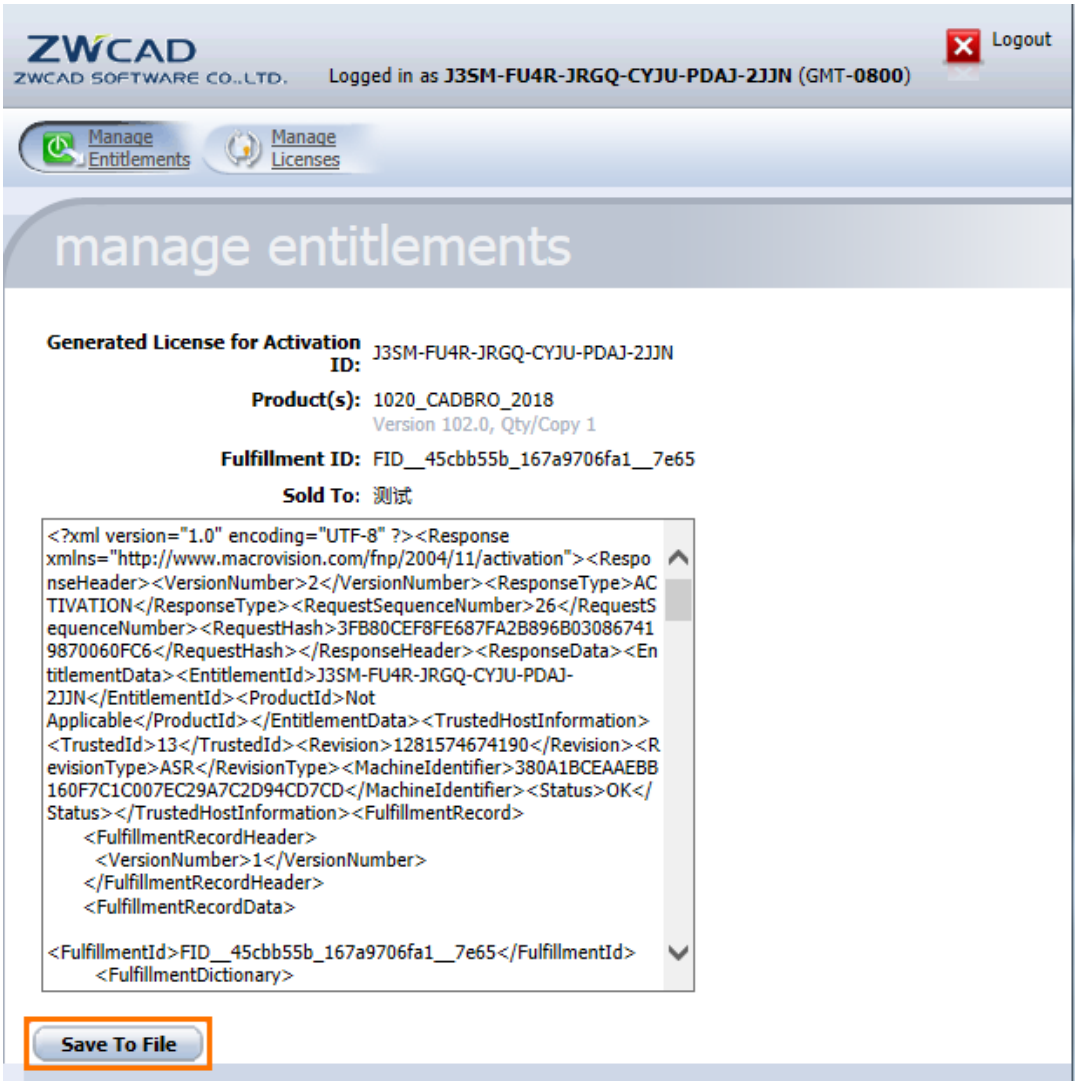
(2) Choose ***Activate*** or ***Manage Entitlements***, and then click ***Manual Activation.***



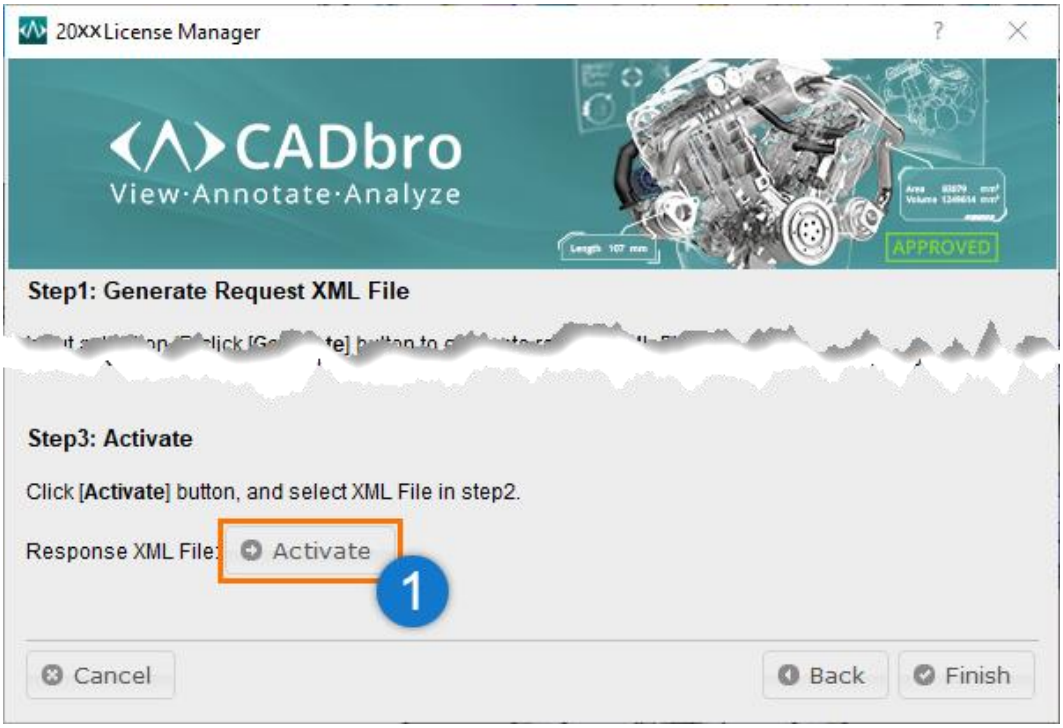
(3) Click ***Browse*** to import your Request XML File, and then click ***Submit***.

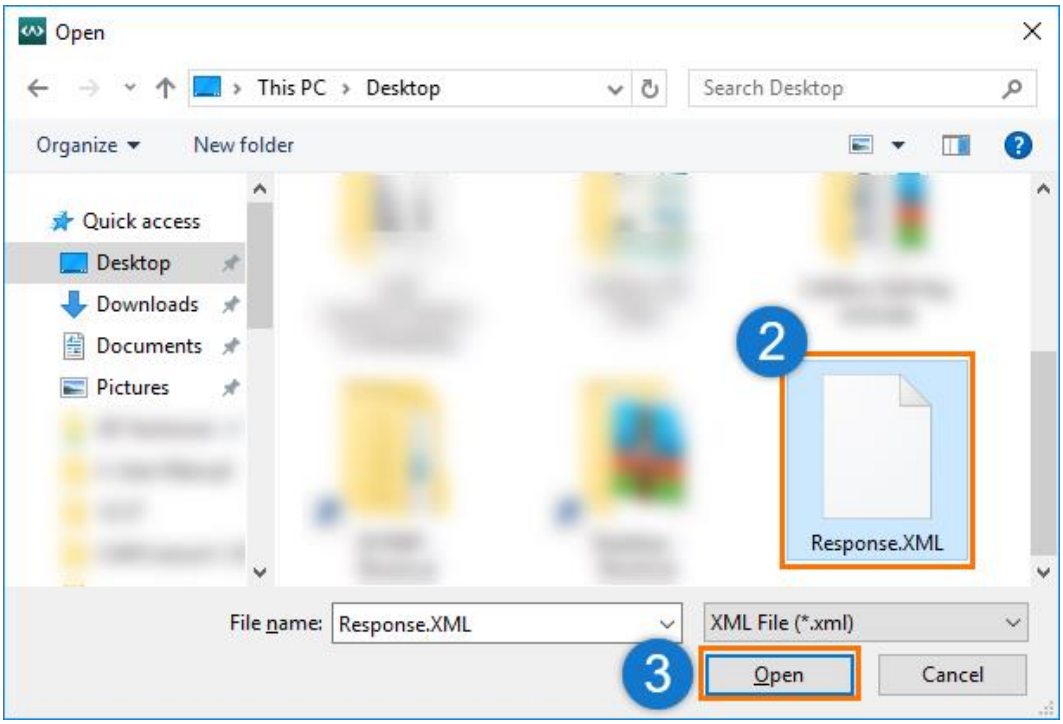


(4) Then you will be able to see the information of the corresponding Response XML File. Please save the file onto the local computer.

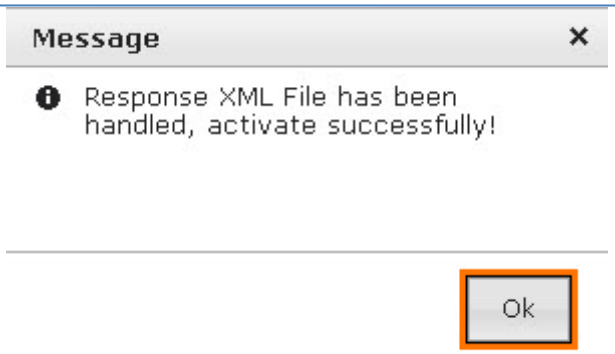


5. Click ***Browse*** to import the Response XML File, and then click ***Activate***. The following message will be shown as follows after the successful activation.

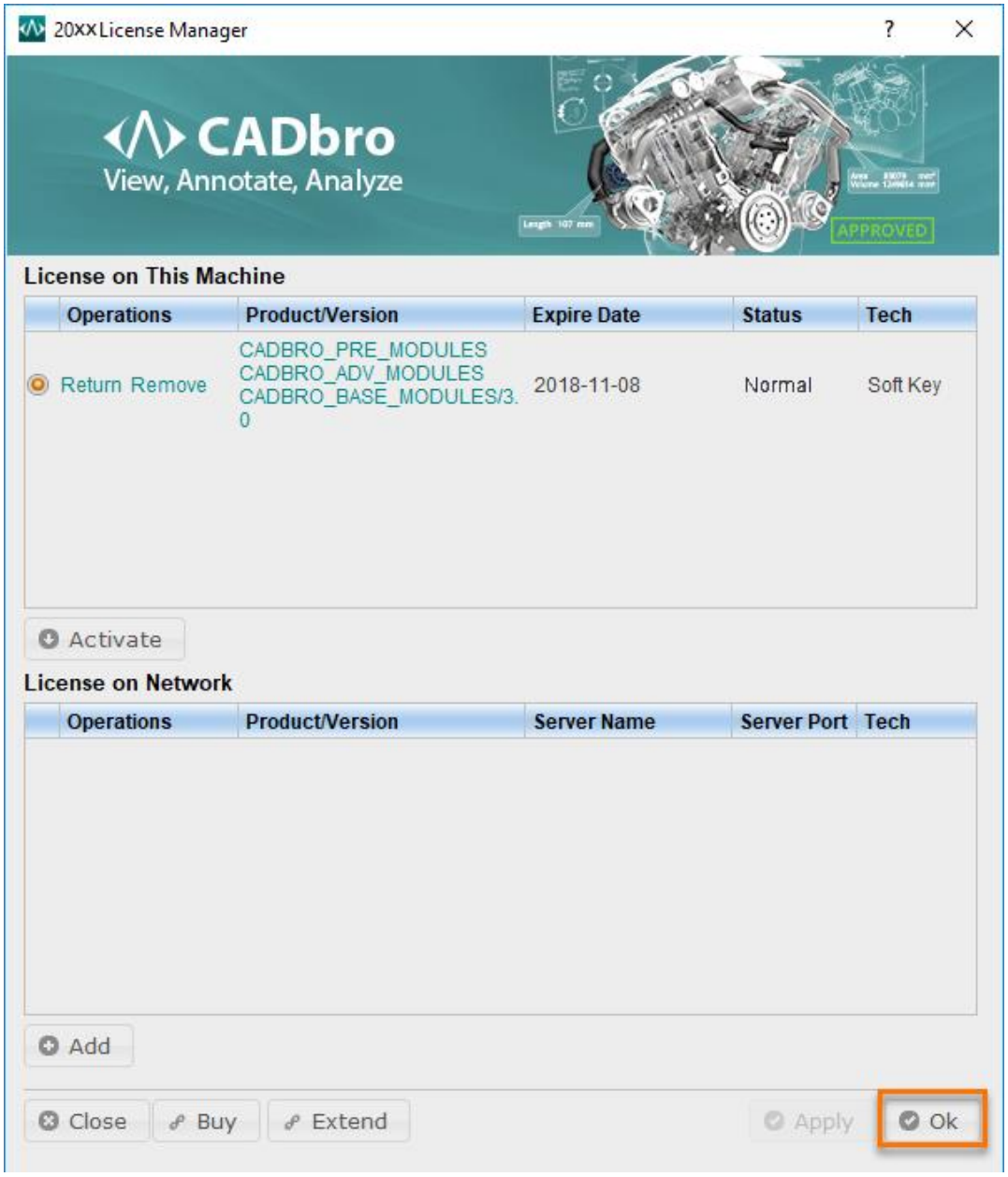




6. Click ***OK*** and back to the license manager.



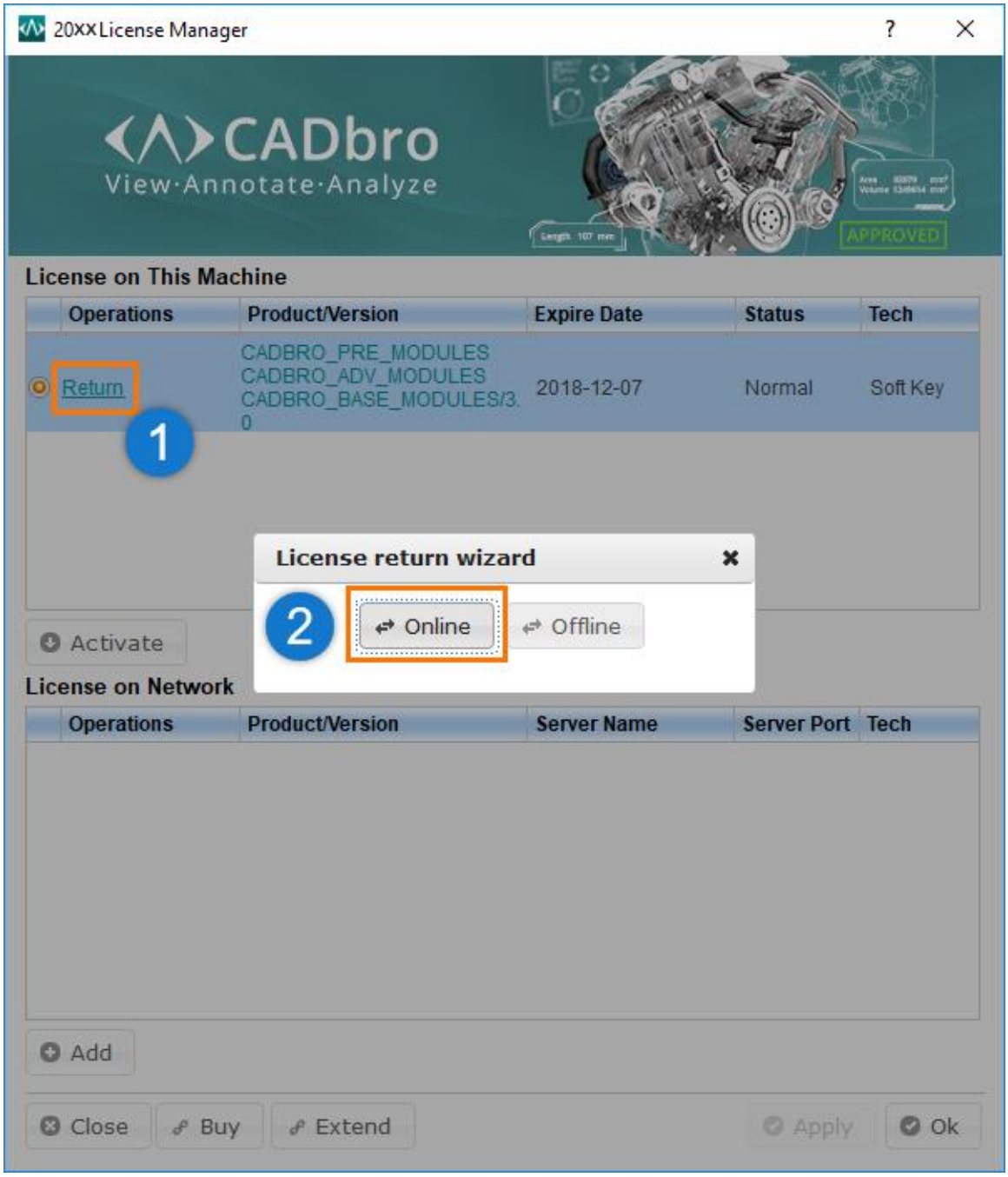
7. Click ***Ok*** to close the license manager dialog box, and ***CADbro*** will be restarted to get the license.



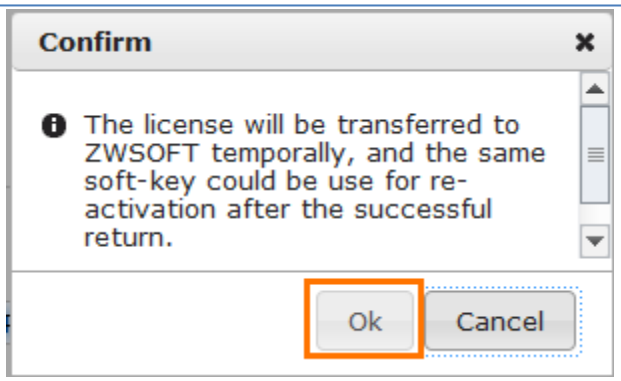
2.1.2 Returning a License

Please follow these steps to return CADbro License **online**:

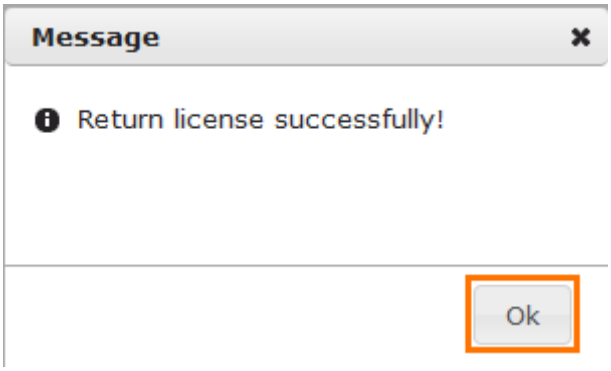
1. Launch ***License Manager*** as follows
   a. On the Windows taskbar, click the ***Start*** button.
   b. Choose ***All Programs*** (or ***Programs***). (Operation system is win7 or win 8 need to choose ***ZWSOFT*** folder first).
   c. Choose ***CADbro20XX*** and then click ***the License Manager***.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



1. Click ***Online*** and the message will be shown. Read the message carefully.



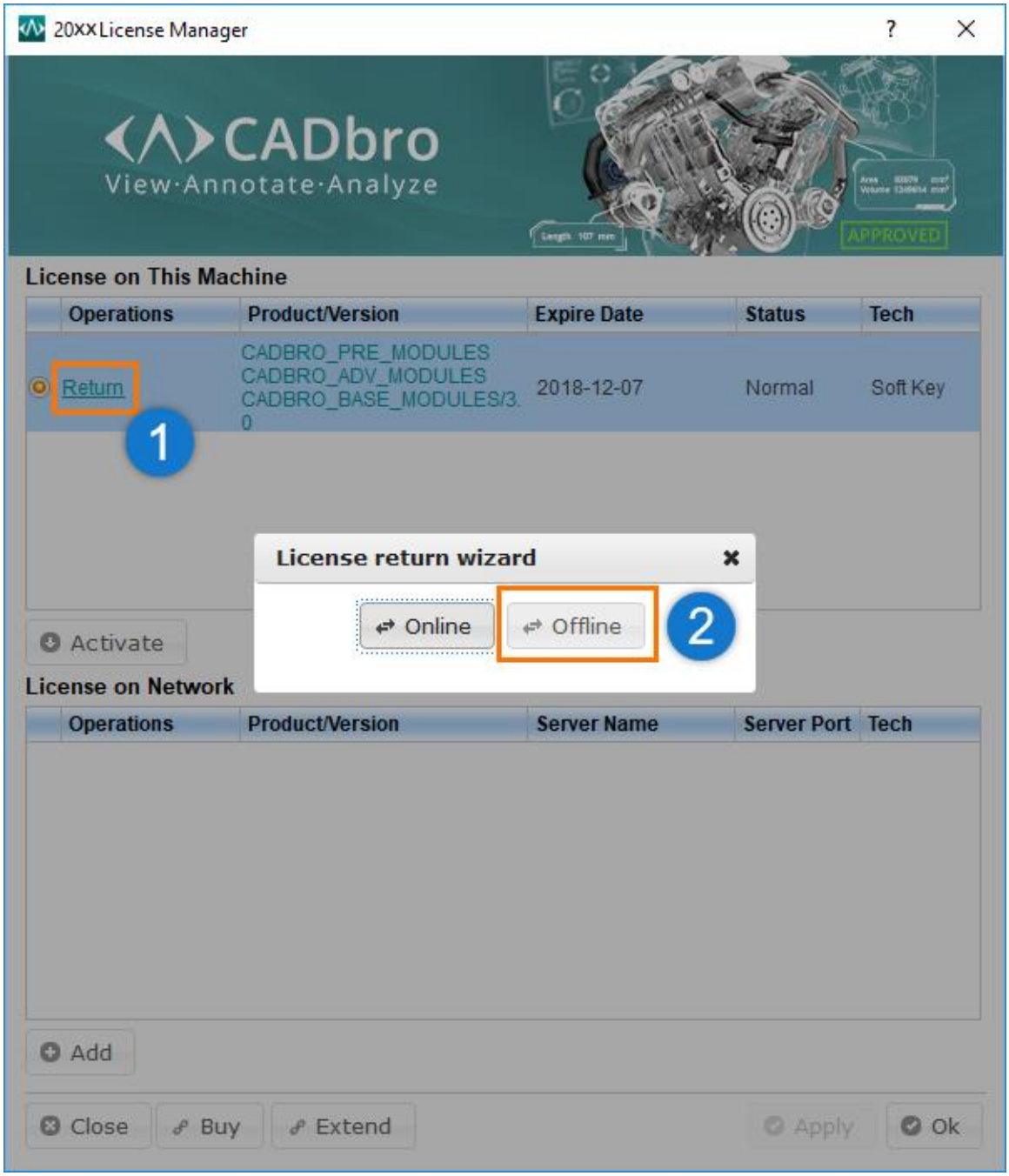
4. Click ***OK*** to return, the license will be successfully returned in a few seconds.



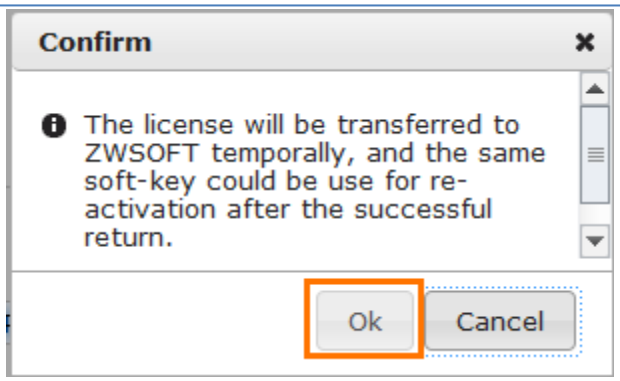
Please follow these steps to return CADbro License **offline**:

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***.

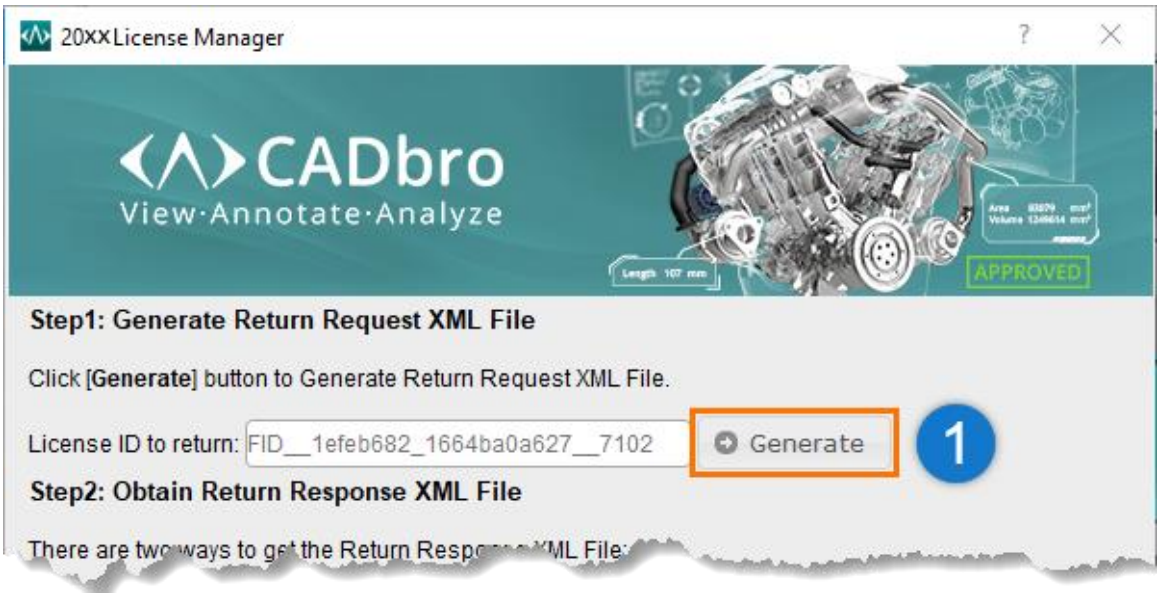
2. Select the license to return, click ***Return***.
3. Click ***Offline*** and the message will be shown. Read the message carefully.

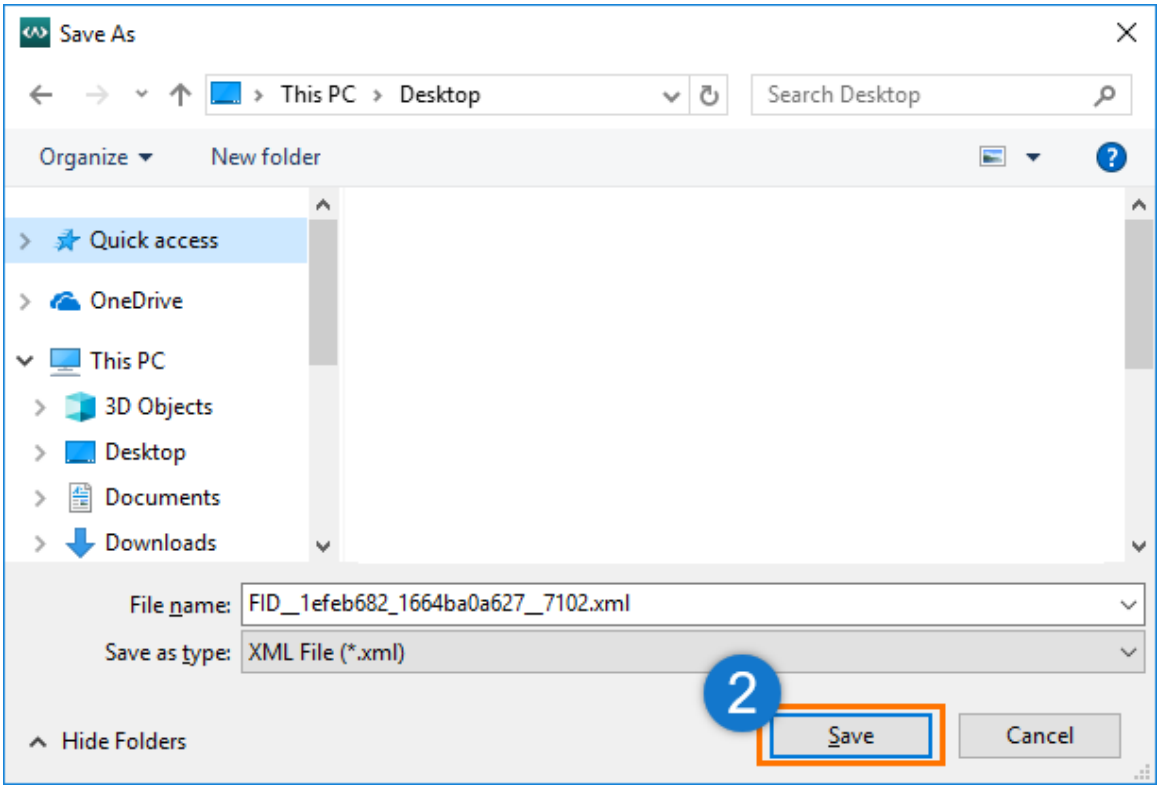


4. Read the message carefully. Follow the steps described in the figure.



5. Press the ***Generate*** button to generate a Request XML File. Save the file onto the local computer.





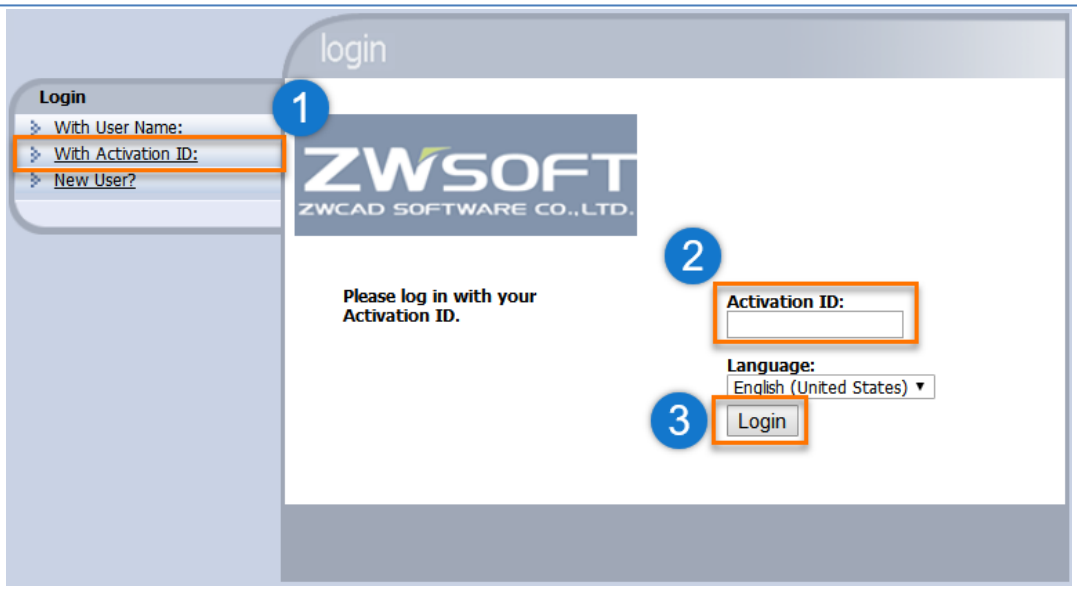
6. Go to another computer with an Internet connection and use the Request XML File to obtain a corresponding Response XML File via E-mail or Online Self-Service

a) Option 1: E-mail

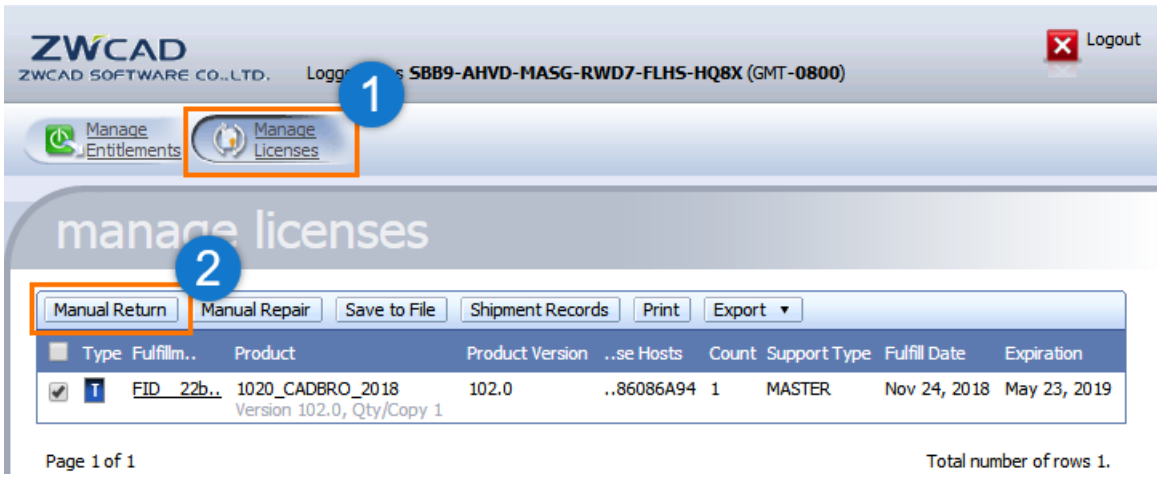
You may send the generated Request XML File to the ZWSOFT License Center at register@zwcad.com. The ZWSOFT License Center will send you the corresponding Response XML File as soon as possible.

b) Option 2: Online Self-Service:

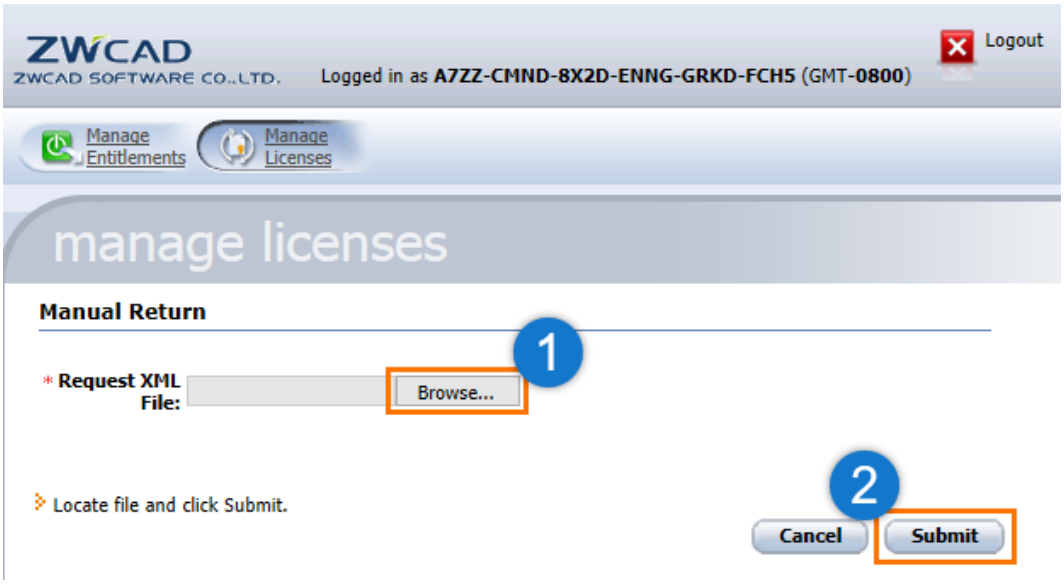
（1）Go to the ZWSOFT License Center at http://support.zwcad.net:8888/flexnet/operationsportal/logon.do and *Login* with the Activation ID (Soft Key).



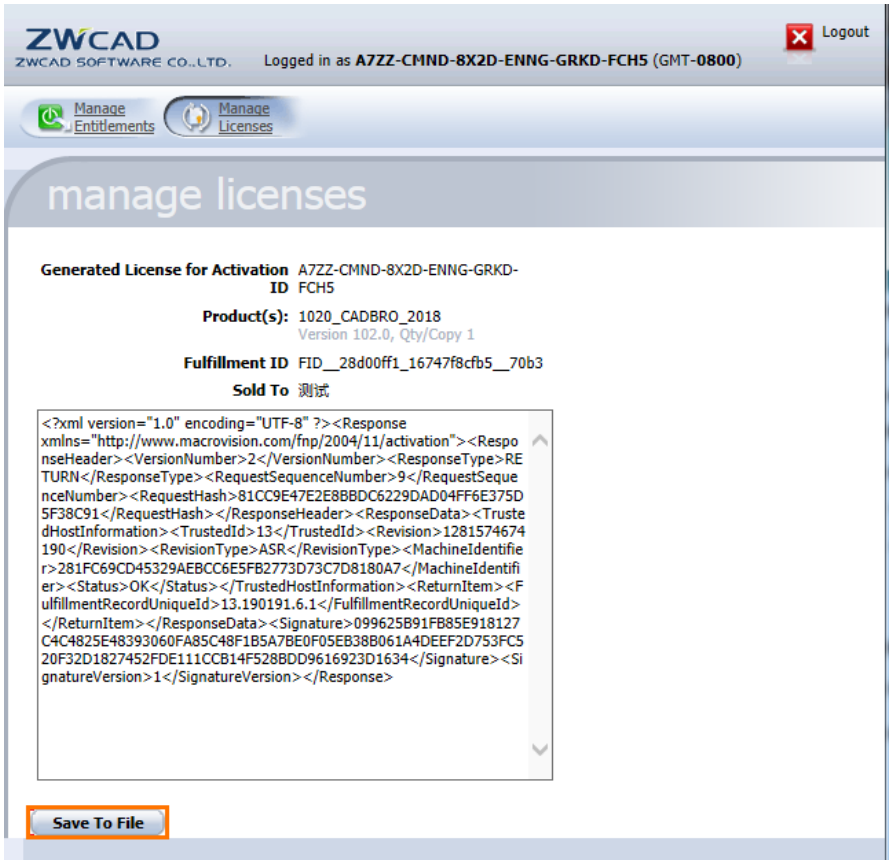
(2) Choose ***Return*** or ***Manage Licenses***, and then click ***Manual Return.***



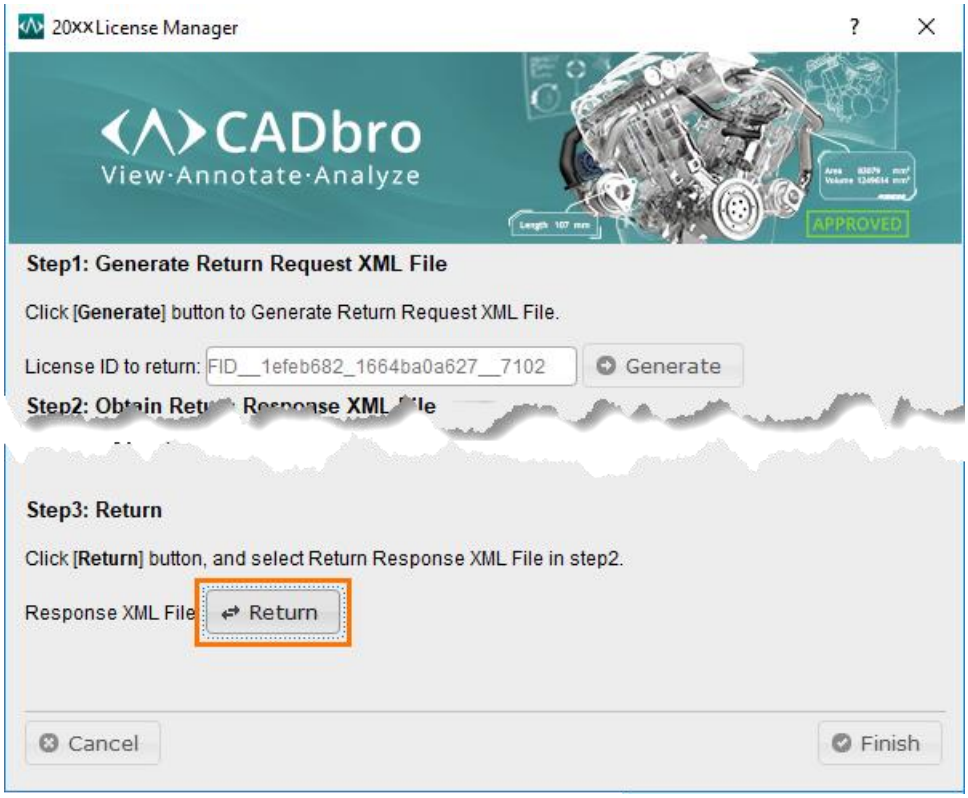
(3) Click ***Browse*** to import your Request XML File, and then click ***Submit***.



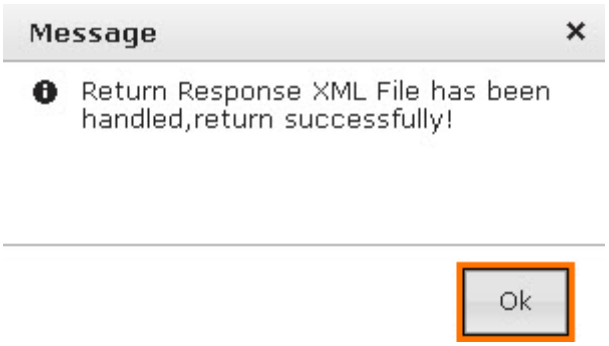
(4) Then you will be able to see the information of the corresponding Response XML File. Please save the file onto the local computer.



7. Press the ***Return*** button to import the Response XML File, and then click ***OK***.



The following message will be shown as follows after the successful return, and the license is disabled in your current computer and available to be activated again.

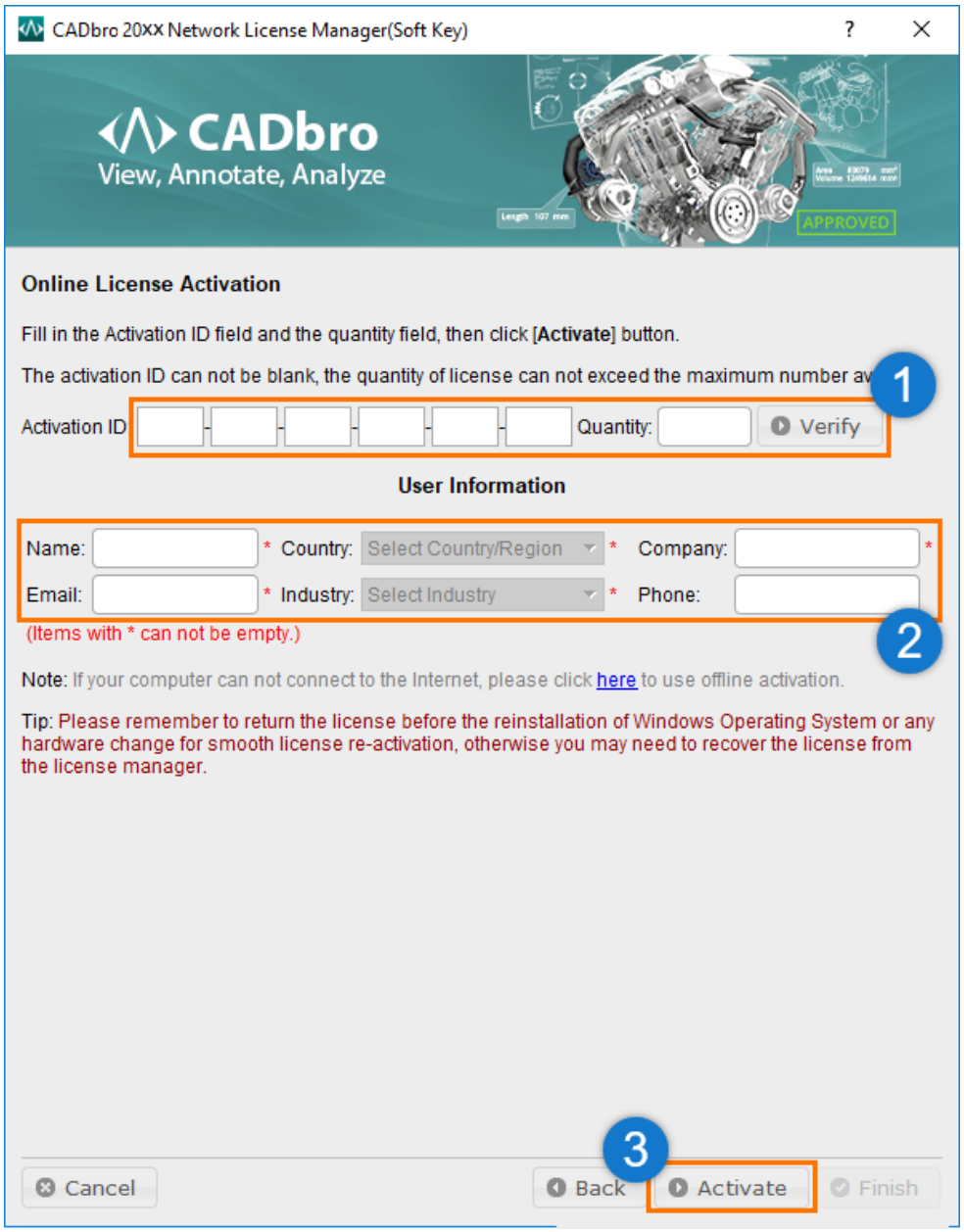


2.2 Network Version

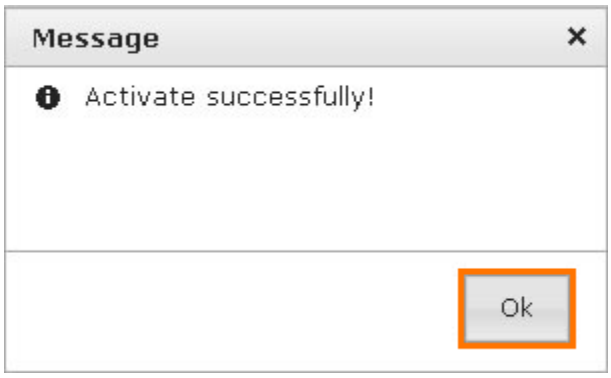
2.2.1 Activation

Network License online:

1. Launch ***Network 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 **CADbro 20XX Network License Manager (Soft-Key)** and then click ***Network License Manager (Soft-Key)***.
2. Enter the Activation ID (Soft Key) and Quantity, and then press the **Verify** button. If this license is used the first time, you should input your information. Click **Activate**, the activation will be successful after a few seconds.



3. Click ***OK*** to close the successful activation message.



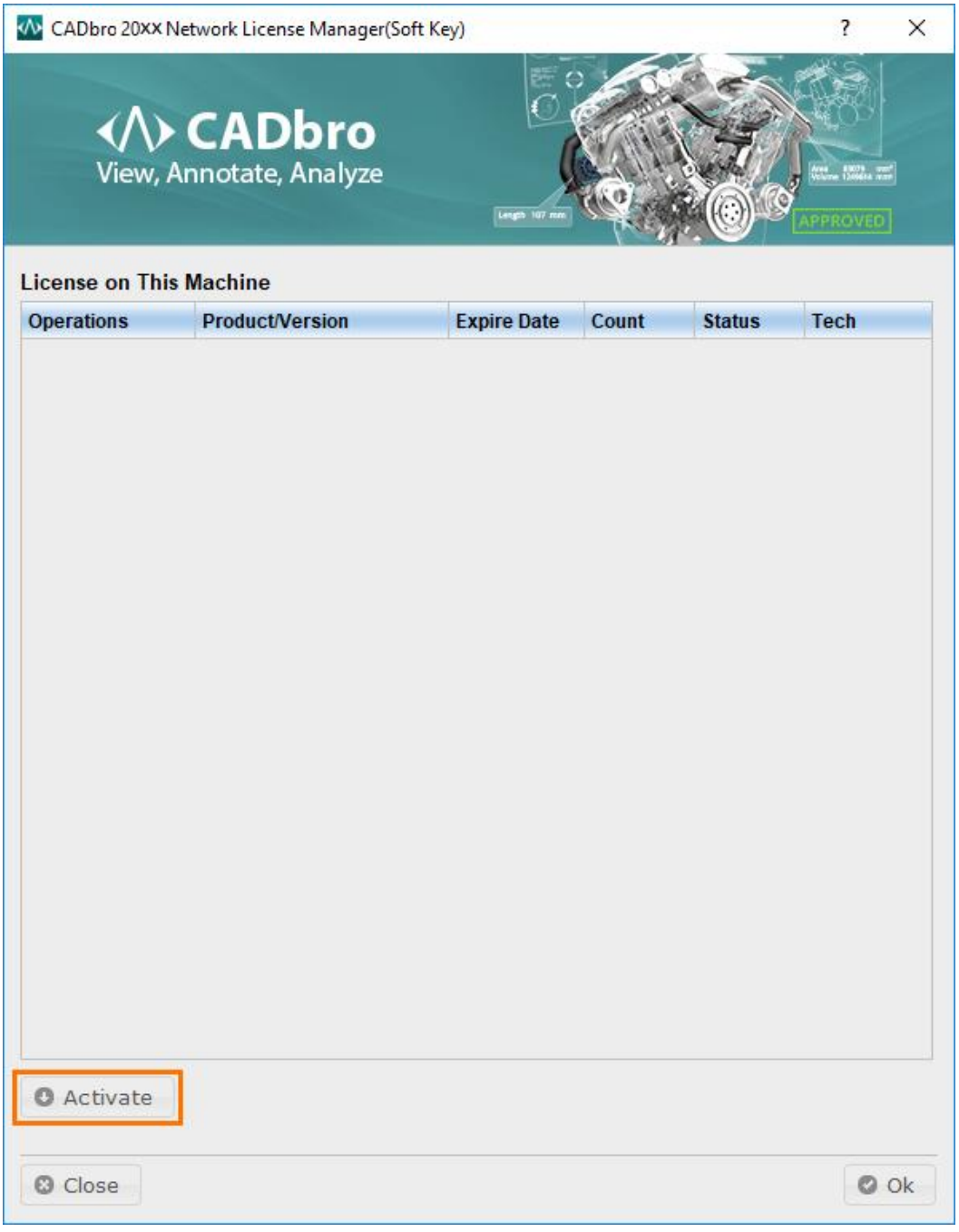
4. Click ***OK*** to close the activation dialog box, and then go to Chapter **Using Network License** to configure the network license in order to get it working.

Please follow these steps to activate CADbro Network License offline:

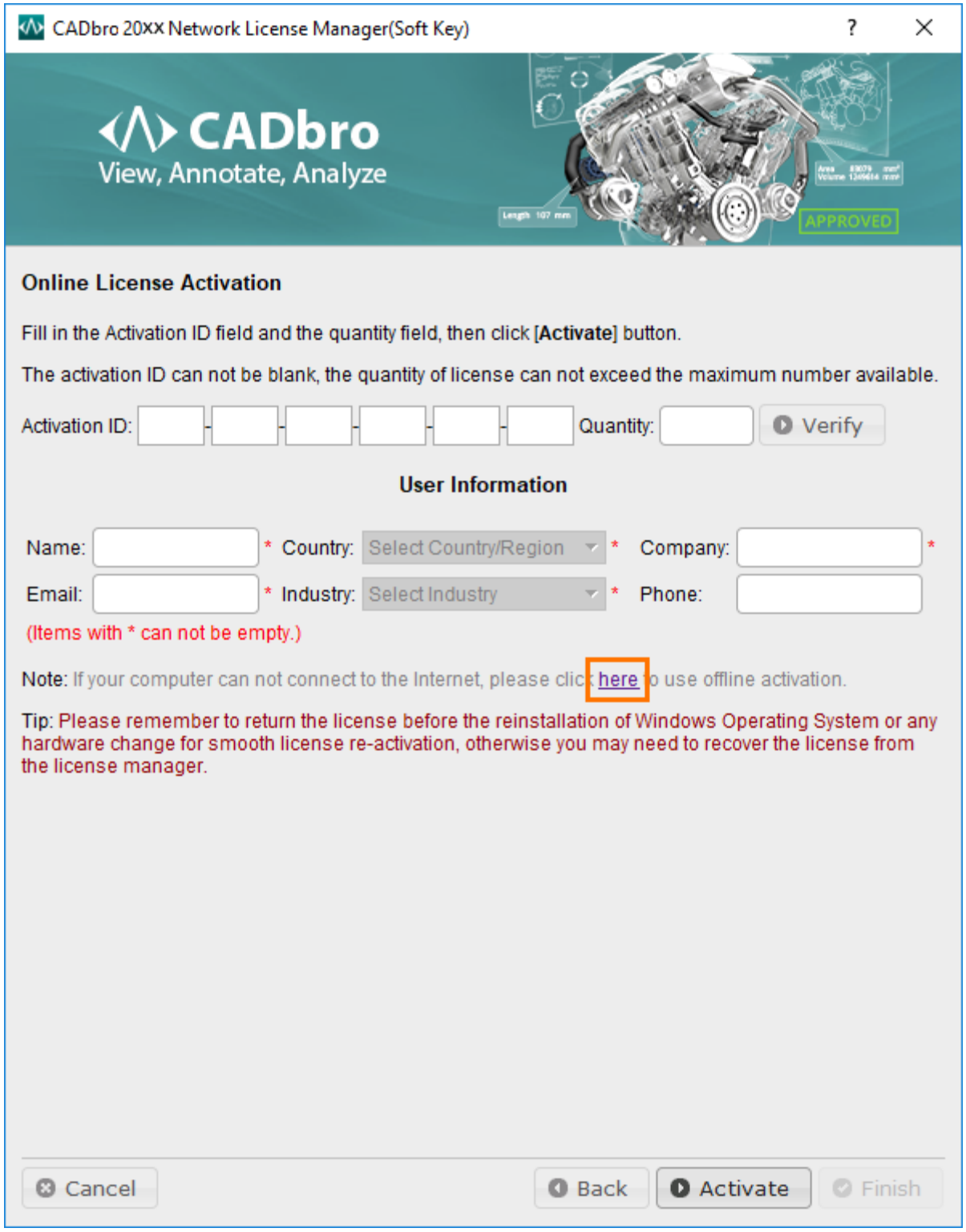
1. Launch ***Network License Manager (Soft-Key)*** as follows

    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 CADbro 20XX Network License Manager (Soft-Key), and then click Network License Manager (Soft-Key).

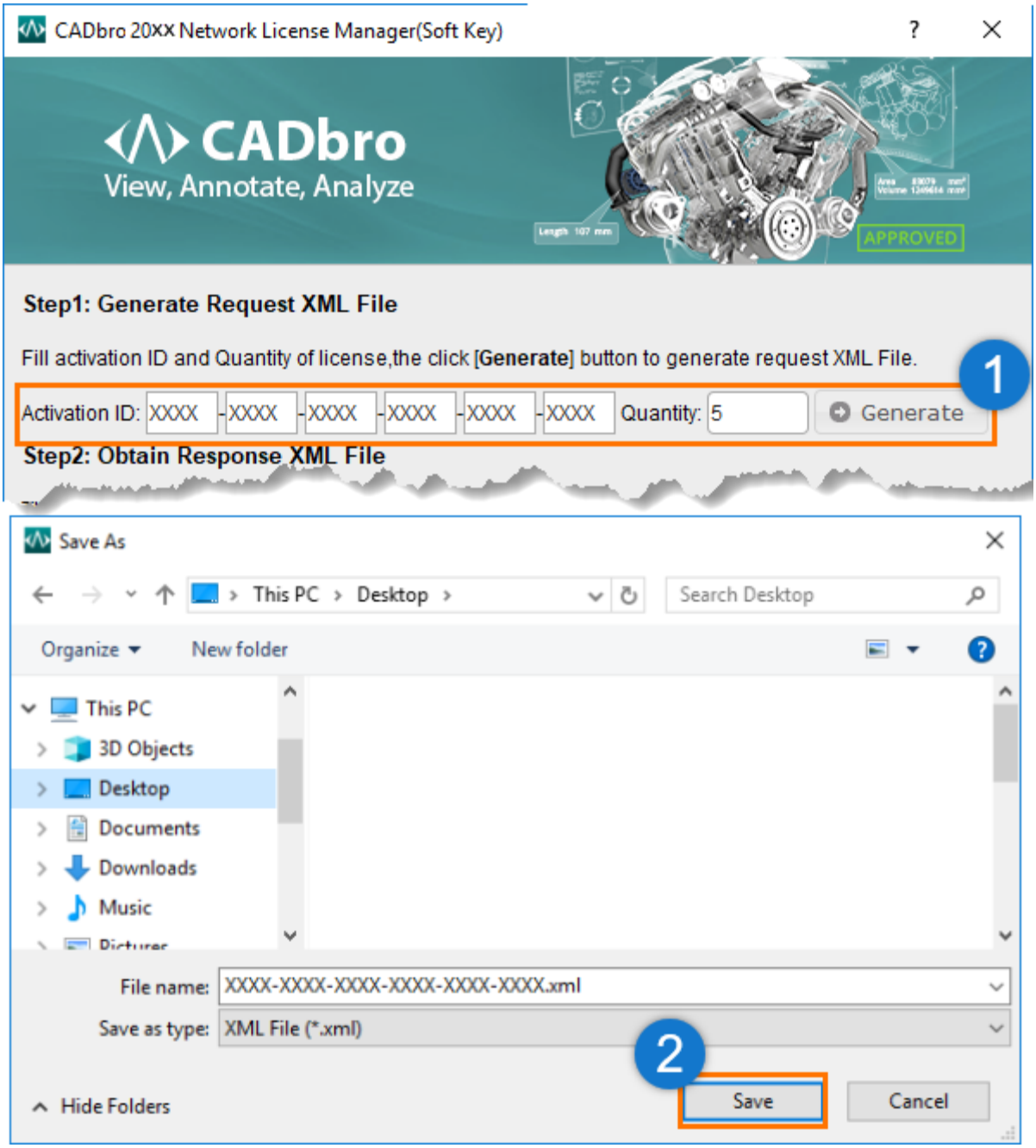
2. Press the ***Activate*** button.



3. In the Online Activation dialog box, click the **here** to go to offline activation dialog.



4. In the Offline Activation dialog box, fill in the Activation ID (Soft Key) and ***Generate*** a Request XML File. Save the file onto the local computer.



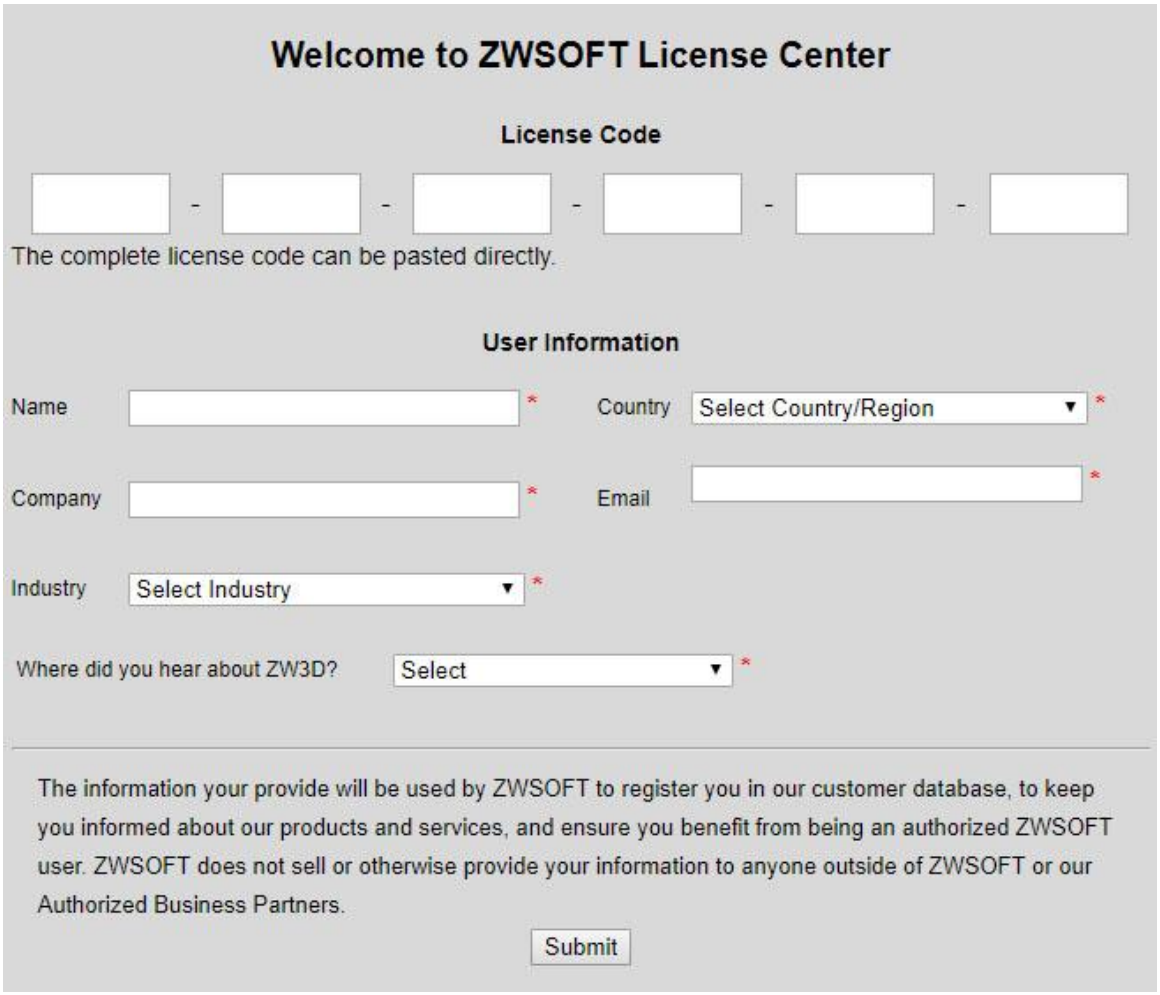
5. Go to another computer with an Internet connection and use the Request XML File to obtain a corresponding Response XML File via E-mail or Online Self-Service

   a) Option 1: E-mail

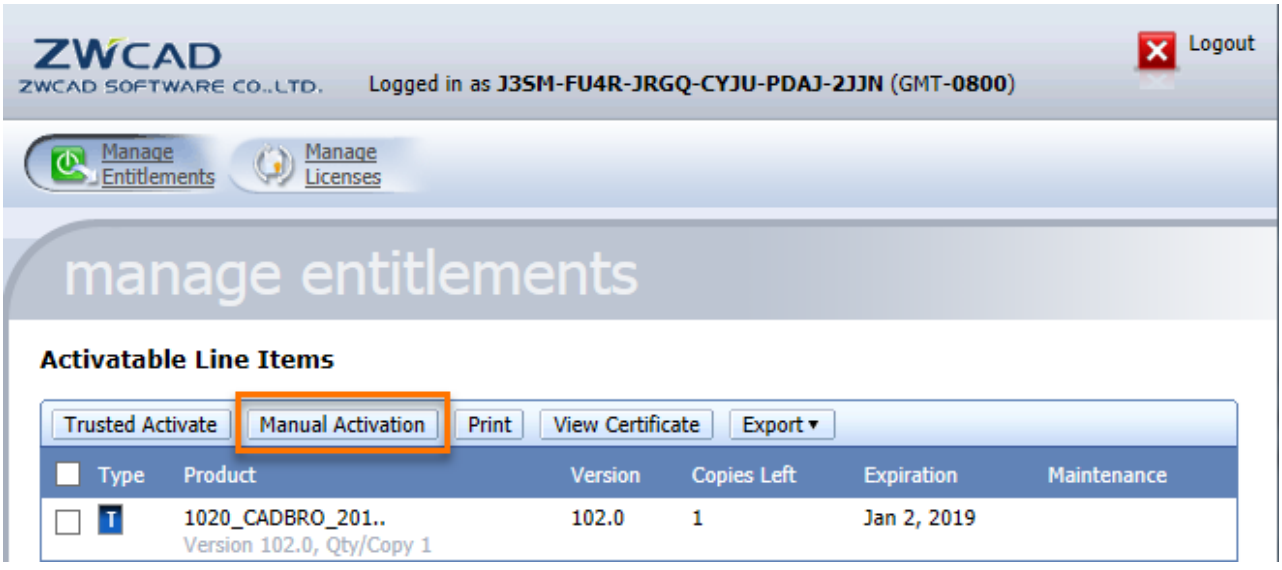
      You may send the generated Request XML File to the ZWSOFT License Center at register@zwcad.com. The ZWSOFT License Center will send you the corresponding Response XML File as soon as possible.

   b) Option 2: Online Self-Service:

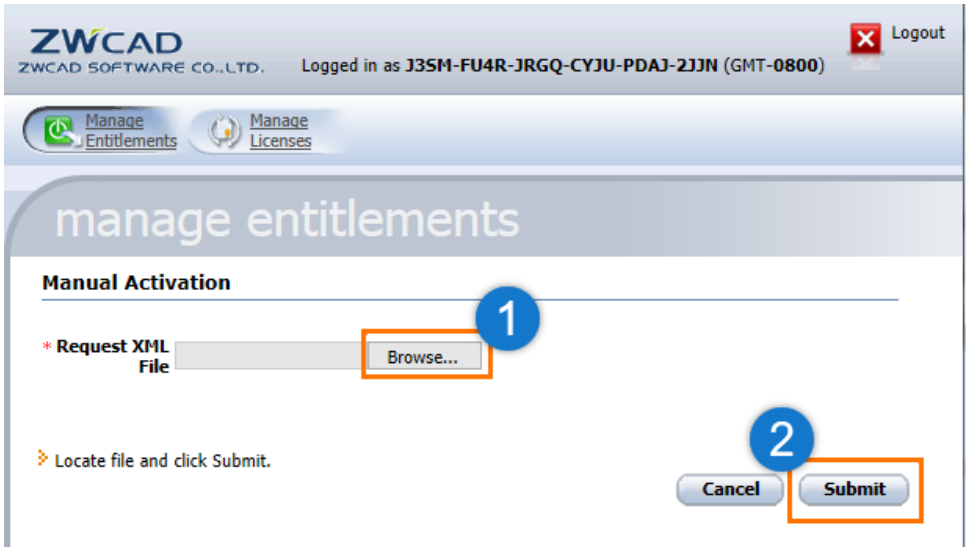
(1) Go to the ZWSOFT License Center at http://license.zwsoft.com/zw3dCustomer.aspx and ***Login*** with the Activation ID (Soft Key).



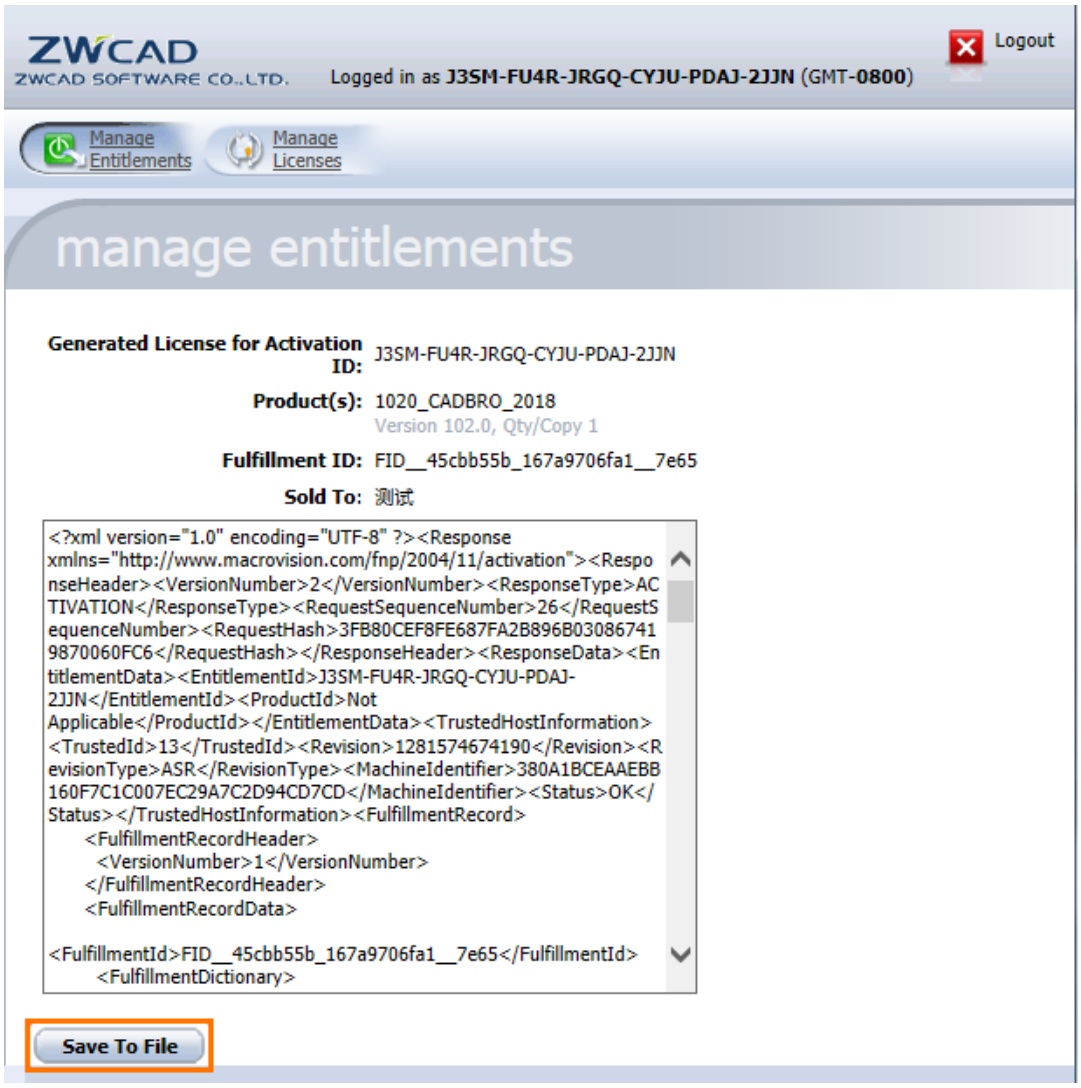
(2) Choose ***Activate*** or ***Manage Entitlements***, and then click ***Manual Activation.***



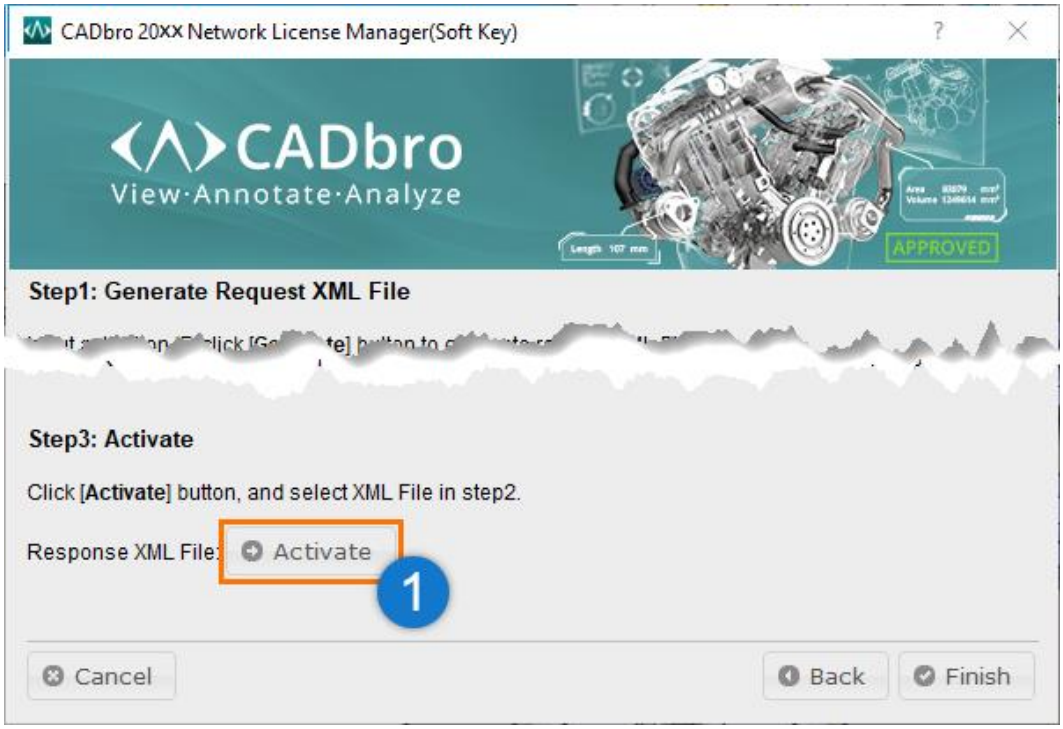
(3) Click ***Browse*** to import your Request XML File, and then click ***Submit***.

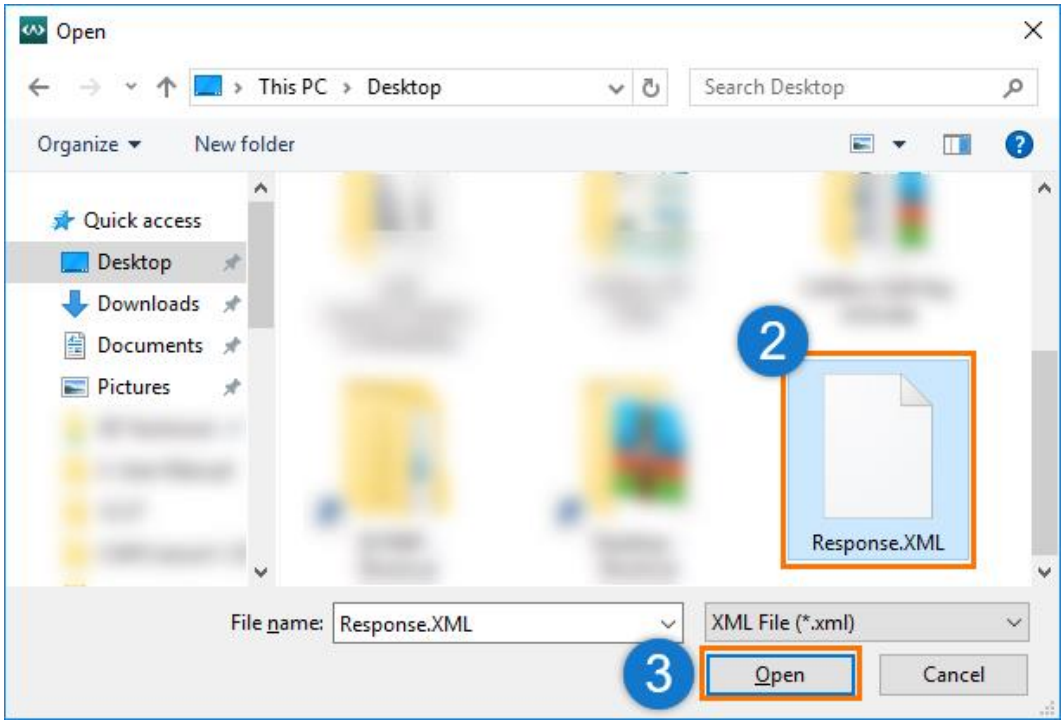


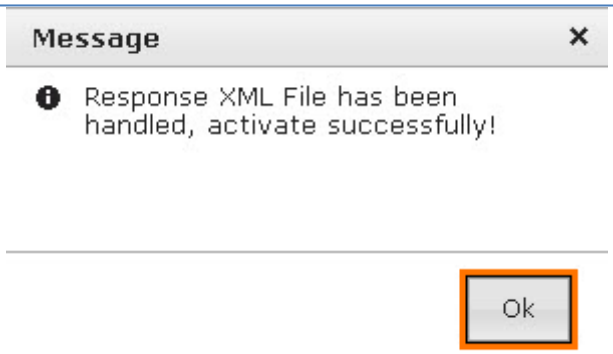
(4) Then you will be able to see the information of the corresponding Response XML File. Please save the file onto the local computer.



6. Press the Activate button to import the Response XML File, and then click OK. The following message will be shown as follows after the successful activation.







2.2.2 Using Network License

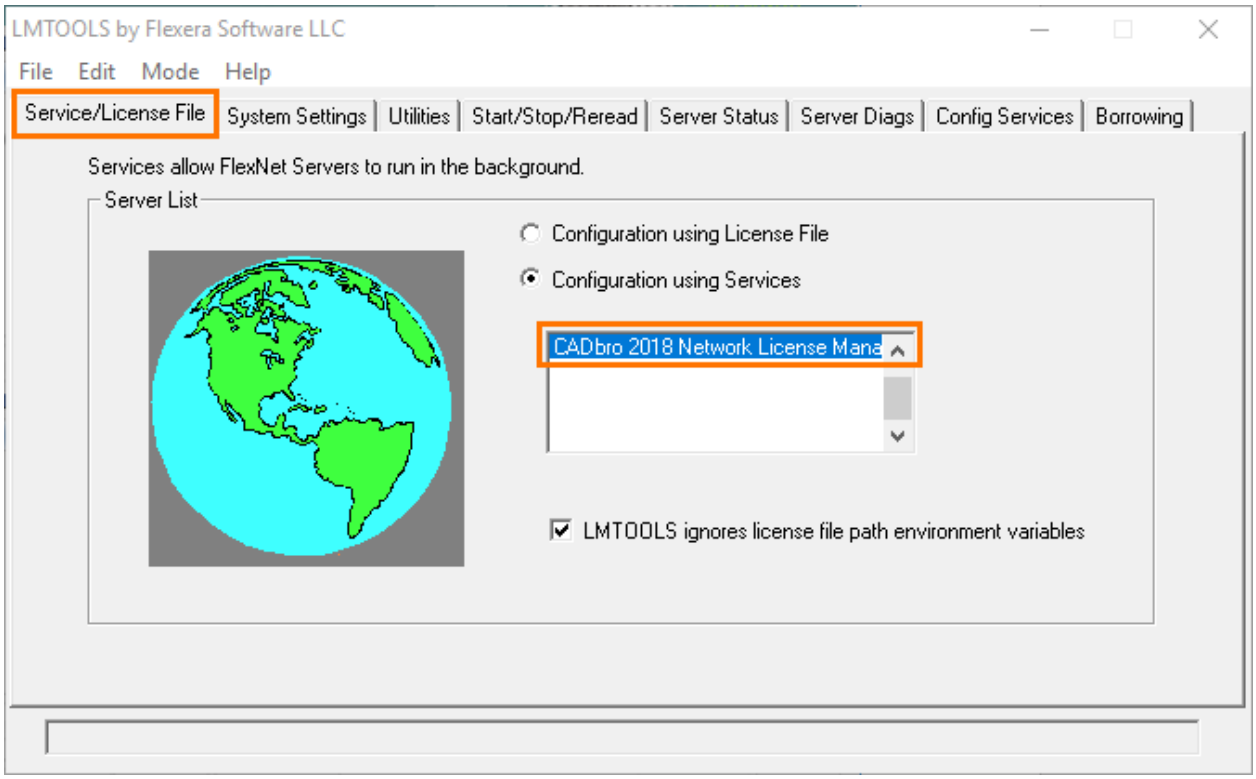
Configurations should be performed on both the Server and Workstation Computers.

•Configuring the Server

1. Launch ***Network License Configuration (Soft-Key)*** 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 ***CADbro 20XX Network License Manager (Soft-Key),*** and then click ***Network License Configuration (Soft-Key).***

2. In the LMTools program, select the Service/License File tab, and then choose the *Configuration Using Services* option.



3. Click the ***Config Services*** tab.
4. Select a service name from the Service Name list, or follow one of the options below:

   •**If a service name has been selected** -- check whether this is the one you would like to use for managing licenses.
   •**If no service name exists** -- please enter a service name to use for managing licenses.

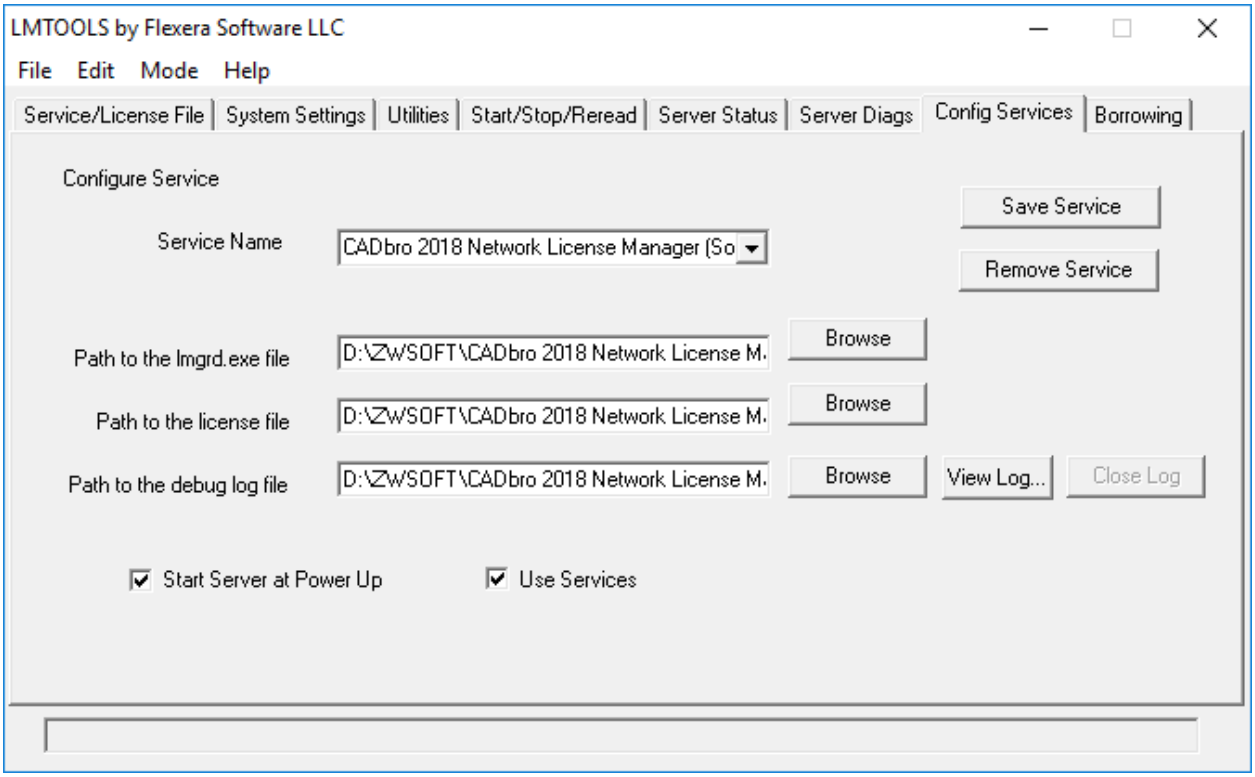
5. In the Path to Lmgrd.exe File box, enter the path to the Network License Manager daemon (***lmgrd.exe***), or click ***Browse*** to locate.

   Note: The ***lmgrd.exe*** program is located by default in the installation folder of CADbro 20XX Network License Manager.

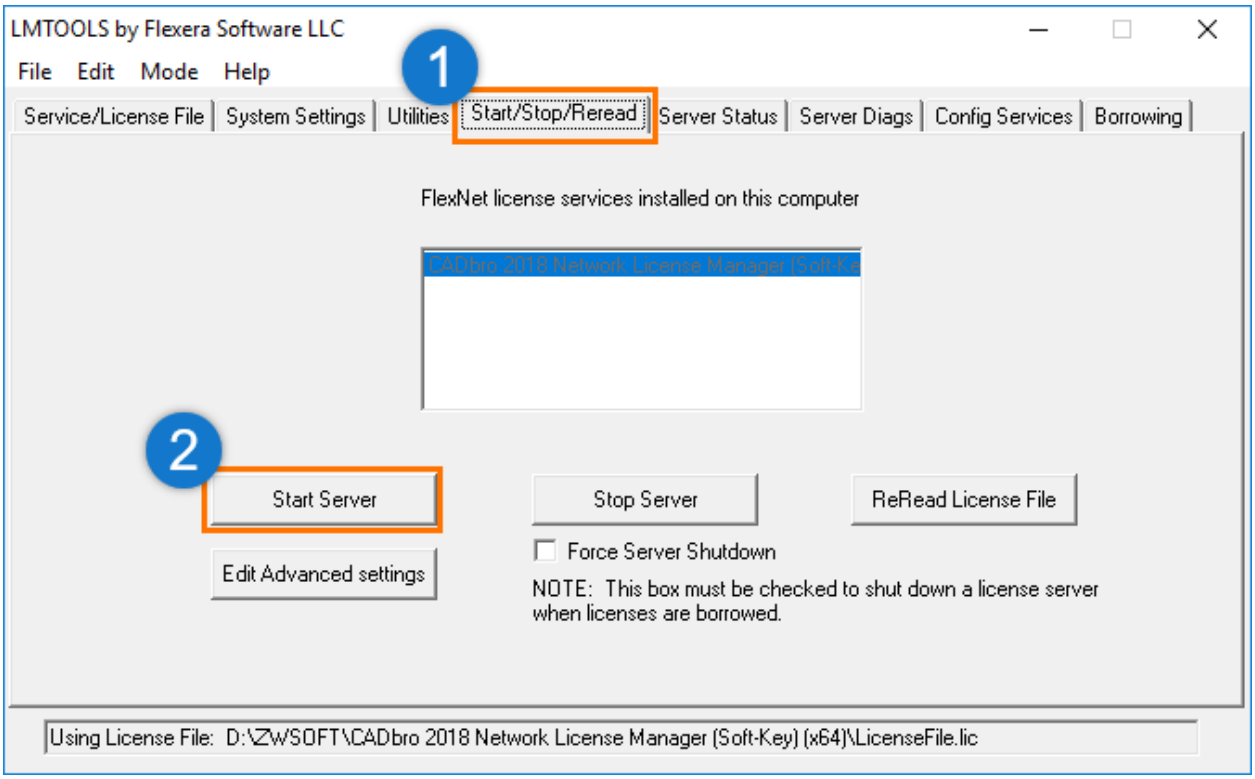
6. In the Path to the License File box, enter the path to your license file (***LicenseFile.lic)***, or click ***Browse*** to locate the file.

   Note: The ***LicenseFile.lic*** file is located by default in the installation folder of CADbro 20XX Network License Manager.

7. In the Path to the Debug Log File box, enter a path to store the debug log file, or click ***Browse*** to locate an existing log file.
8. Check ***Use Services*** to run the ***lmgrd.exe*** application as a service.
9. Check ***Start Server at Power Up*** to start ***lmgrd.exe*** automatically when the system starts.
10. Click ***Save Service*** to save the new configuration under the service name specified in the procedure (4). Then click ***Yes*** to confirm.

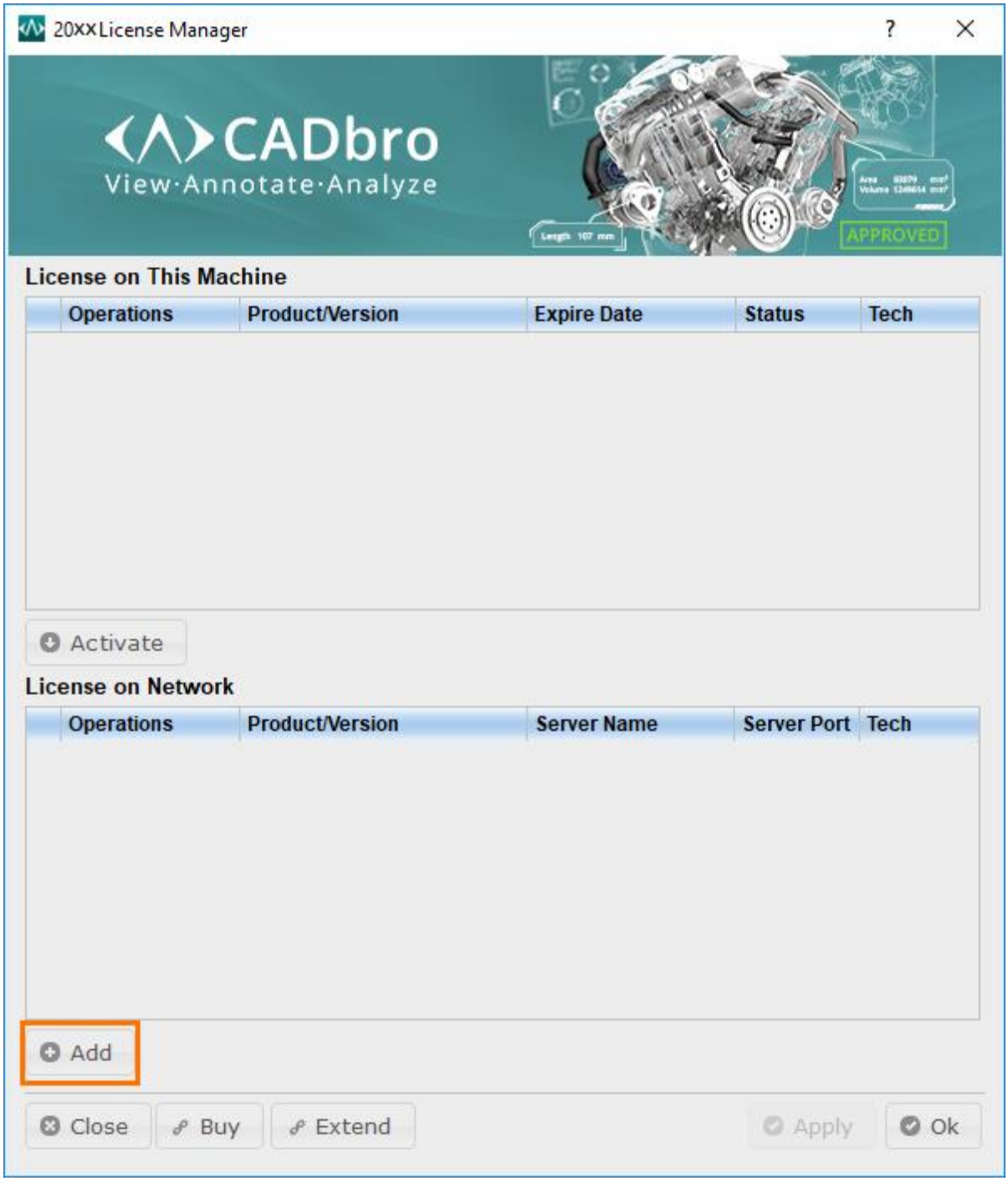


11. Click the Start/Stop/Reread tab. Make sure that the service saved in previous steps is selected, and then click ***Start Server***. Then the Server will begin serving.

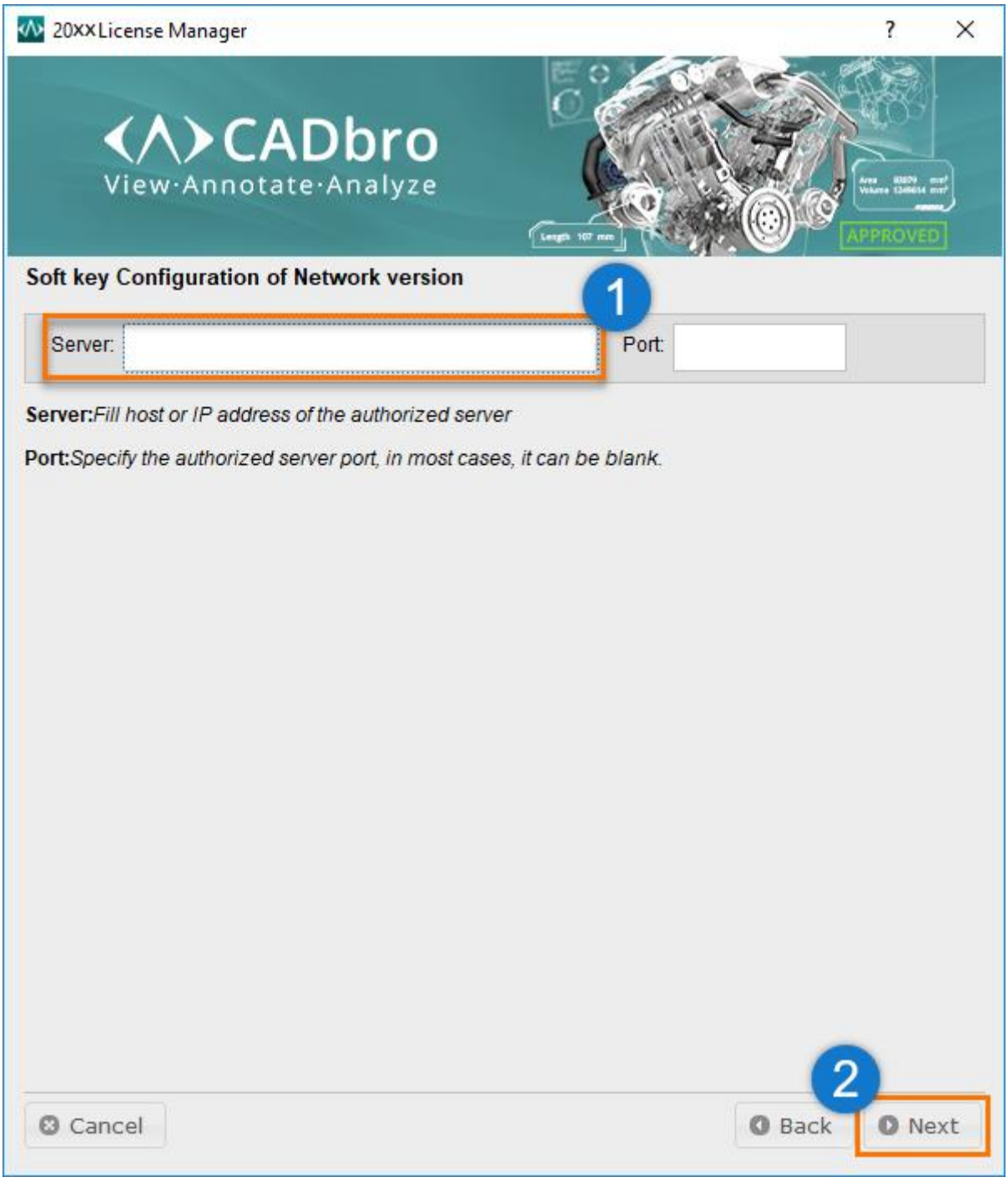


•Configuring Workstations

1. On the desktop, double-click the ***CADbro 20XX*** icon to launch CADbro.
2. Click the ***Add*** button on the License Manager page.

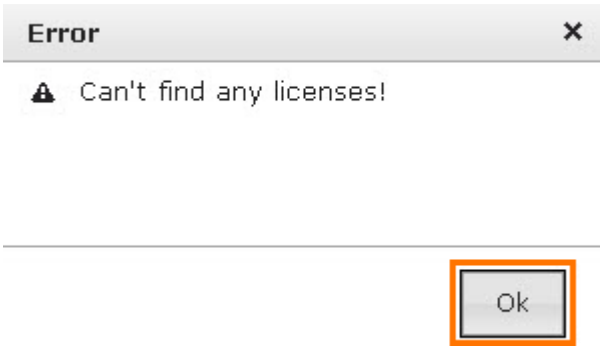


3. Fill in the ***Server*** Name (or IP).



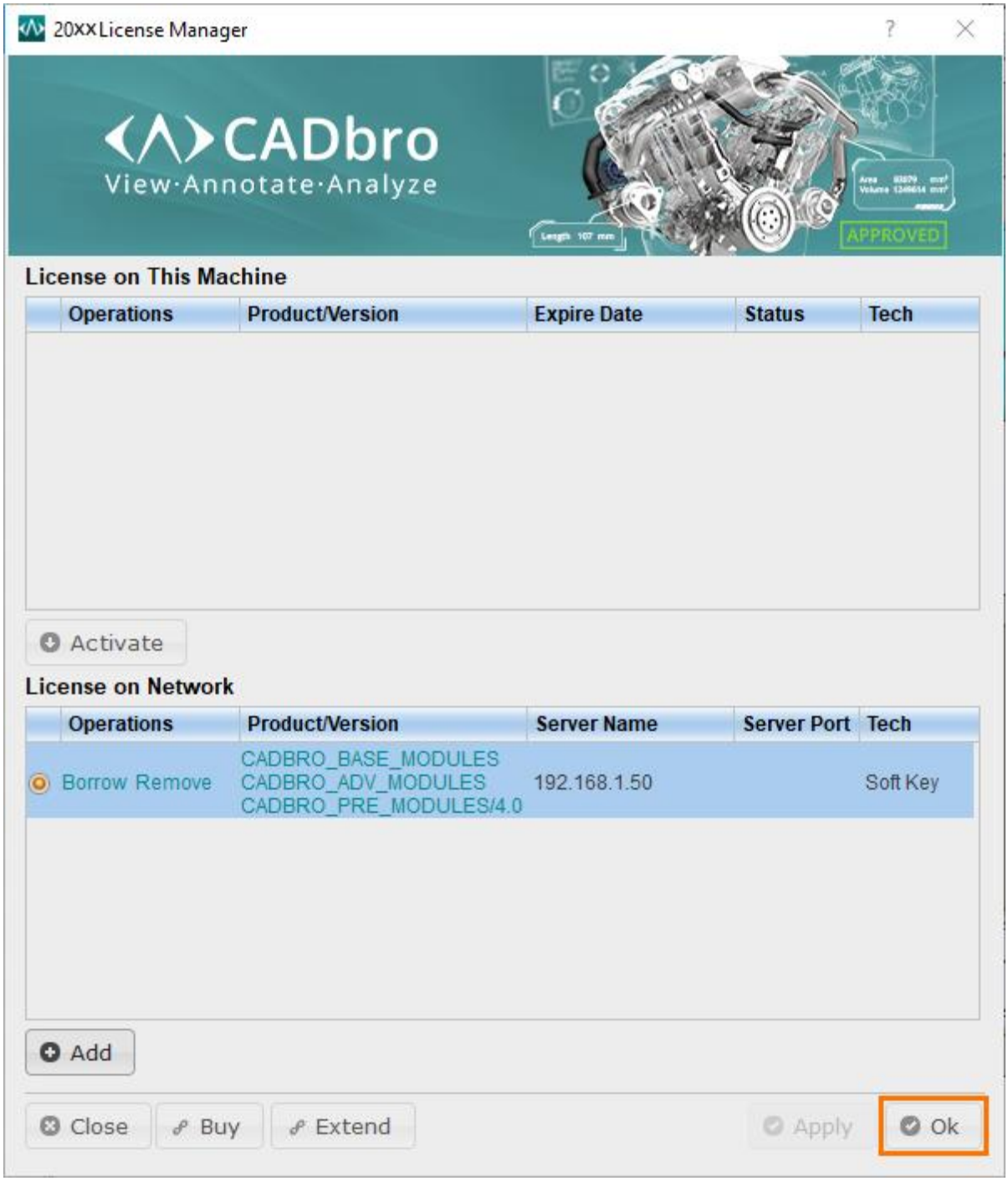
4. Click ***Next*** button

   a) if the Authorized Server you connect is not available, the message will be shown as follows, you need to confirm the connection and retry again:



b) If the Authorized Server you connect is available, the existing license list will be shown if the connection succeeds, and then you can pick some of them.

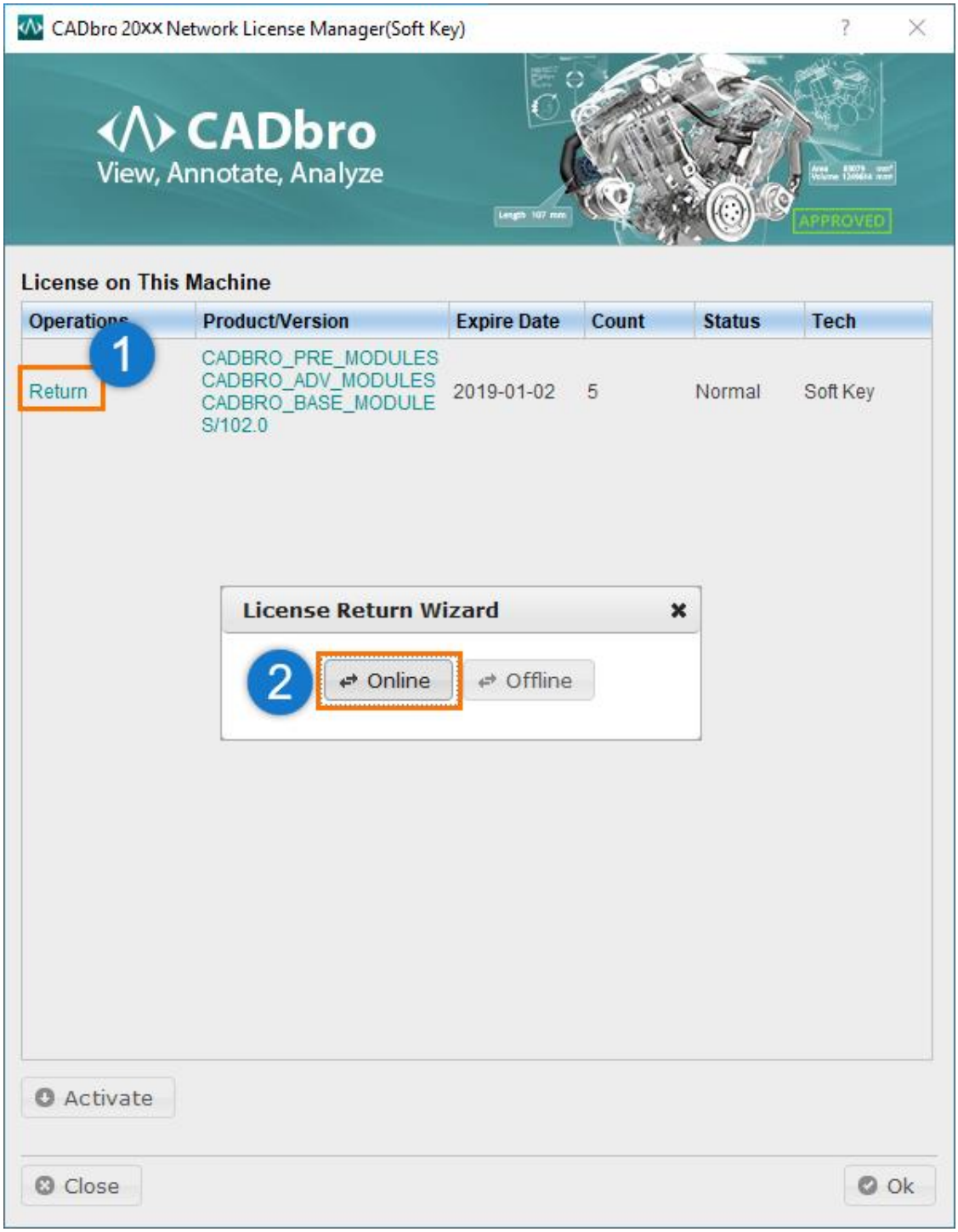
5. Click ***Finish*** to return to License Manager Page, you will see a record has been added to the list.
6. Make sure the dot has been added to the server list, otherwise you need to pick it and click ***Apply*** button.
7. Click ***OK*** to close the license manager, and then double click ***CADbro 20XX*** icon on the desktop to run CADbro.



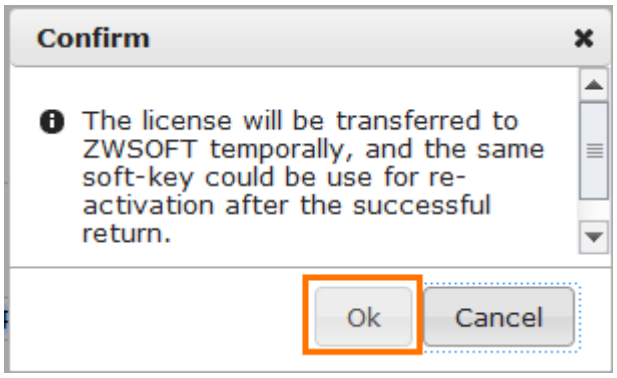
2.2.3 Returning Network License

Please follow these steps to return CADbro License online:

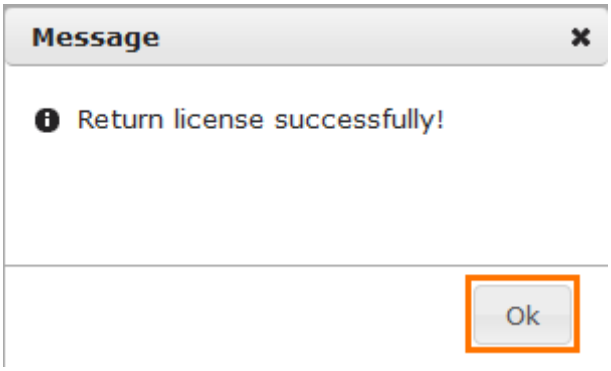
1. Launch ***Network License Manager*** as follows
   a. On the Windows taskbar, click the ***Start*** button.
   b. Choose ***All Programs*** (or ***Programs***). (Operation system is win7 or win 8 need to choose ***ZWSOFT*** folder first).
   c. Choose **CADbro 20XX Network License Manager (Soft-Key)** and then click ***Network License Manager (Soft-Key)***.
2. Select the license to return, click Return.



3. Read the message carefully and click ***OK*** to return the license.



4. Click Online to return, the license will be successfully returned in a few seconds.

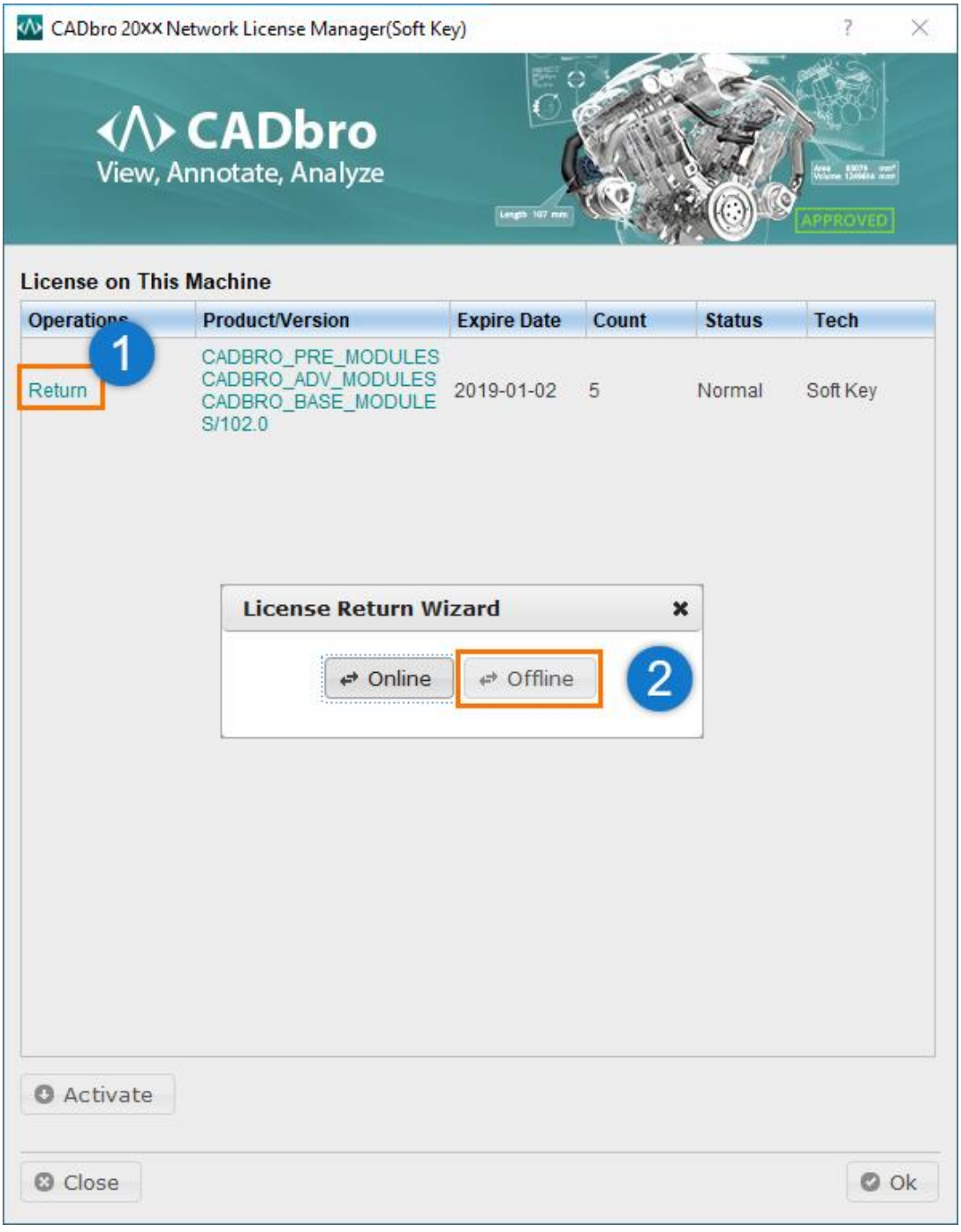


Please follow these steps to return CADbro License offline:

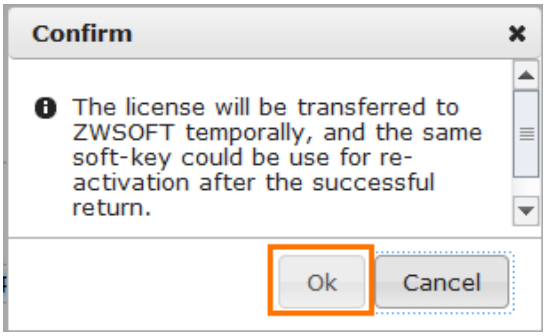
1. Launch ***Network License Manager*** as follows
   a. On the Windows taskbar, click the ***Start*** button.
   b. Choose ***All Programs*** (or ***Programs***). (Operation system is win7 or win 8 need to choose ***ZWSOFT*** folder first).
   c. Choose **CADbro 20XX Network License Manager (Soft-Key)** and then click ***Network License Manager (Soft-Key)***.

2. Select the license to return, click Return.

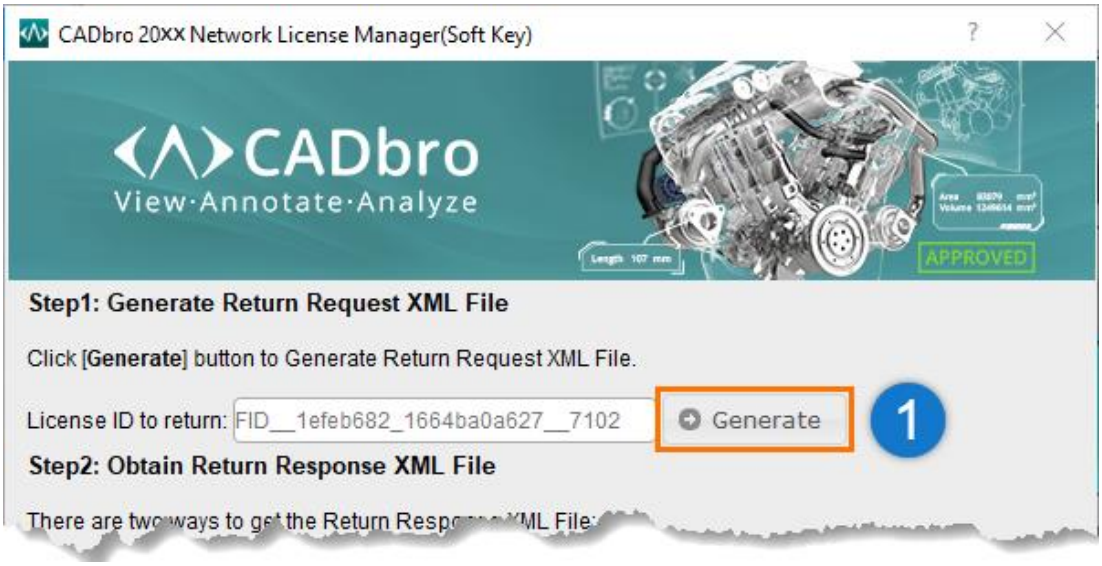
3. Click ***Offline*** to return and the following message will be shown.

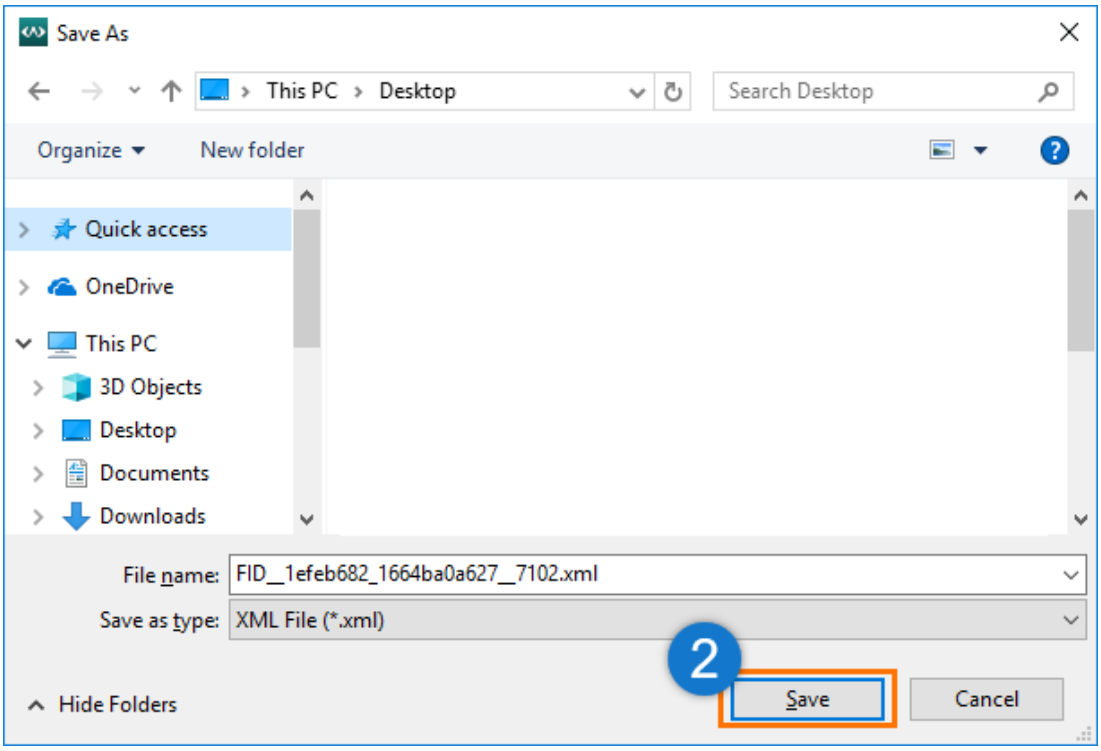


4. Read the message carefully and click ***OK*** to return the license.



5. Press the ***Generate*** button to generate a Request XML File. Save the file onto the local computer.





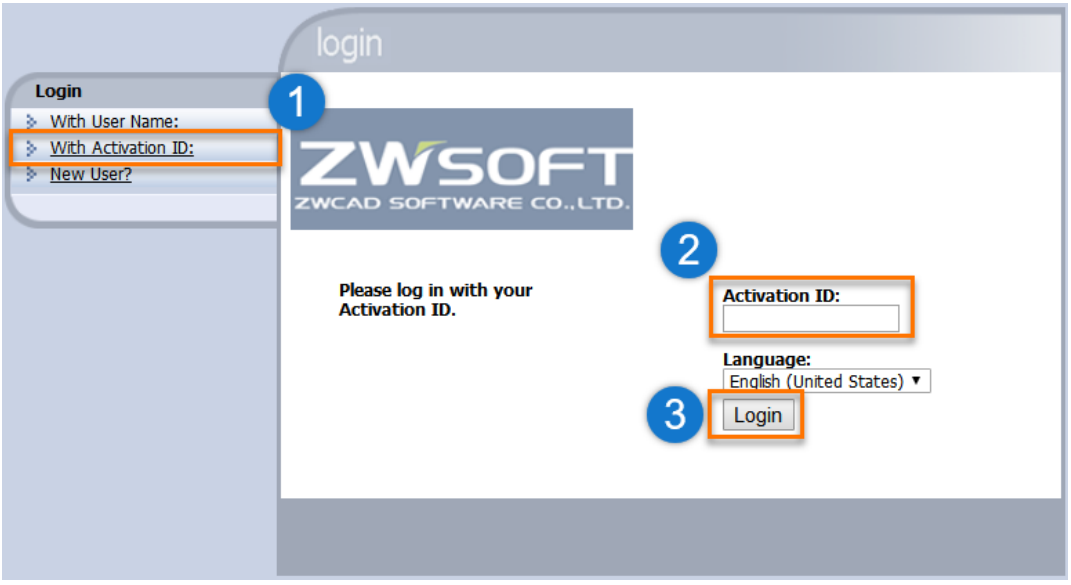
6. Go to another computer with an Internet connection and use the Request XML File to obtain a corresponding Response XML File via E-mail or Online Self-Service

a) Option 1: E-mail

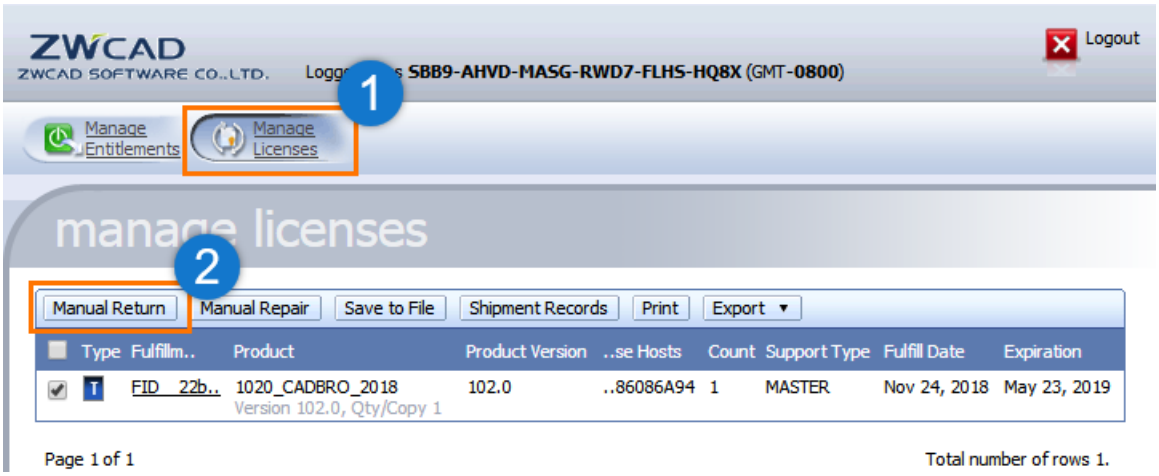
You may send the generated Request XML File to the ZWSOFT License Center at register@zwcad.com. The ZWSOFT License Center will send you the corresponding Response XML File as soon as possible.

b) Option 2: Online Self-Service:

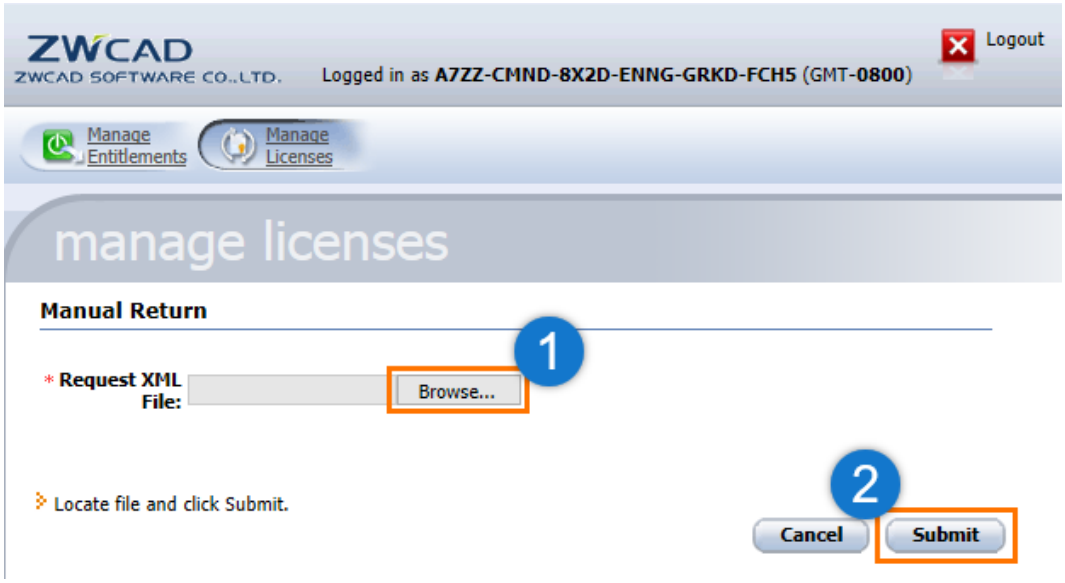
(1) Go to the ZWSOFT License Center at http://support.zwcad.net:8888/flexnet/operationsportal/logon.do and Login with the Activation ID (Soft Key).



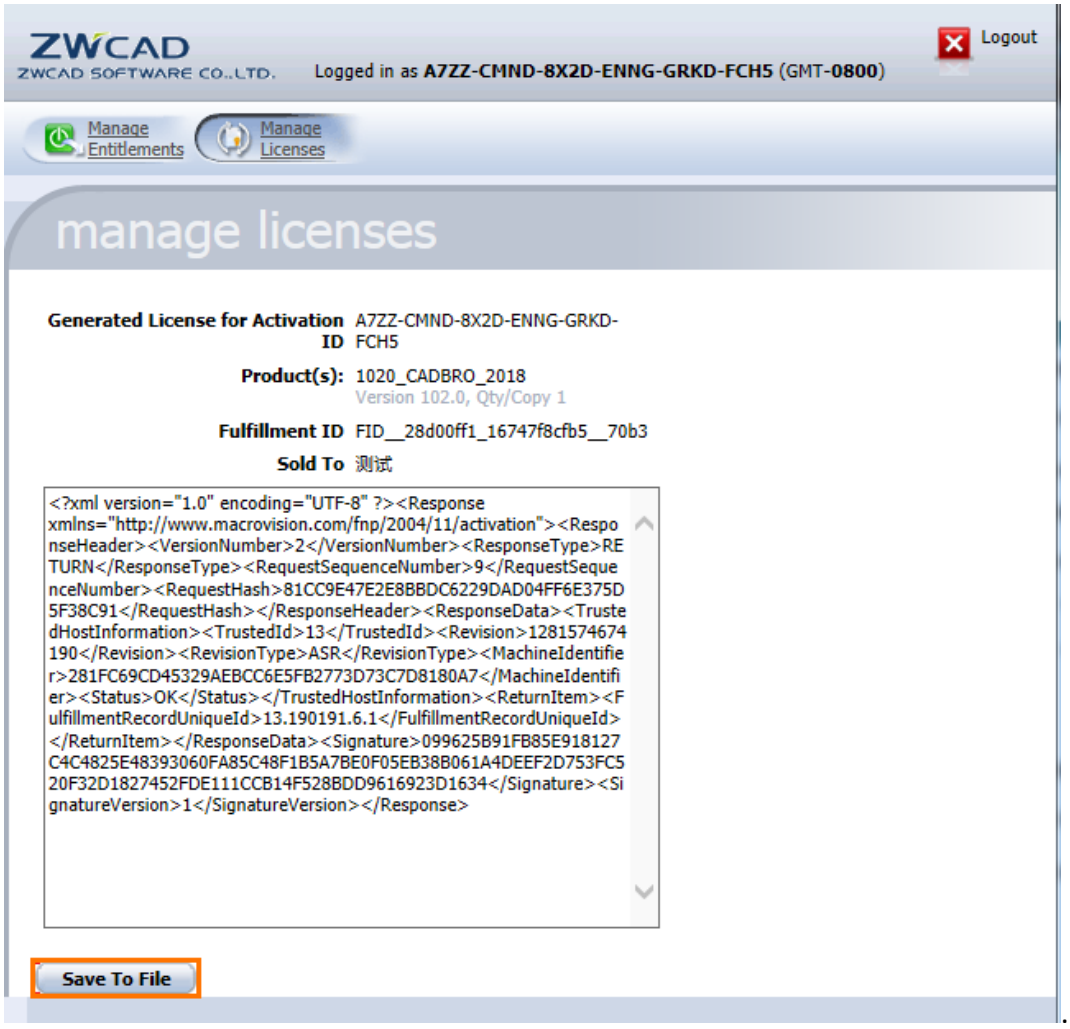
(2) Choose Return or Manage Licenses, and then click Manual Return.



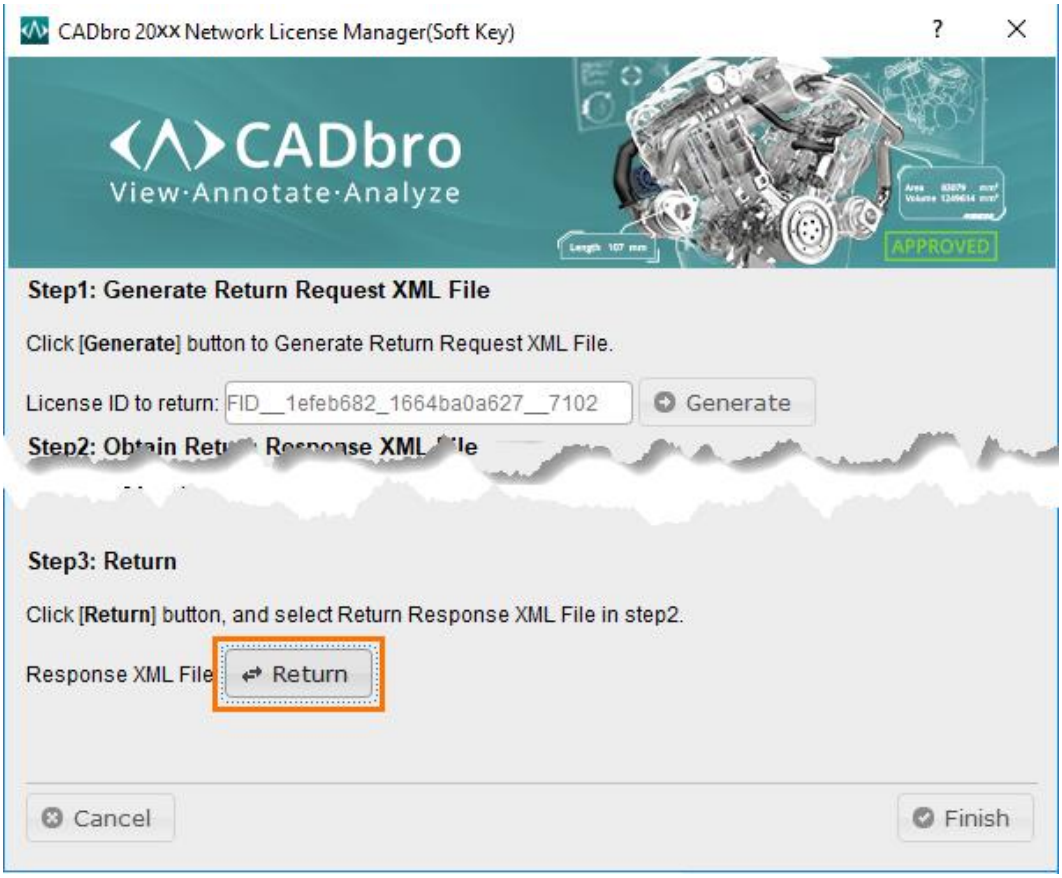
(3) Click Browse to import your Request XML File, and then click Submit.



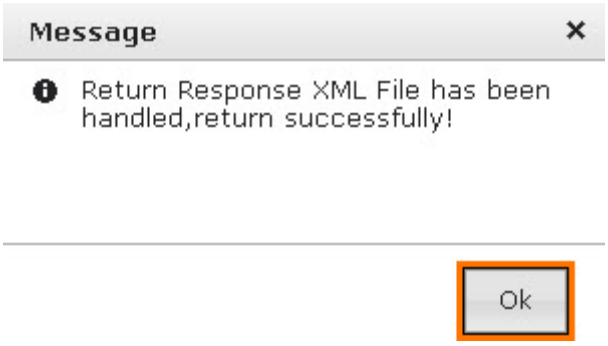
(4) Click Browse to import your Request XML File, and then click Submit



7. Press the ***Return*** button to import the Response XML File, and then click ***OK***.



The following message will be shown as follows after the successful return.



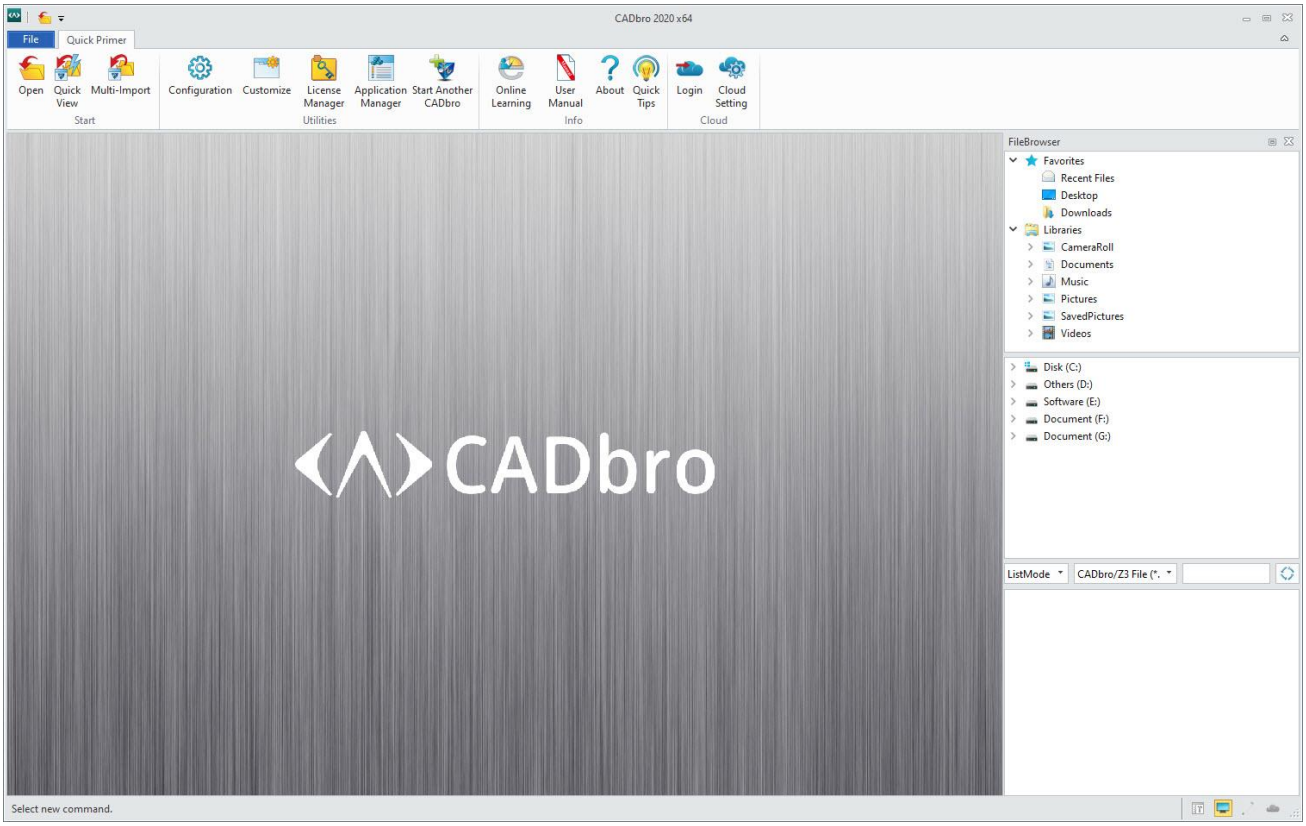
3. Basic knowledge

3.1 Start CADbro

Click the CADbro icon from the desktop

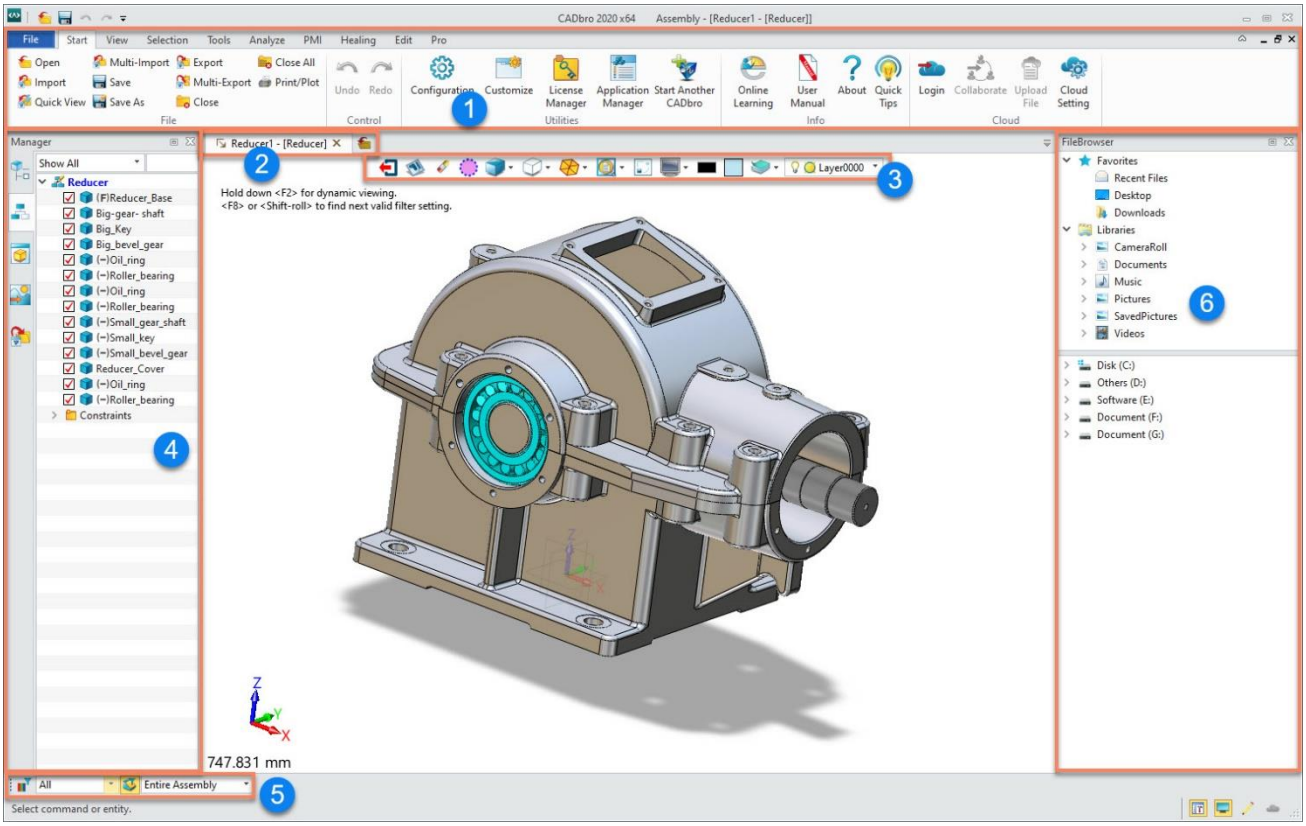
Or Start-> Program->CADbro

The interface below when you first start **CADbro**



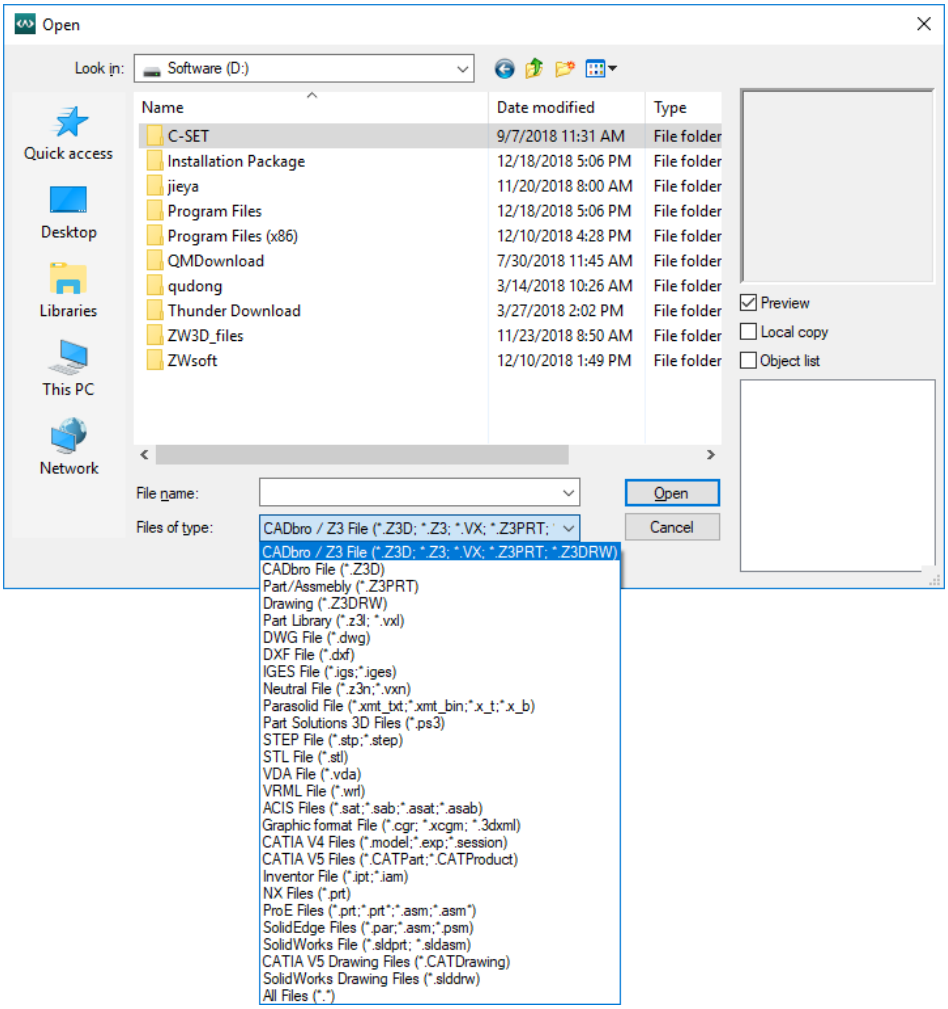
3.2 Interface introduction

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



3.3 Open a new file

File-> Open



3.4 File Browser

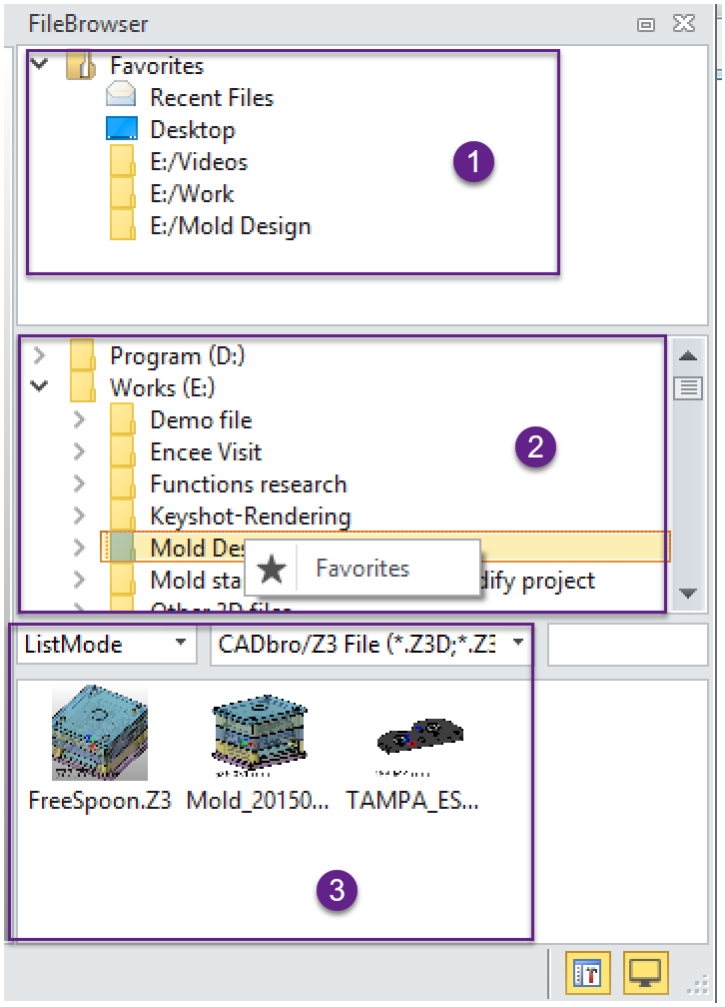
Using the File Browser can quickly locate the directory and add your frequently-used folder to as Favorites.

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

1-> Favorites and Recent files

2-> Disk directory

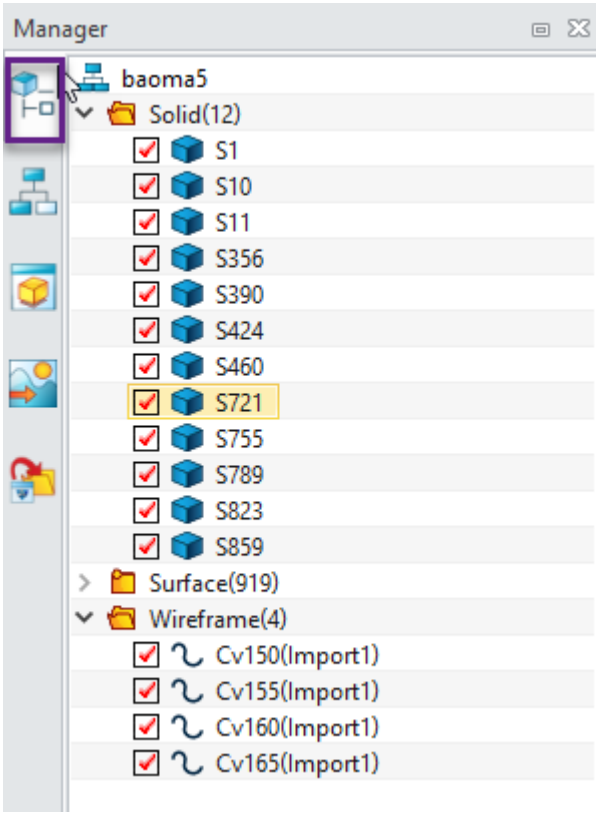
3-> File list windows

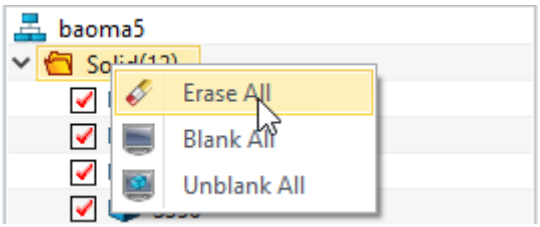

FileBrowser
Favorites
Recent Files
Desktop
E:/Videos
E:/Work
E:/Mold Design
1
Program (D:)
Works (E:)
Demo file
Encee Visit
Functions research
Keyshot-Rendering
Favorites
2
ListMode
CADbro/Z3 File (*.Z3D;*.Z3
FreeSpoon.Z3
Mold_20150...
TAMPA_ES...
3

4. Manager

4.1 Part Browser

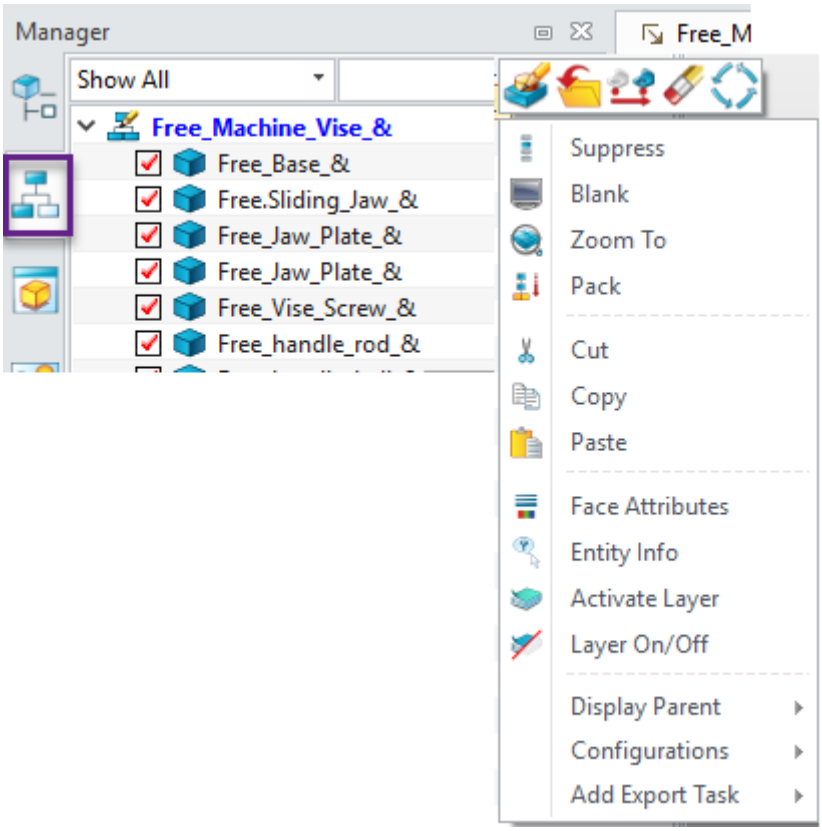
Part Browser used to list all the entities belong to the part, like solid, surface, wireframe and so on, you can uncheck the checkbox to hide the entity or right-click the folder to erase, hide or show all.





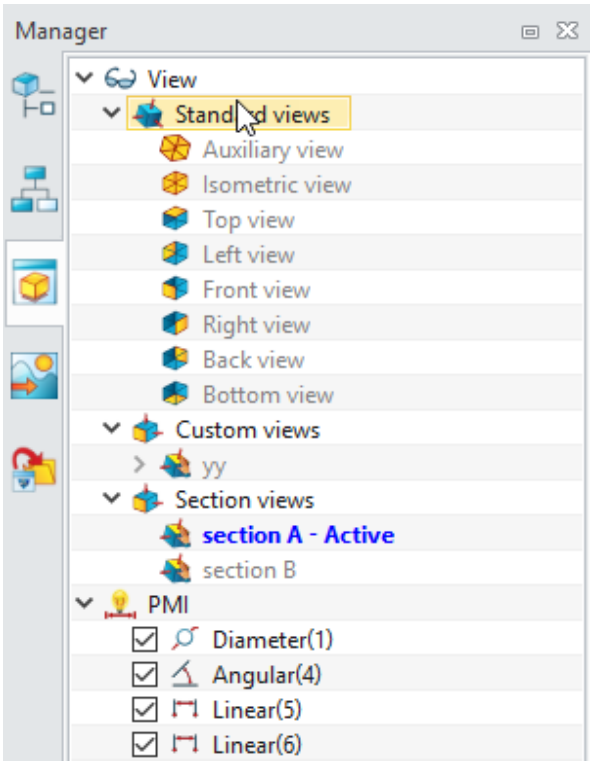
4.2 Assembly manager

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.



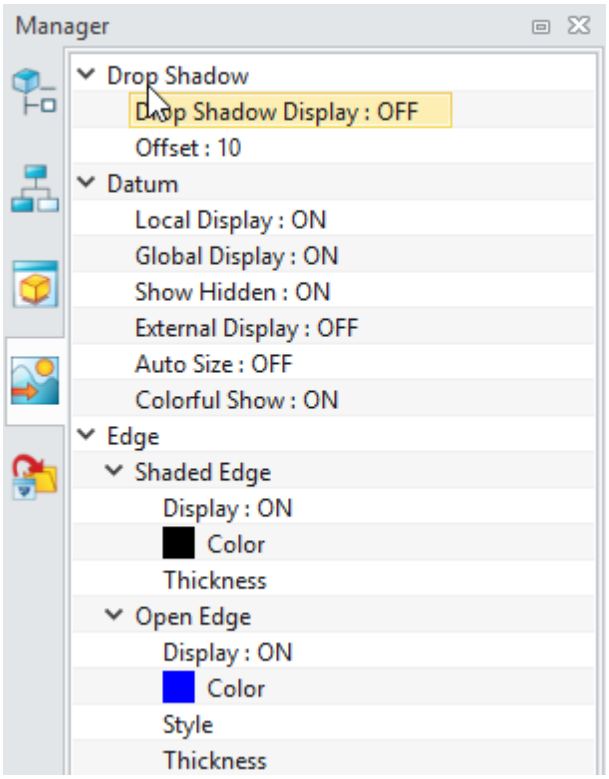
4.3 View manager

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



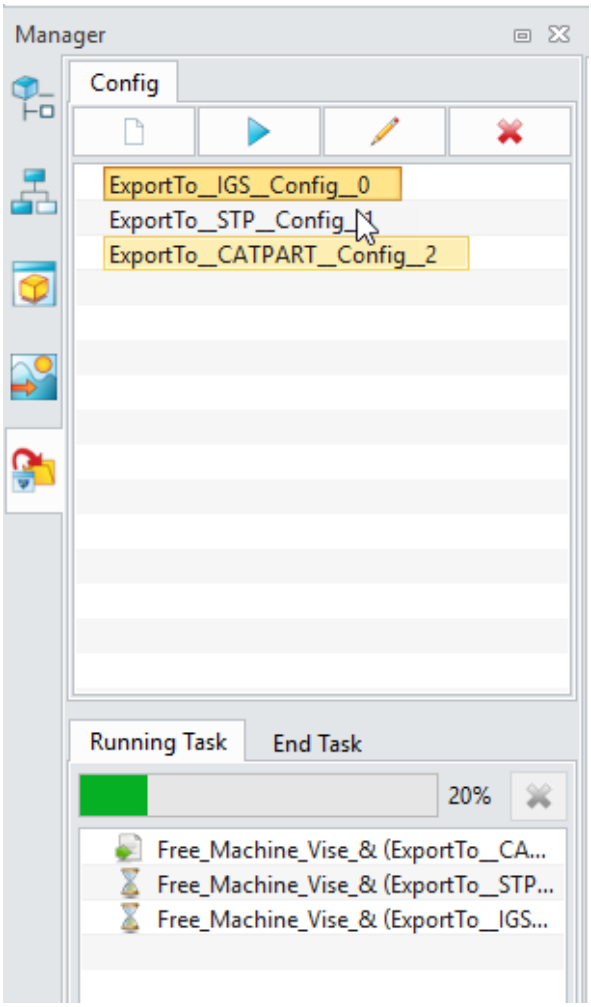
4.4 Visual manager

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.

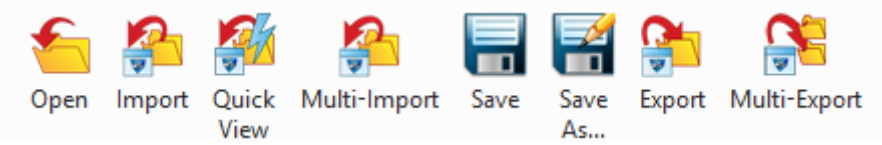


4.5 Export manager

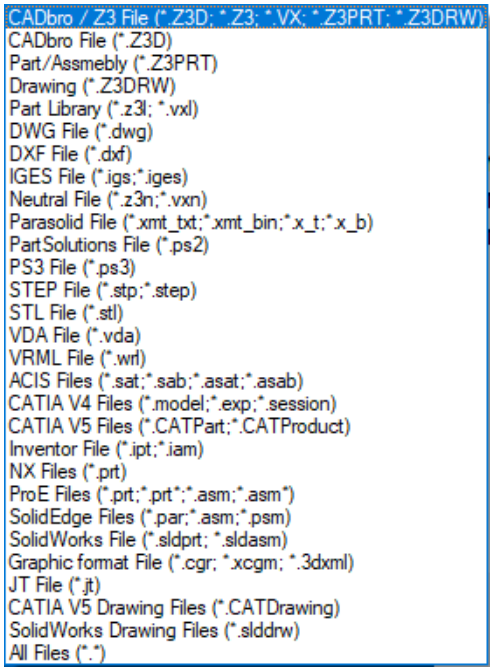
Export manager support to predefine your exporting configuration, like exporting the file format to a specified folder, it can be saved as configuration, after that, you can put the current 3d model from graphic window to run this task.


Manager
Config
ExportTo__IGS__Config__0
ExportTo__STP__Config__1
ExportTo__CATPART__Config__2
Running Task
End Task
20%
Free_Machine_Vise_& (ExportTo__CA...
Free_Machine_Vise_& (ExportTo__STP...
Free_Machine_Vise_& (ExportTo__IGS...

5. Covert files

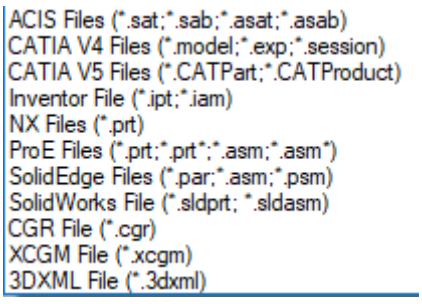


Open: Open all the supported files in CADbro, including the Z3D file.



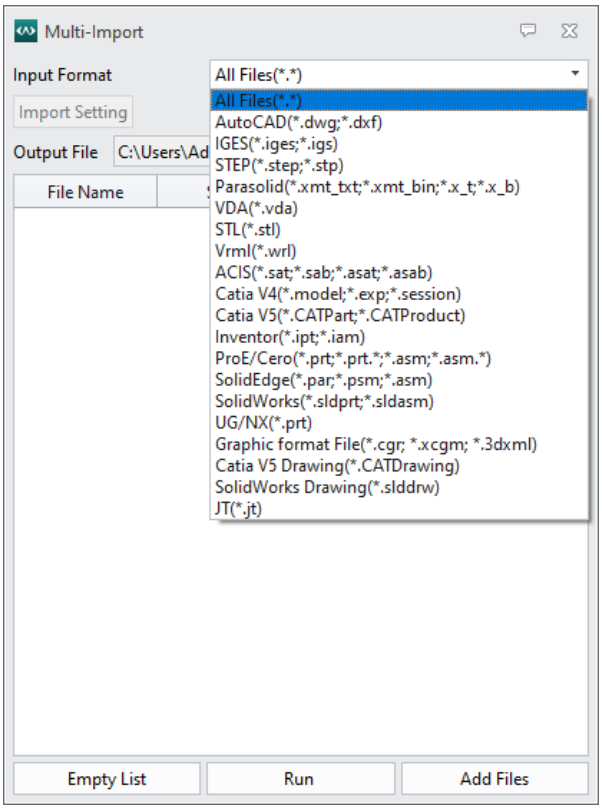
Import: Import all other files into current Z3D. It will not create a new Z3D file.

Quick View: Quick import all file below in a new Z3D file with only display data, this kind of importing is faster than normal importing, but there is no Nurbs data on it. This means limited editing on such an imported model.



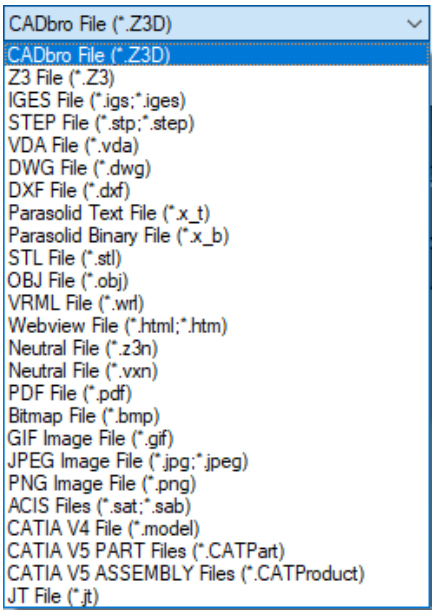
Multi-Import:

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.



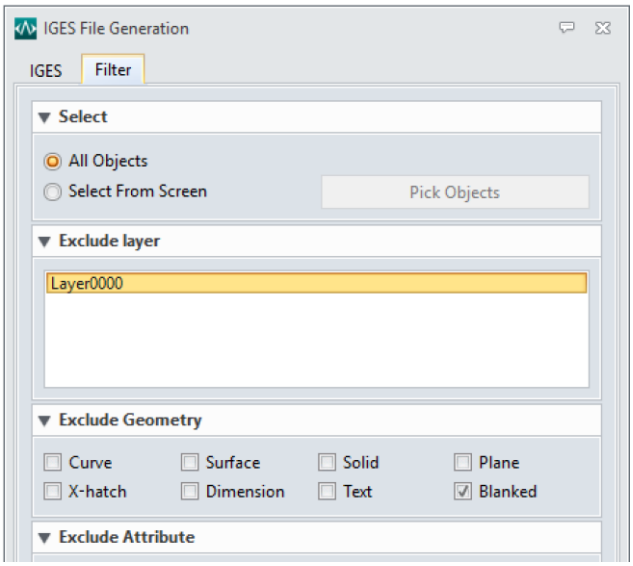
Save: Direct save the current Z3D file.

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

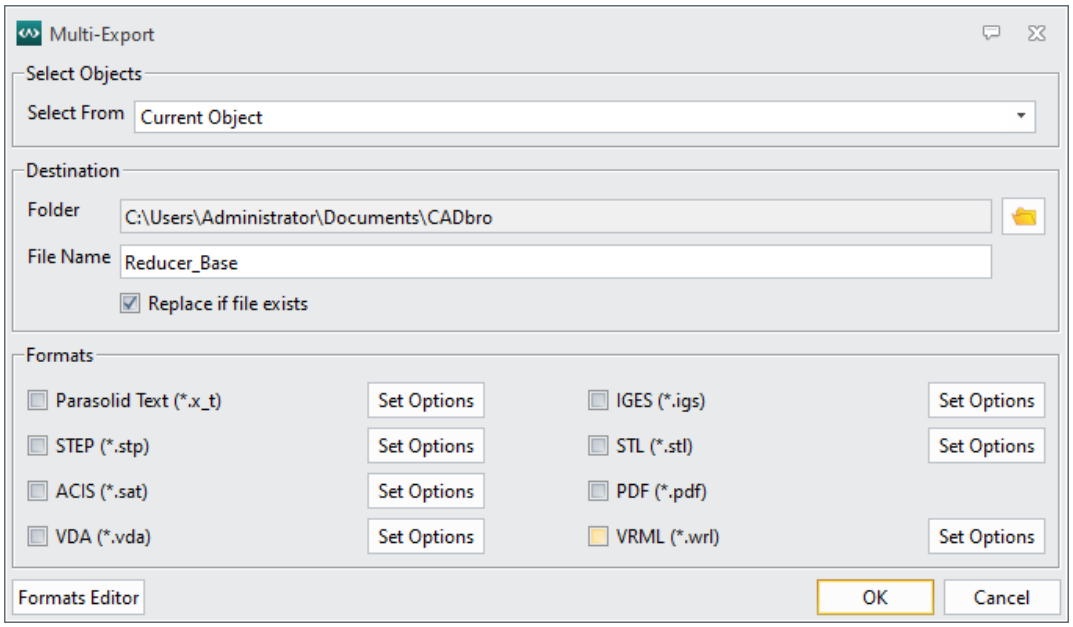


Export: Compare to **Save As**, **Export** command support to select specify shape or component

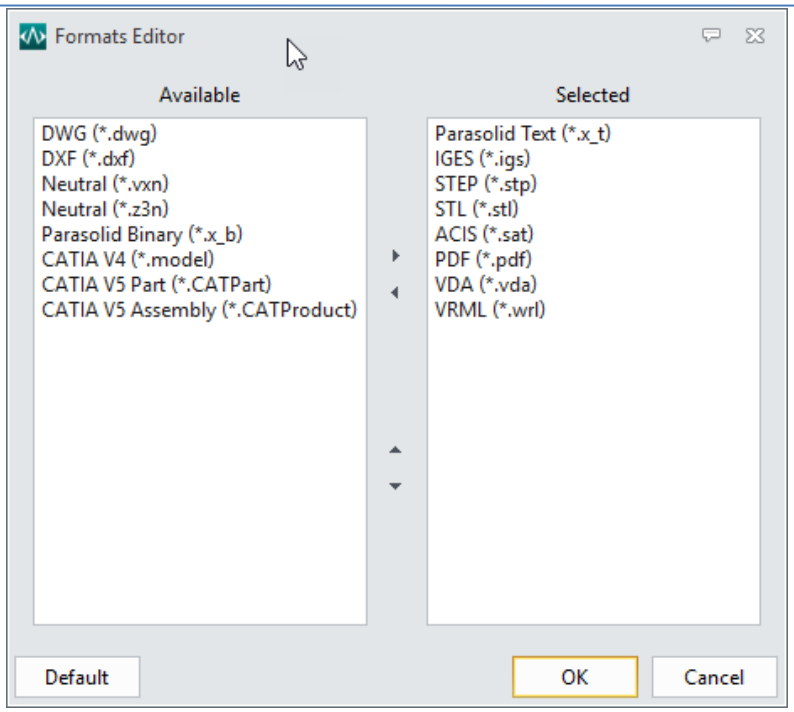
from the current file to export. Even can make a filter to include or exclude by Layer, Geometry type and Attribute.



Multi-Export: Using the default exporting setting to export currently opened files in the graphic window to multiple file formats at one time.



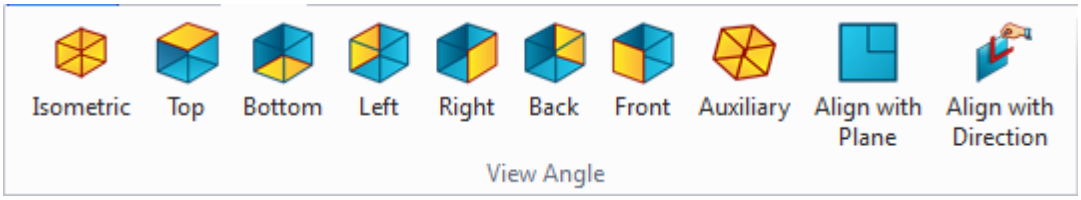
Click the **Formats Editor** button, you can add more default formats to the dialog box.


Formats Editor
Available
DWG (*.dwg)
DXF (*.dxf)
Neutral (*.vxn)
Neutral (*.z3n)
Parasolid Binary (*.x_b)
CATIA V4 (*.model)
CATIA V5 Part (*.CATPart)
CATIA V5 Assembly (*.CATProduct)
Selected
Parasolid Text (*.x_t)
IGES (*.igs)
STEP (*.stp)
STL (*.stl)
ACIS (*.sat)
PDF (*.pdf)
VDA (*.vda)
VRML (*.wrl)
Default
OK
Cancel

6. View

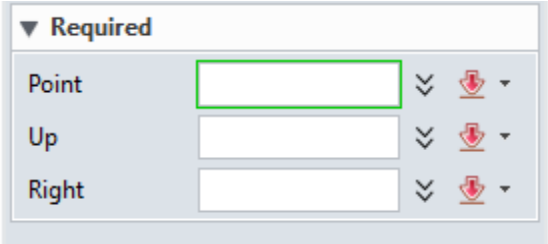
6.1 View Angle

Click different View icon to change the view angle

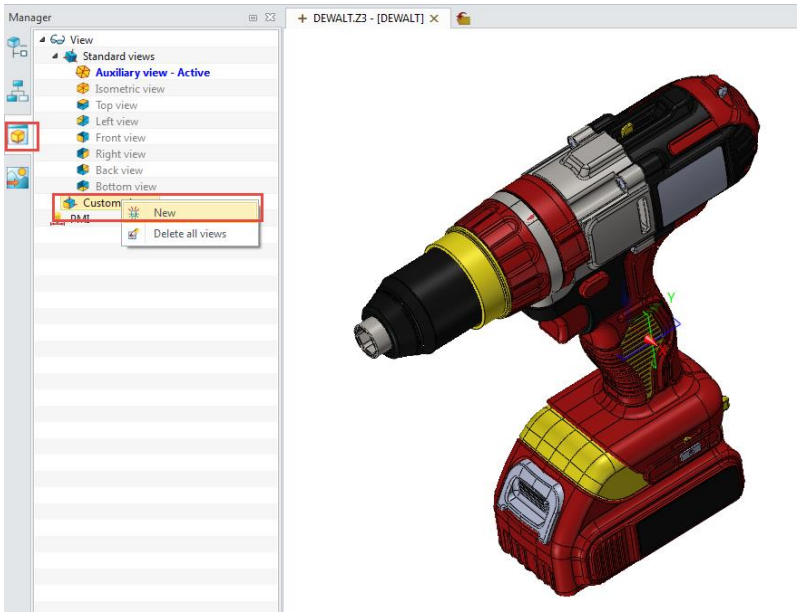


Align with Plane:
Click this command then select one plane, the view will change to auto-align this plane

Align with Direction:
Define the origin point, Up direction and Right direction to customize the view angle

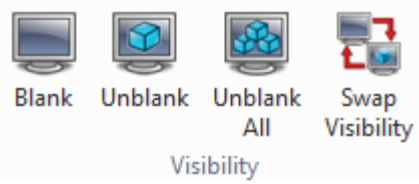


How to save the View angle and reuse it later?
Click the **View manager**, then right-click the **Custom View -> New**, then you can save current view angle

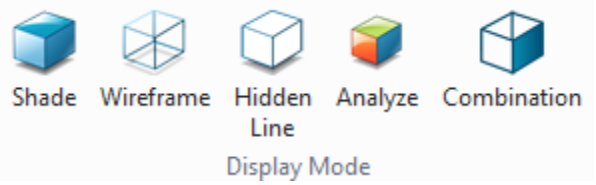


6.2 View

6.3 Visibility

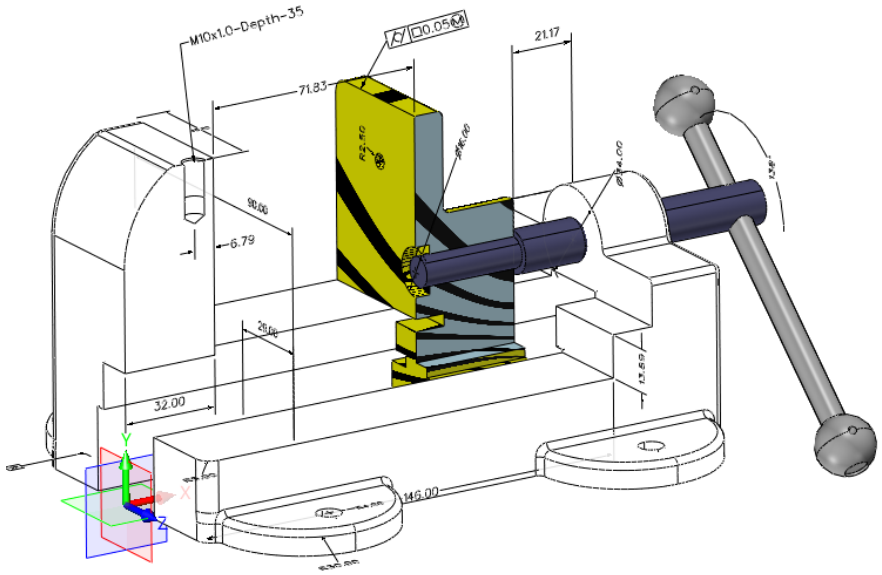


6.4 Display Mode

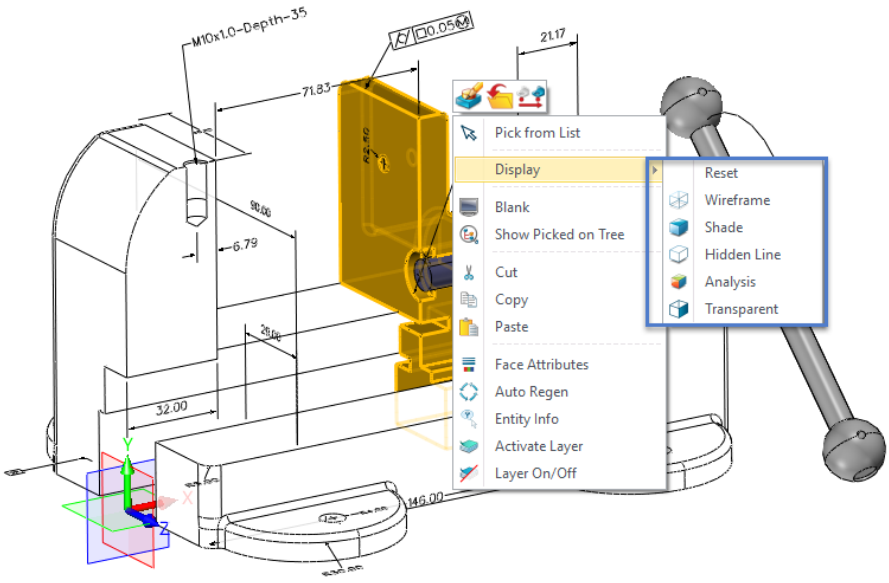


Analyze: Display the previous face analyze status

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.*

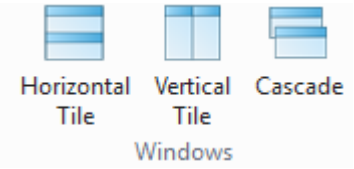


6.5 Screen

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

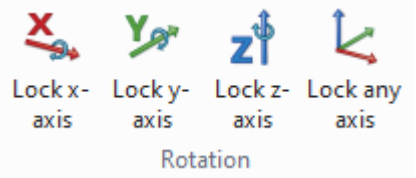
6.6 Windows

The list method for all open files.

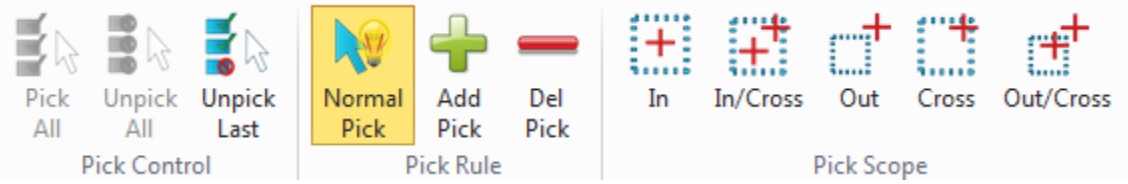


6.7 Rotation

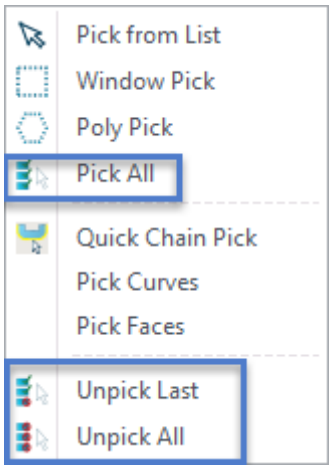
Rotate the 3D model around the X/Y/Z axis. Users can also difine a specific axis by **Lock any axis** for rotation



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.

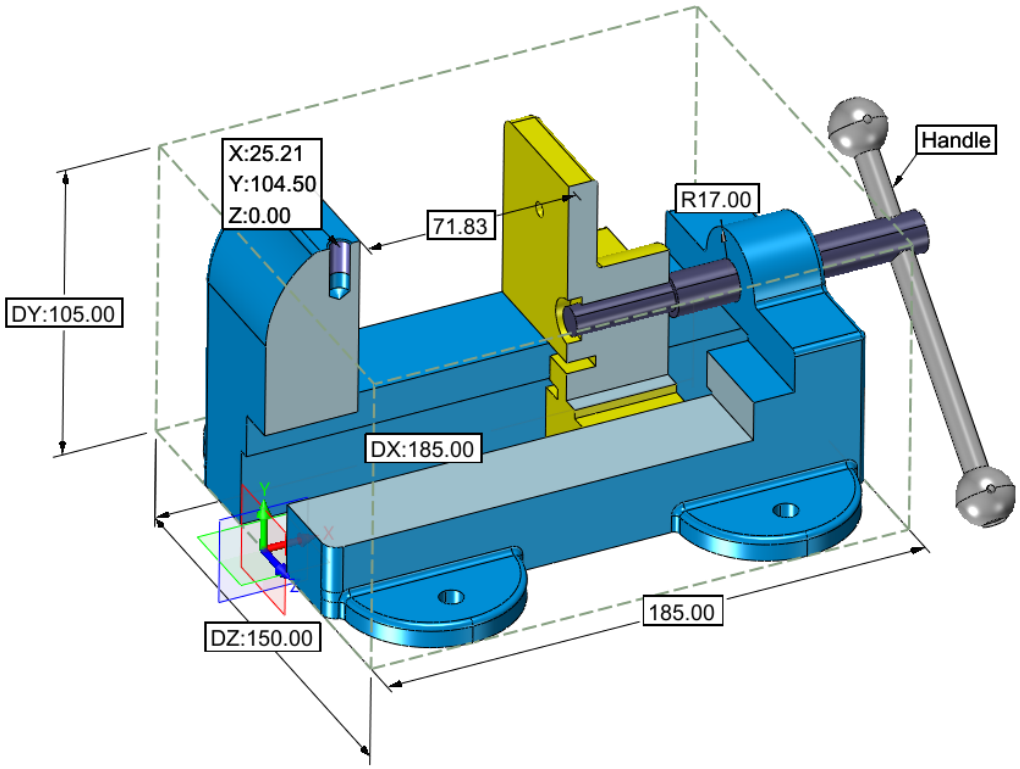
Pick Scope: The window pick behavior, for example, it means all the entity in the windows frame will be selected.

8. Tools

8.1 Dimension

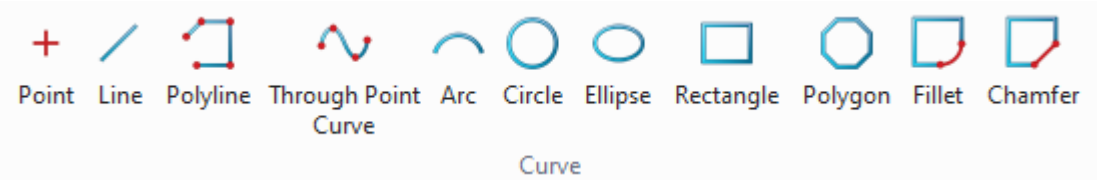
New style dimensions can be created by the following tools, which are always parallel to the view window.

The final effect is shown below.

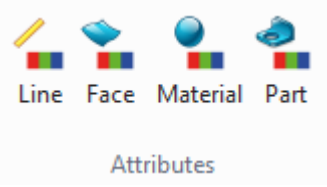


8.2 Curve creating

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



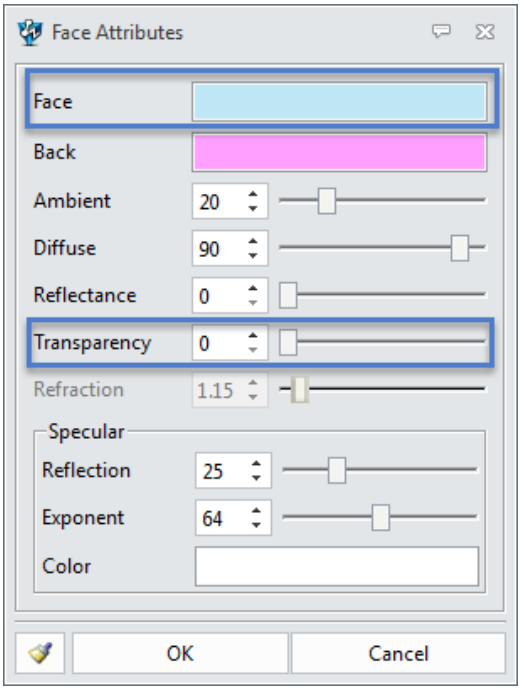
8.3 Attribute



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

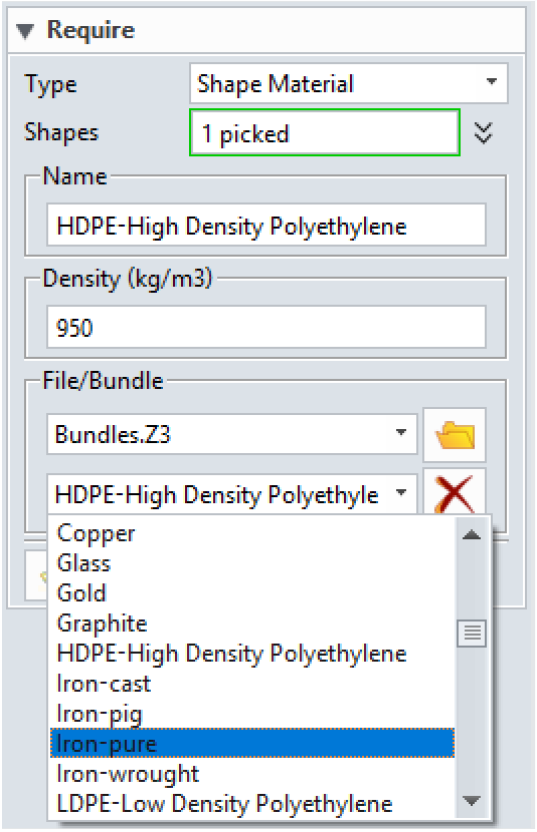
Face attribute: Use to change the face color, or you can select face set or shape body first then click this command

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

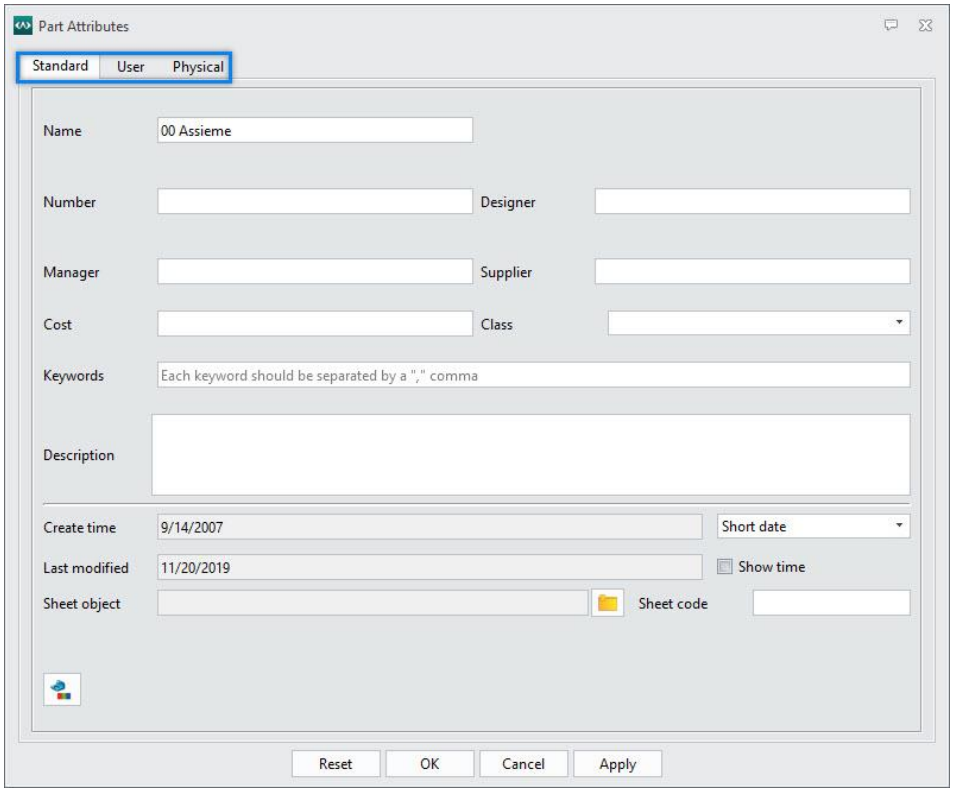


You can also find the command from the Quick toolbar as below.

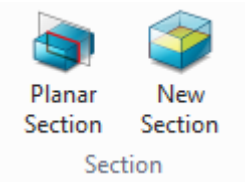
Material attribute: Use to change the shape or part materials, which allow you to select the existed material from the list or add your new material.



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.



8.4 Section



Planar Section:

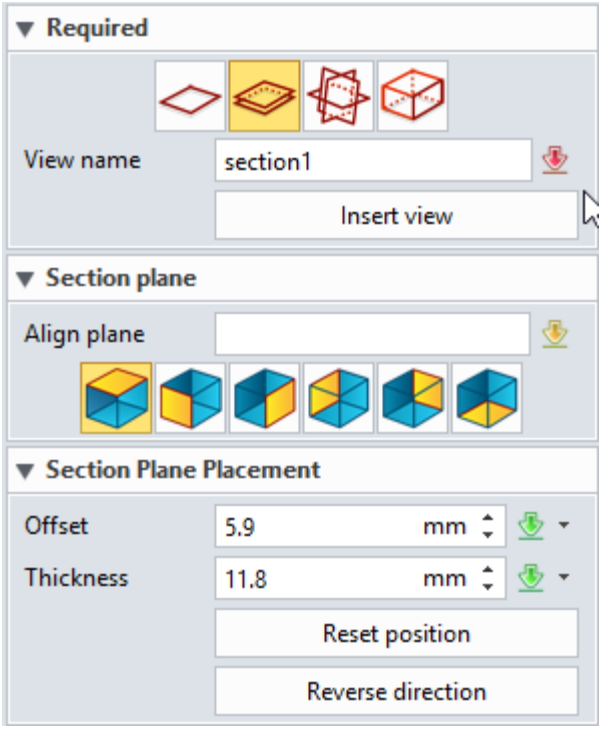
Use this command to dynamically section the active part or assembly. Completing this command will create section curves in the active part and the active component in an assembly. It can also be used as a visual aid to dynamically clip away portions of the part to view features that might normally be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



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

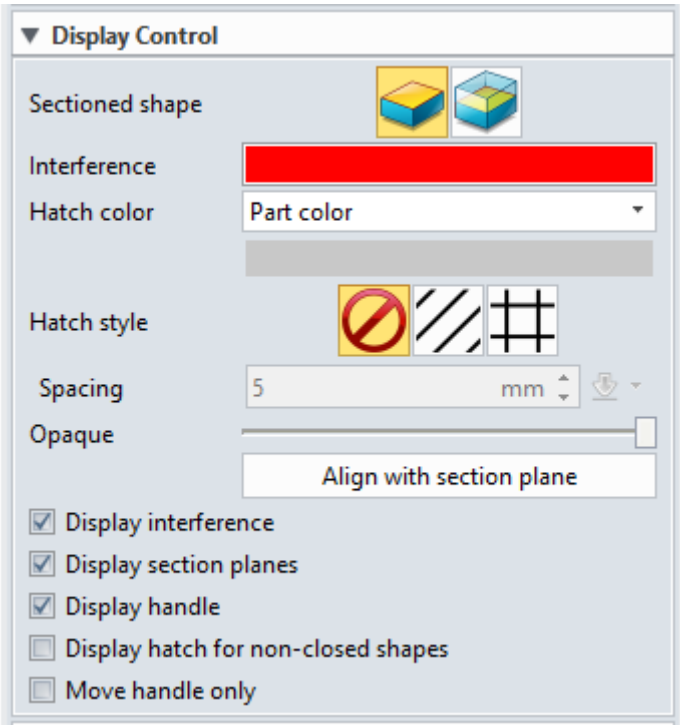
Align with section plane- can align the current model to the section plane

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

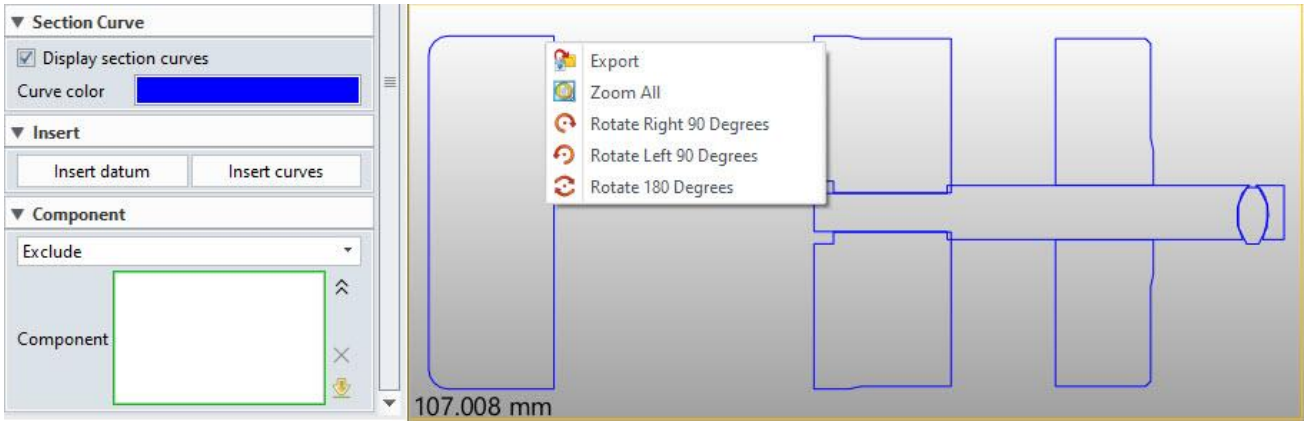
CADbro supports to display the section curves with extra window, working on the extra window, you can rotate the section profile even export it to DWG/DXF/IGES and STEPS.

Insert

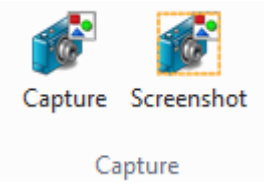
CADbro supports to insert datum and curves into the current model.

Component

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



8.5 Capture

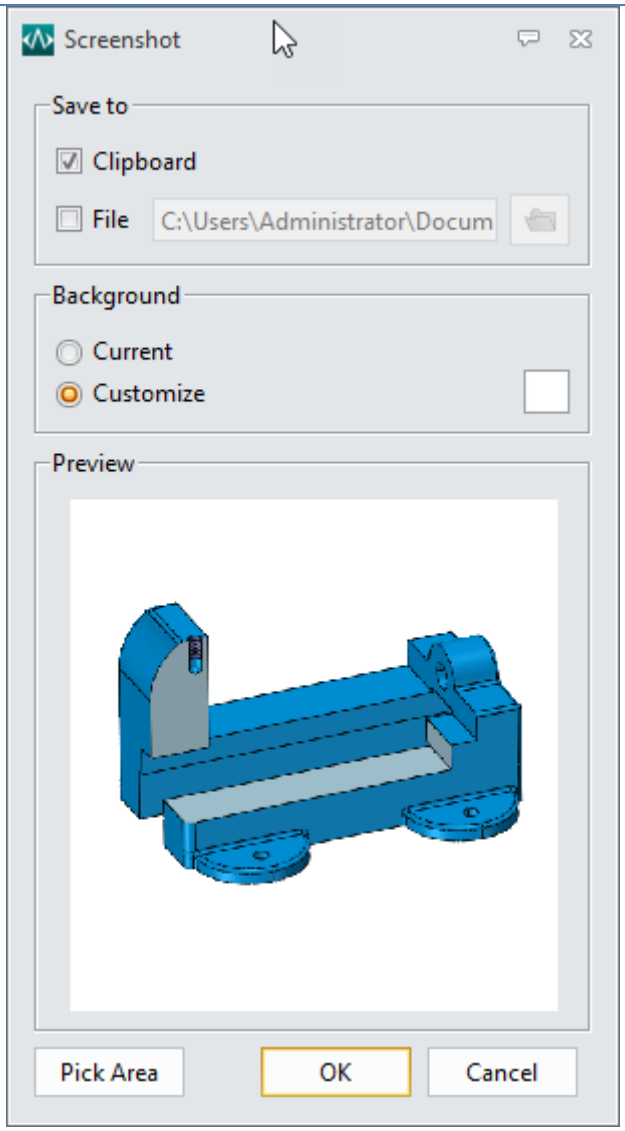


Capture:

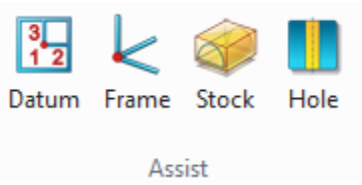
Use this command to capture the active display to a file. It automatically turns off the view extents readout, the world axis triad, and the default datum display. You can save to bmp, gif, jpg, and tiff graphics file formats. It also allows up to the maximum pixel resolution supported by your graphics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



8.6 Assist



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.

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

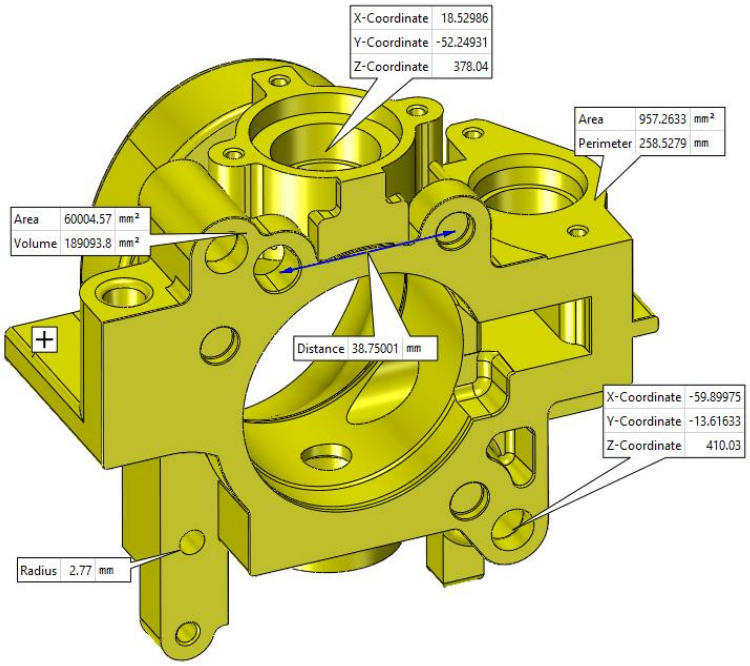
Use this command to create a simple hole in CADbro.

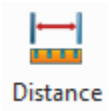
9. Analyze

9.1 Measure

Measure:

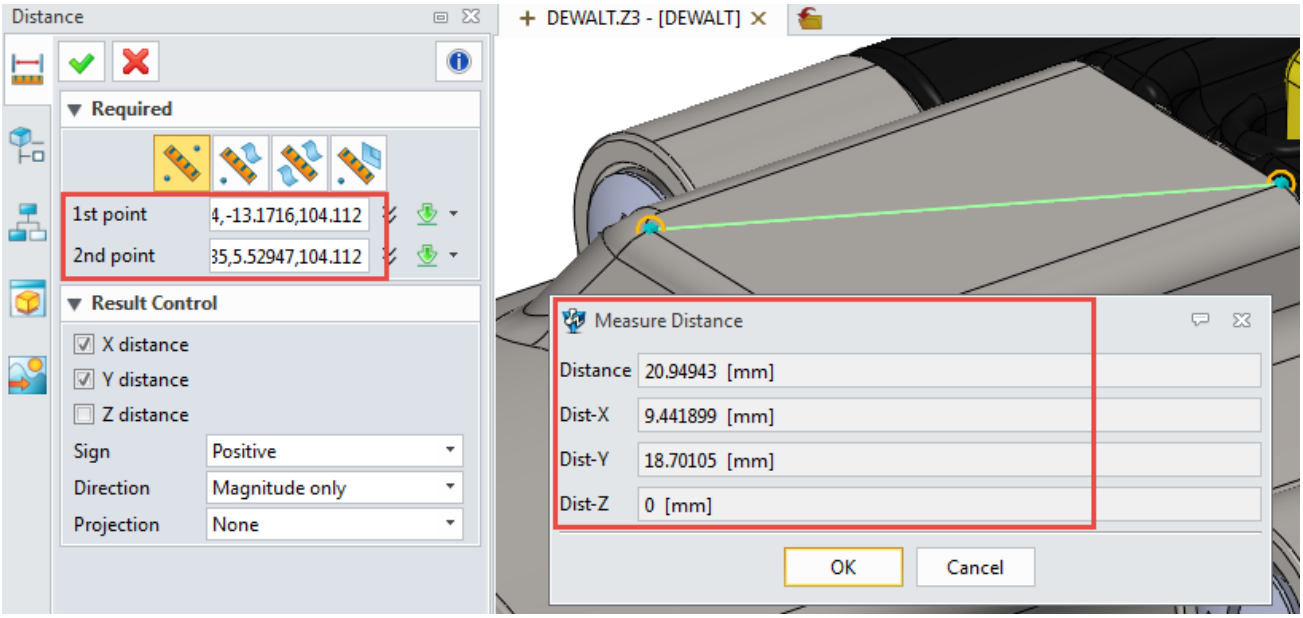
This command can base on the selected entity to generate the measurement result, which can be also recorded and attached in the model and minimize the label.





Distance:

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.



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:

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.

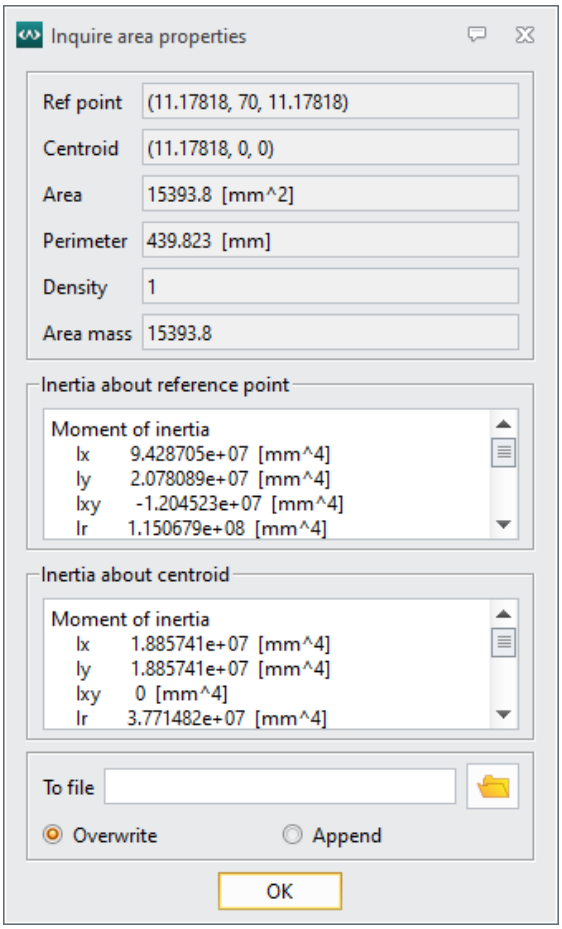
Length：

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.
2) 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:

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.



3D Surface:

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

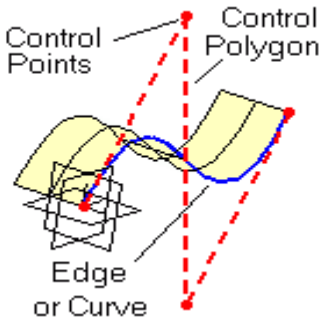
Use this command to inquire curve information on any line, arc, curve, or edge.

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

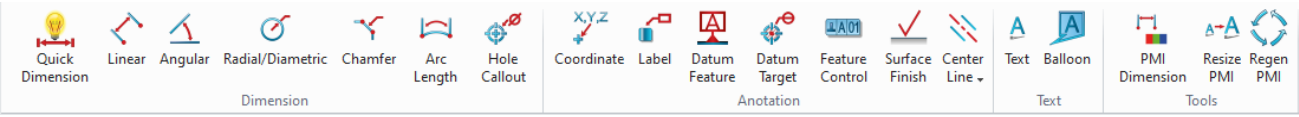
Use this command to analyze the tangent continuity across two mating faces at a selected edge. The edge must be shared by both faces. The results displayed are the maximum and average error measured in degrees. You can also check for C0 continuity (gaps) along the edge by selecting **Position** from the **Continuity** option.

9.3 Inspect Model

Mass Properties

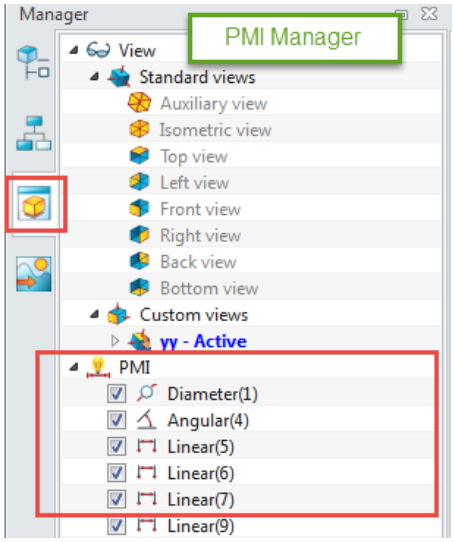
Use this command to calculate and display mass properties information (e.g., volume, area, mass, etc.) for a selected shape (open or closed), component, or the entire assembly. Blanked and/or suppressed components in an assembly will be ignored during these calculations. (Surface) The area is the only 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.

10. PMI



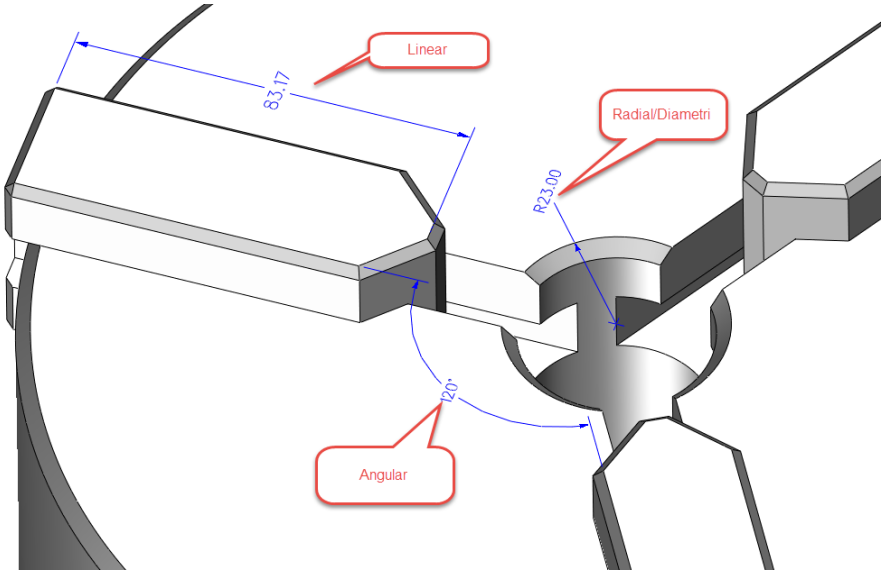
10.1 Manager

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.



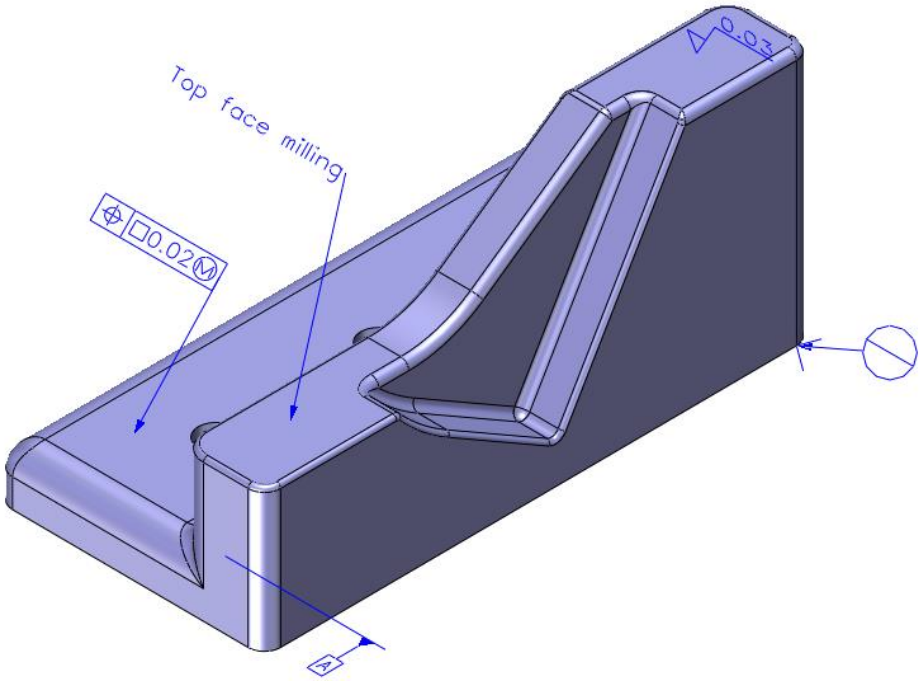
10.2 Dimension

Use to create different 3d dimension types as below.



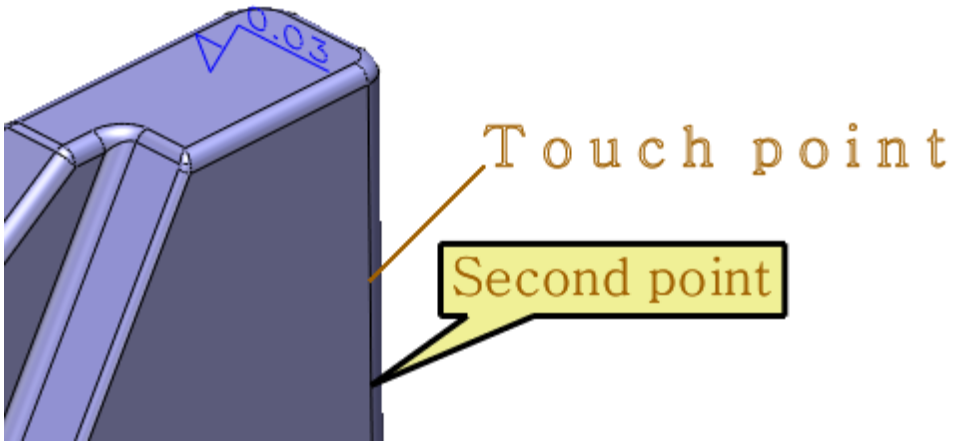
10.3 Annotation

Use to add different 3D annotations on the model.



10.4 Text

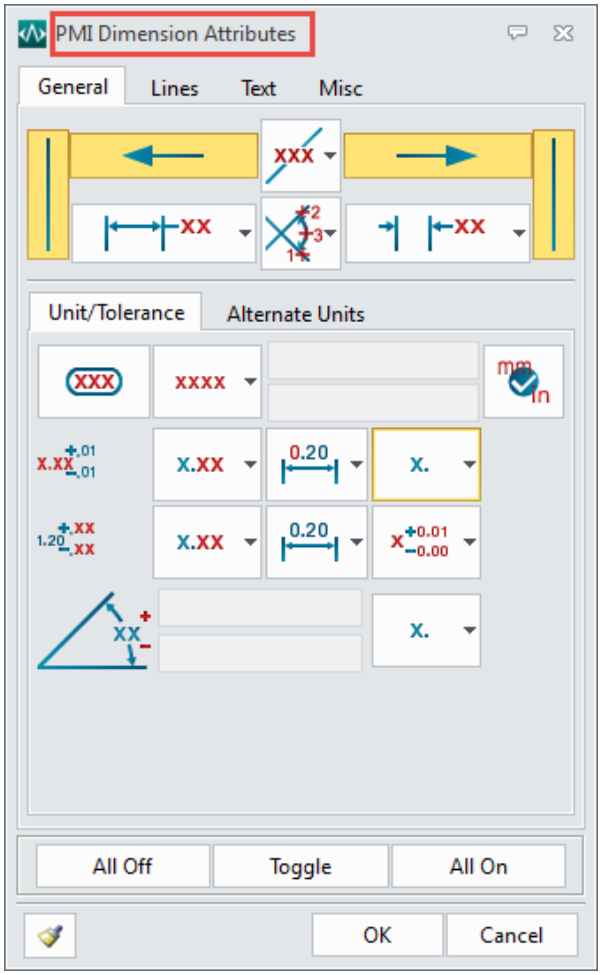
Simple text and the text with balloon



10.5 Tools

PMI Dimension

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



Resize PMI

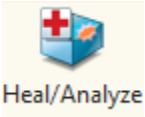
When creating a PMI note, it will generate a different note sizes base on a different view, this command can reset the global PMI size to match all notes.

Regen PMI

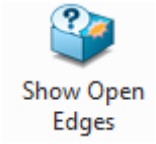
This command uses to do regeneration for PMI, if you want the PMI to be refresh or update, you can use this command to do that

11. Healing

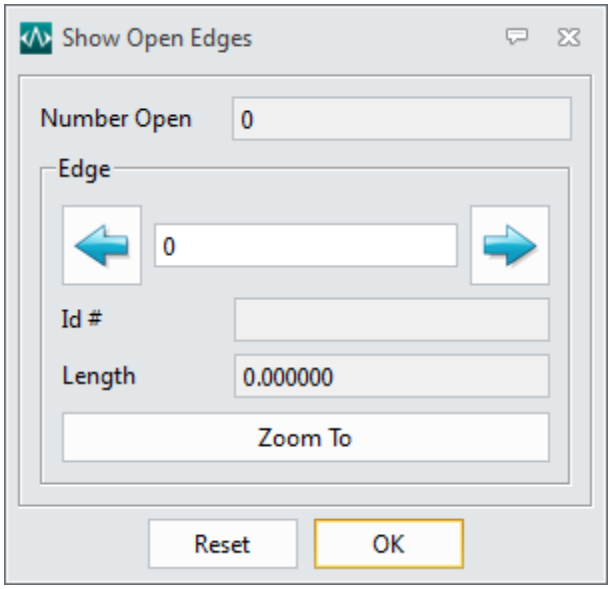
11.1 Checking



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.



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.

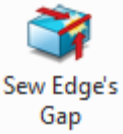


This command checks for and/or delete tiny edges whose curve lengths are smaller than a specified tolerance and that are not required to define the topology of the active part. This command will attempt to bring the encountered conditions within the current Geometry Tolerance setting. Select the edges to check/delete or middle-click for all edges.

11.2 Healing

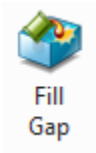
Sew the open edge face to be merge one

Explode the selected face set to be multiple faces.



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.

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

DE Move

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

Align Move Face

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**

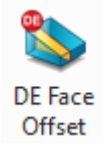
  Select a stationary object. The motion face will keep the specified constraint with this object.
- **Angle**

  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.

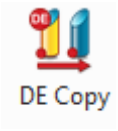


DE Face offset

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.

DE Draft

This command creates a draft feature about selected faces.

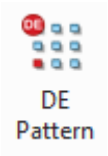


DE Copy

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

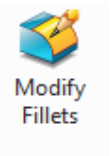
DE Mirror

Use this command to mirror the selected faces.



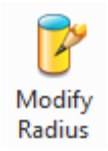
DE Pattern

Use this command to pattern the selected face.



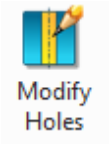
Modify Fillets

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.

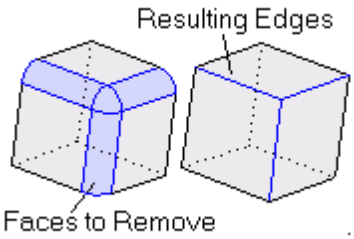
Replace

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

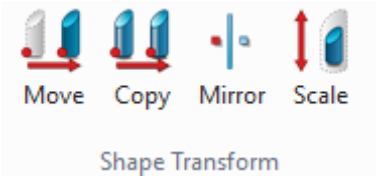
Simplify

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.

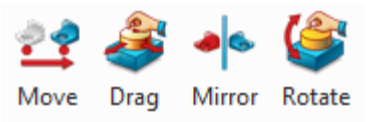


12.2 Shape transform



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

12.3 Part transform

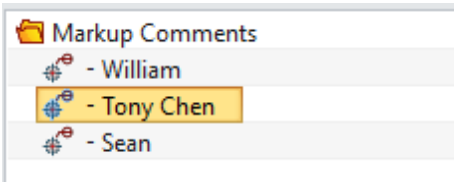


Part transform commands support to move, drag, mirror and rotate the selected component. Compare to shape transform, the target entity type is different. One for shapes, one for components.

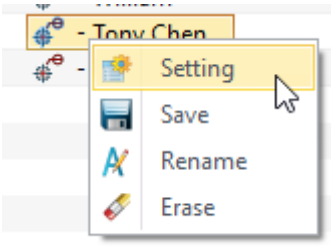
13.Mark up

Create new comment

Create a new comment will list on the manager as below

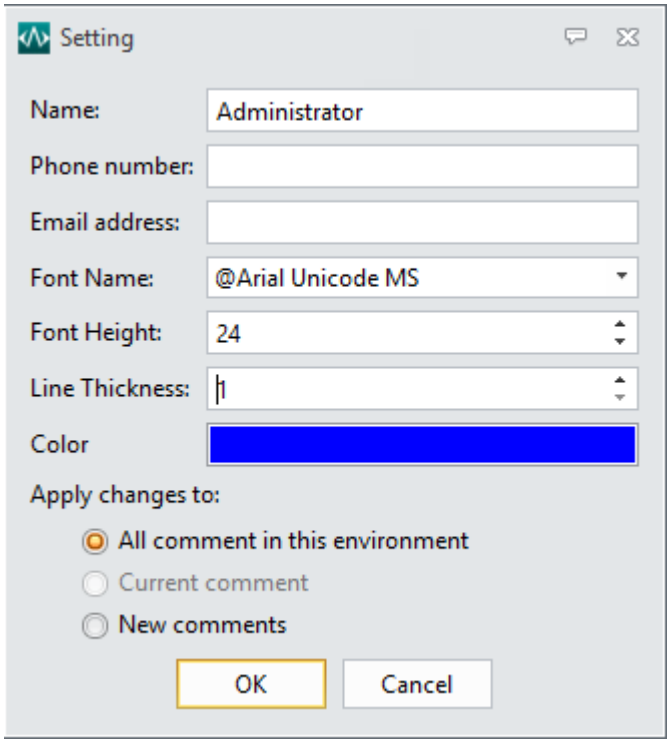


Right-click the comment you will see the list below, you can change the setting, save the comment to Z3DM file, rename or erase it.



Setting

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



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 right-click a comment and use the setting.
New comments: Will affect the new creating comment, but not the existed comments.

Open markup

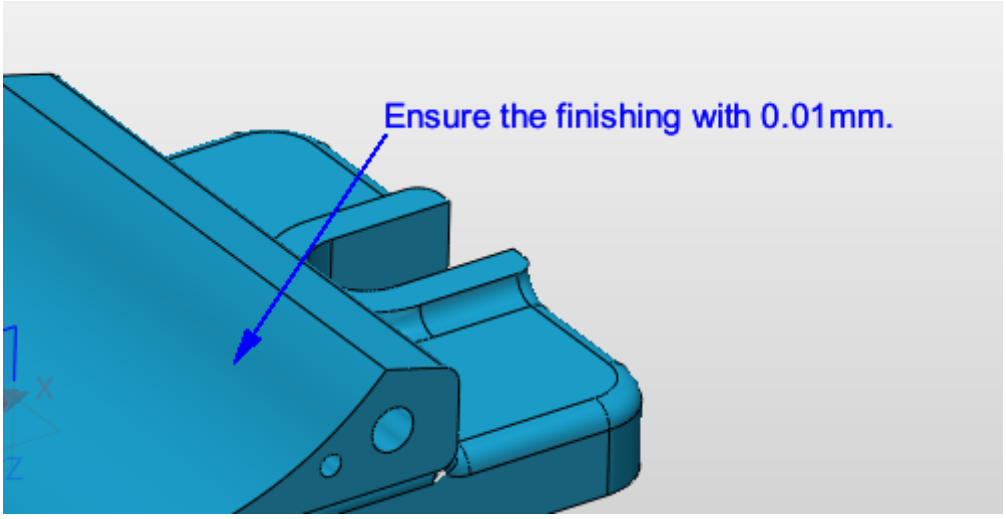
Open the saving comment file, which is Z3DM file.

Save markup

Save all comments or select one of them to save.

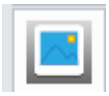
Text with leader

Create the label text with a leader.



Text

Only create the text without a leader

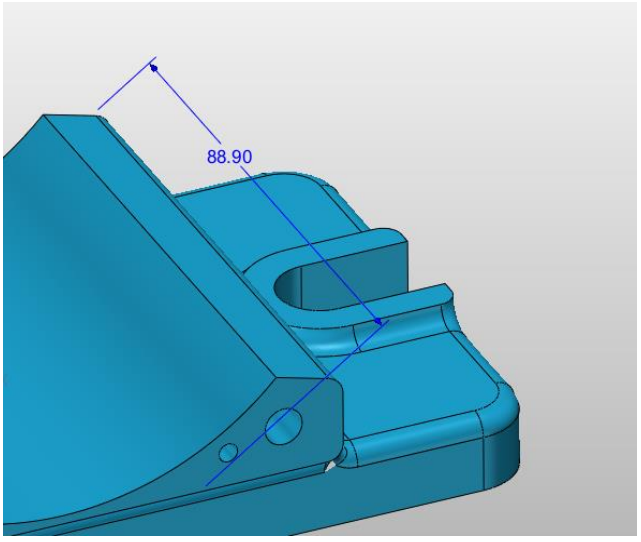


Image

Insert the extra image.

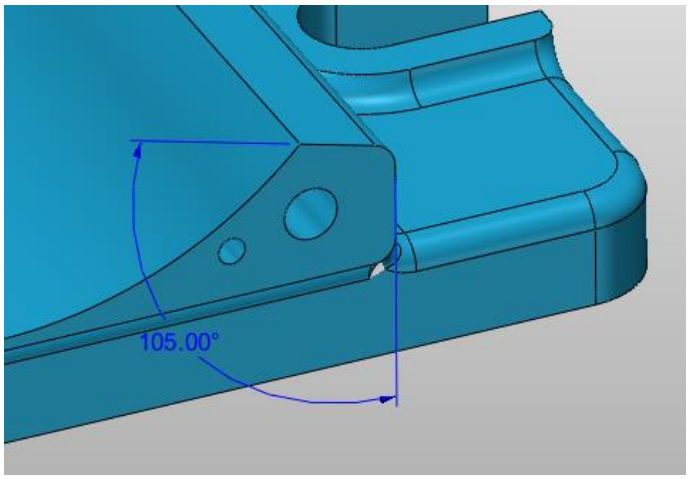
Linear dimension

Create the linear dimension as below



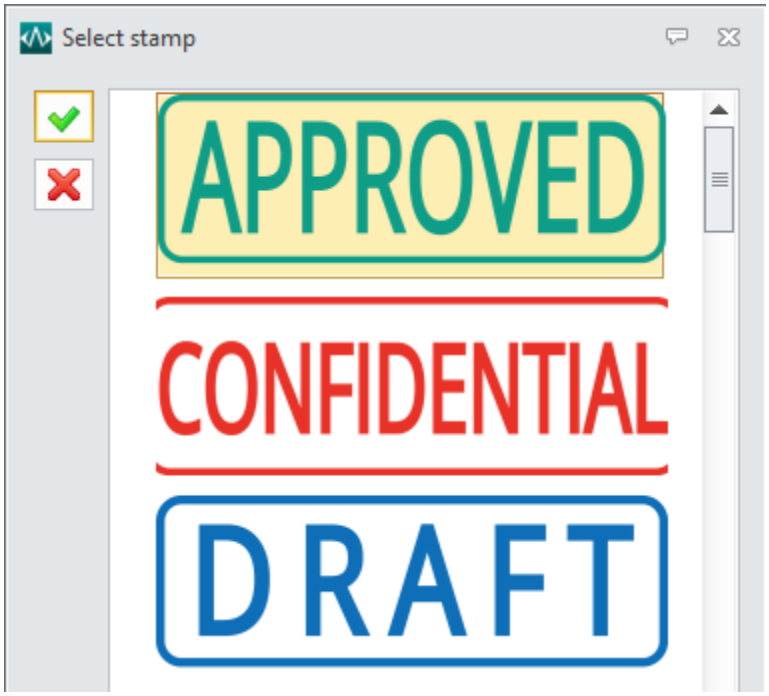
Angular dimension

Create the angular dimension as below.



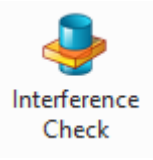
Stamp

Insert the stamp to the screen



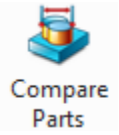
14. Pro

14.1 Assembly



Interference check

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.

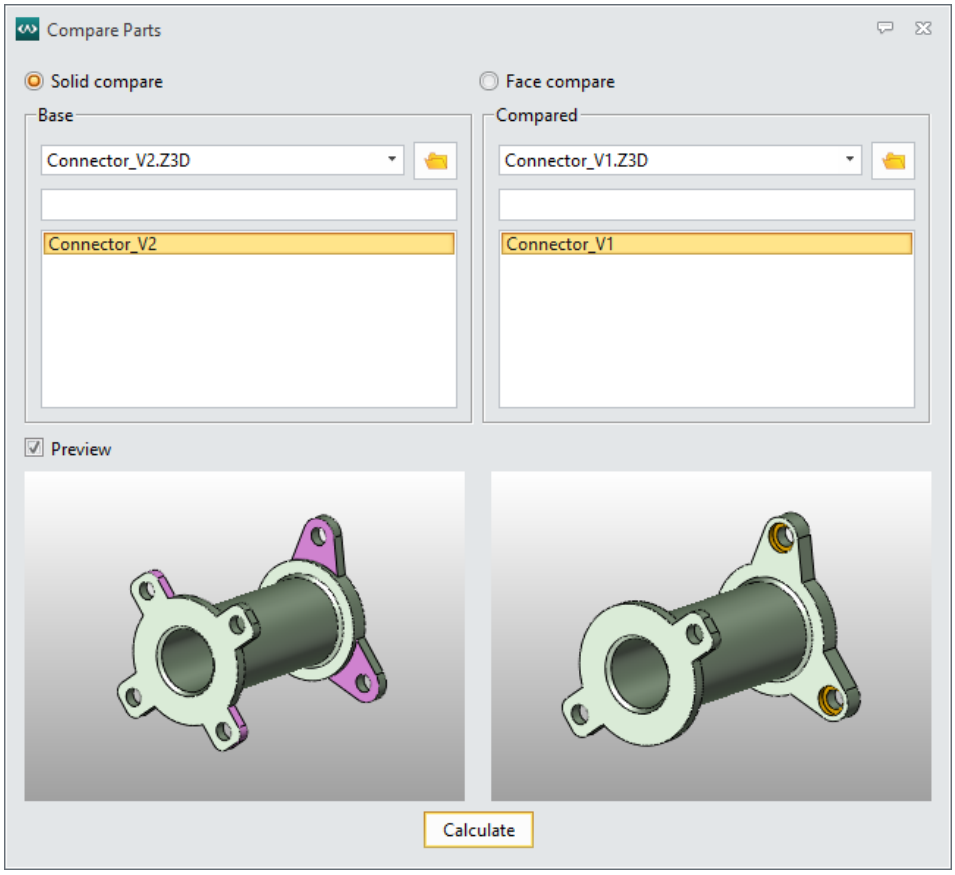


Compare parts

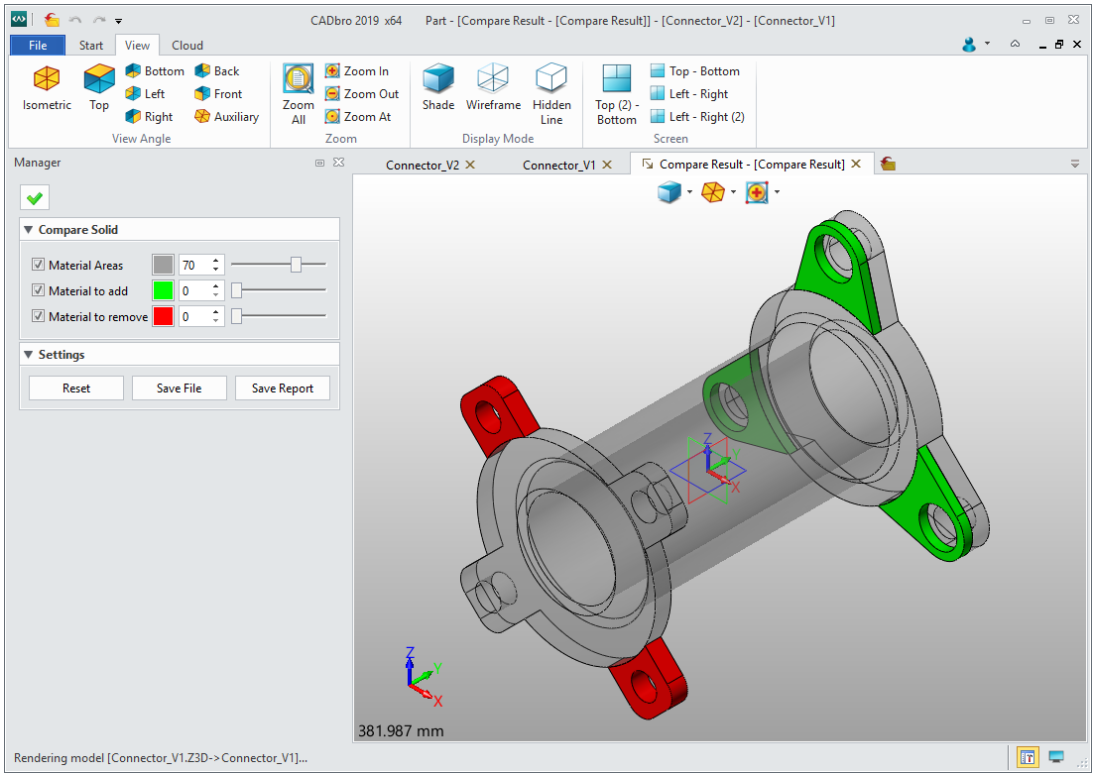
Use this command to compare two parts and show the differences between them. CADbro provides two methods for comparison, solid compare and face compare
This command support to select the Z3D file from local disk without opening in CADbro.

Solid Compare

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



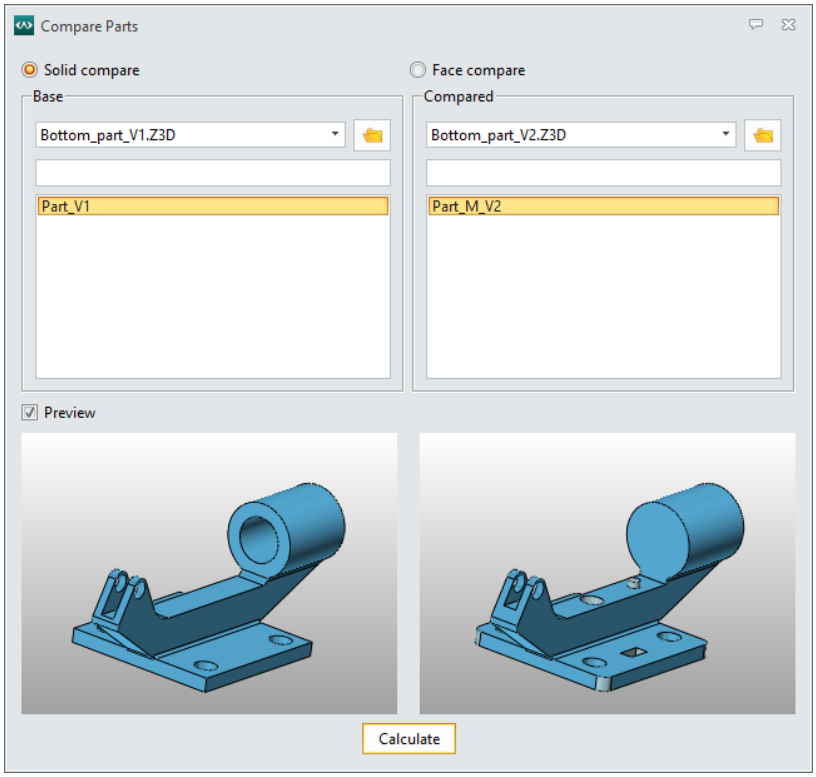
Below is the compare result, which shows the different regions, you can also change the color and the transparency and hide the regions.



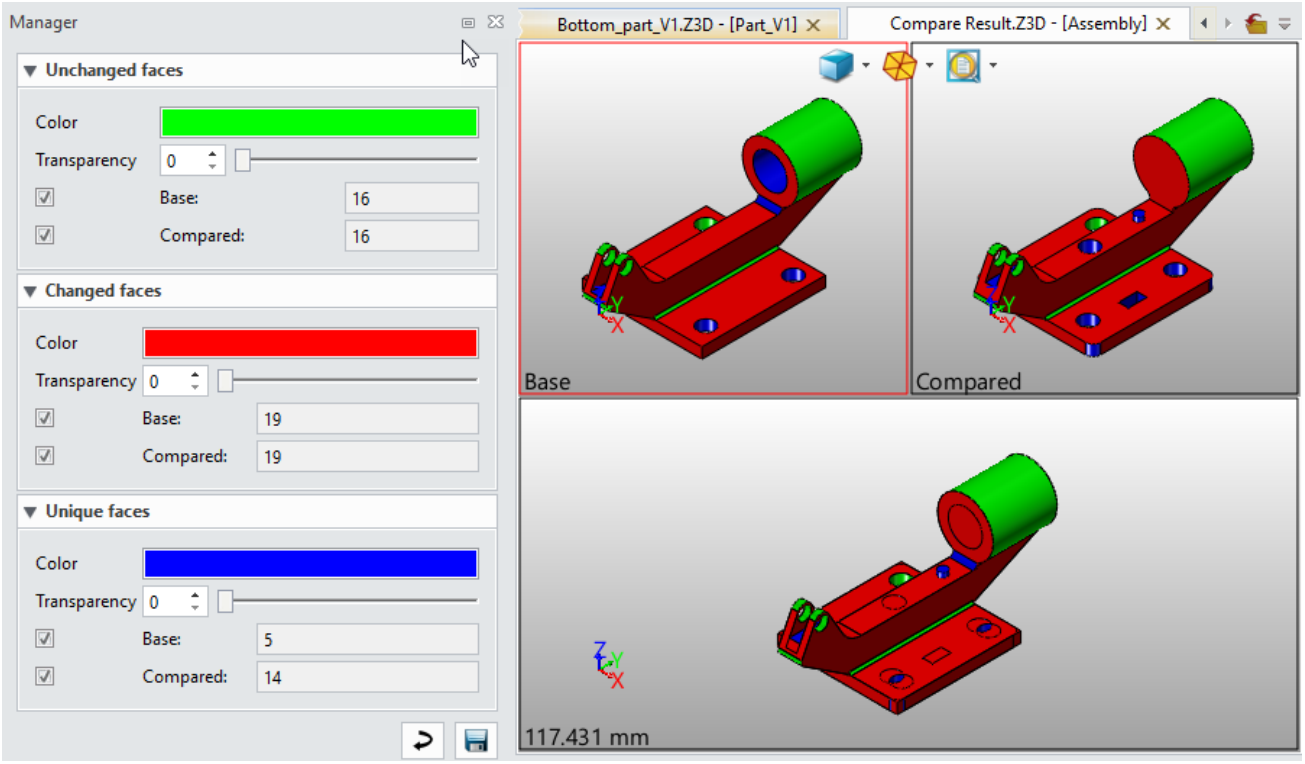
Reset: Reset all the settings for display, color, and transparency.
Save file: Save the comparison result as Z3D or other neutral formats.
Save report: Generate a comparison report.

Face compare

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



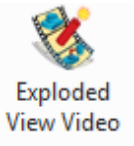
Below is the calculating result, which shows the unchanged faces, changed faces and unique faces, you can also change the transparency and hide the faces.



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 drag-dropping the picked step.



Exploded View Video

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

3D BOM

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

- **Level Display Settings**

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.

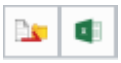
- **Search Function**

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 conten

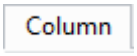
t will be highlighted.

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

- **Export Data** 

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** 

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

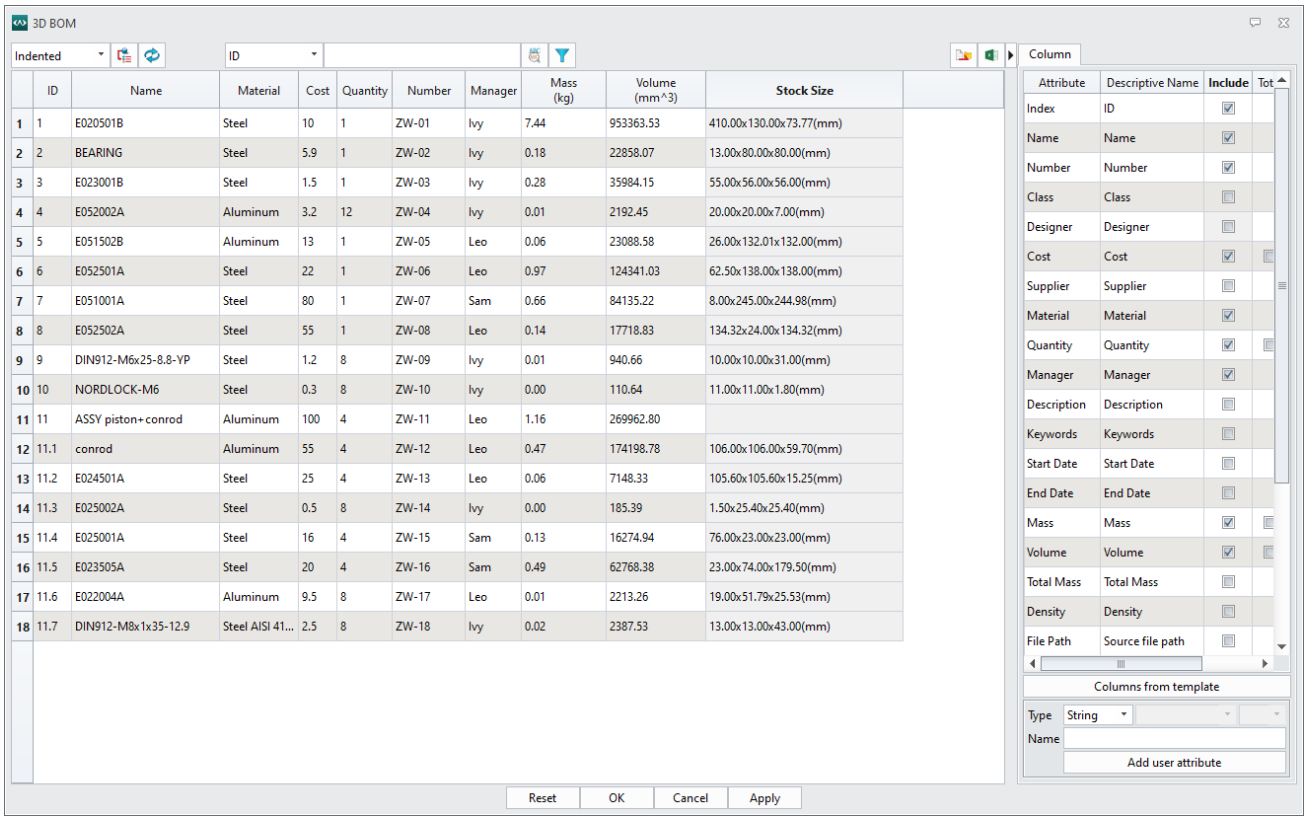
Attribute: Attribute is the corresponding name in the system and can not be modified.

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.



	ID	Name	Material	Cost	Quantity	Number	Manager	Mass (kg)	Volume (mm^3)	Stock Size
1	1	E020501B	Steel	10	1	ZW-01	Ivy	7.44	953363.53	410.00x130.00x73.77(mm)
2	2	BEARING	Steel	5.9	1	ZW-02	Ivy	0.18	22858.07	13.00x80.00x80.00(mm)
3	3	E023001B	Steel	1.5	1	ZW-03	Ivy	0.28	35984.15	55.00x56.00x56.00(mm)
4	4	E052002A	Aluminum	3.2	12	ZW-04	Ivy	0.01	2192.45	20.00x20.00x7.00(mm)
5	5	E051502B	Aluminum	13	1	ZW-05	Leo	0.06	23088.58	26.00x132.01x132.00(mm)
6	6	E052501A	Steel	22	1	ZW-06	Leo	0.97	124341.03	62.50x138.00x138.00(mm)
7	7	E051001A	Steel	80	1	ZW-07	Sam	0.66	84135.22	8.00x245.00x244.98(mm)
8	8	E052502A	Steel	55	1	ZW-08	Leo	0.14	17718.83	134.32x24.00x134.32(mm)
9	9	DIN912-M6x25-8.8-YP	Steel	1.2	8	ZW-09	Ivy	0.01	940.66	10.00x10.00x31.00(mm)
10	10	NORDLOCK-M6	Steel	0.3	8	ZW-10	Ivy	0.00	110.64	11.00x11.00x1.80(mm)
11	11	ASSY piston+conrod	Aluminum	100	4	ZW-11	Leo	1.16	269962.80	
12	11.1	conrod	Aluminum	55	4	ZW-12	Leo	0.47	174198.78	106.00x106.00x59.70(mm)
13	11.2	E024501A	Steel	25	4	ZW-13	Leo	0.06	7148.33	105.60x105.60x15.25(mm)
14	11.3	E025002A	Steel	0.5	8	ZW-14	Ivy	0.00	185.39	1.50x25.40x25.40(mm)
15	11.4	E025001A	Steel	16	4	ZW-15	Sam	0.13	16274.94	76.00x23.00x23.00(mm)
16	11.5	E023505A	Steel	20	4	ZW-16	Sam	0.49	62768.38	23.00x74.00x179.50(mm)
17	11.6	E022004A	Aluminum	9.5	8	ZW-17	Leo	0.01	2213.26	19.00x51.79x25.53(mm)
18	11.7	DIN912-M8x1x35-12.9	Steel AISI 41...	2.5	8	ZW-18	Ivy	0.02	2387.53	13.00x13.00x43.00(mm)

Attribute	Descriptive Name	Include	Tot
Index	ID	☑	
Name	Name	☑	
Number	Number	☑	
Class	Class	☐	
Designer	Designer	☐	
Cost	Cost	☑	
Supplier	Supplier	☐	
Material	Material	☑	
Quantity	Quantity	☑	
Manager	Manager	☑	
Description	Description	☐	
Keywords	Keywords	☐	
Start Date	Start Date	☐	
End Date	End Date	☐	
Mass	Mass	☑	
Volume	Volume	☑	
Total Mass	Total Mass	☐	
Density	Density	☐	
File Path	Source file path	☐	

14.2 Mold

Align

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

Shrink

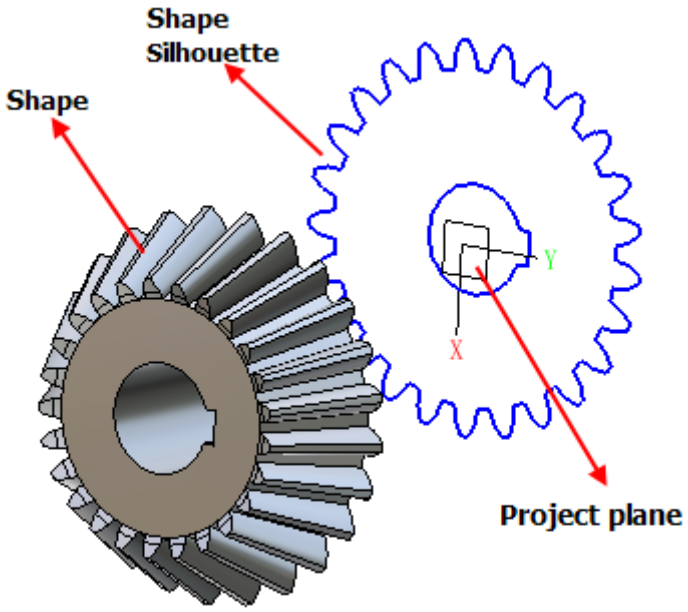
This command is used to define a product's shrinkage.

Region

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.

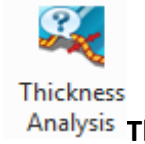
Project Silhouette to Plane

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.



Draft

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

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

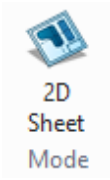
This command is used to record the result of the last time Height analysis. In this mode, the user can view the part with colors, and add PMI or annotation in those faces.

Tips:

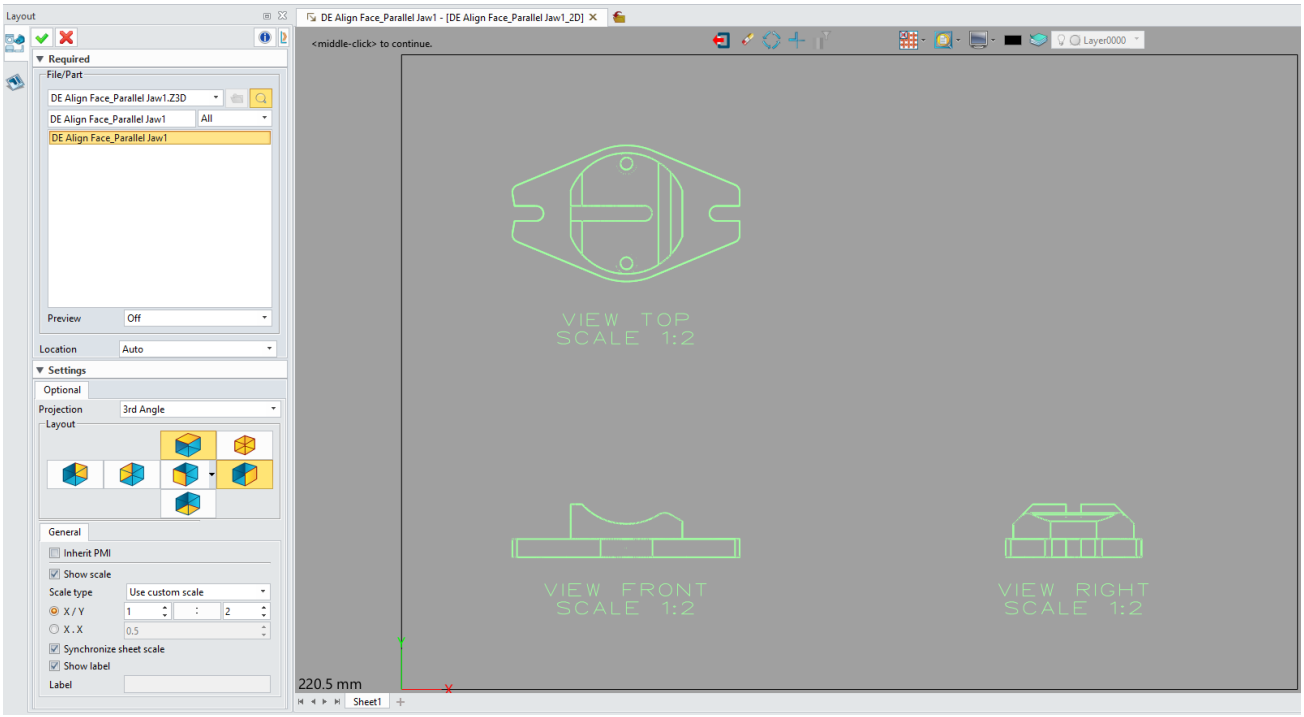
Before Height Analysis, the command will be gray out.

15.2D Drawing

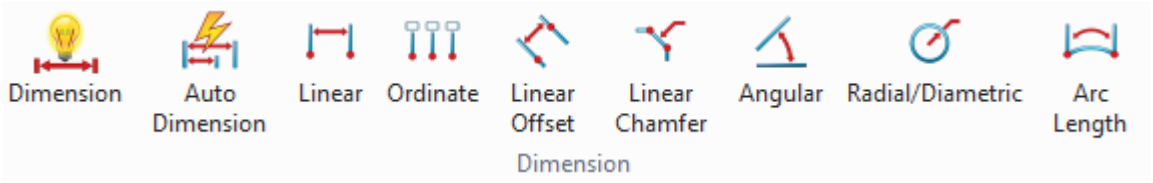
Using the 2D Sheet command from tool ribbon



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.

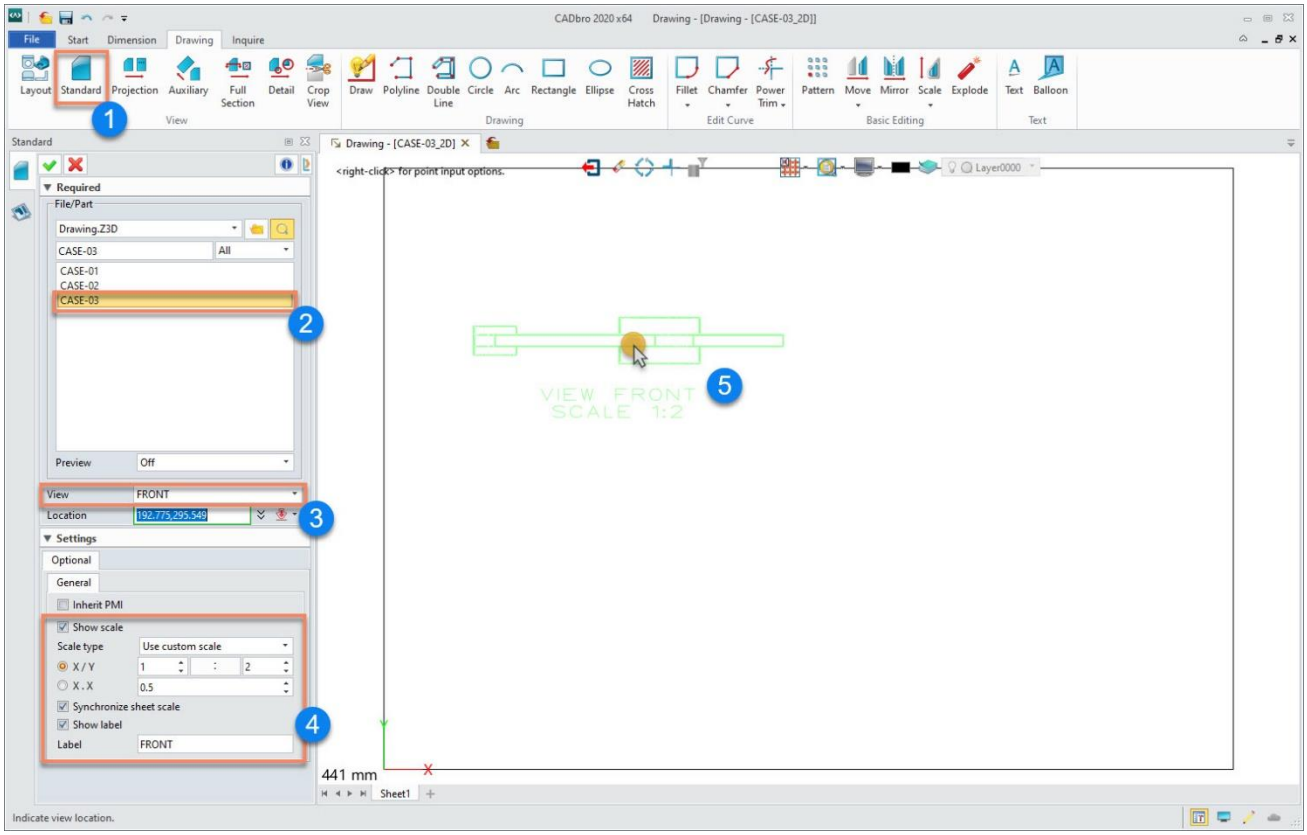


Standard View

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

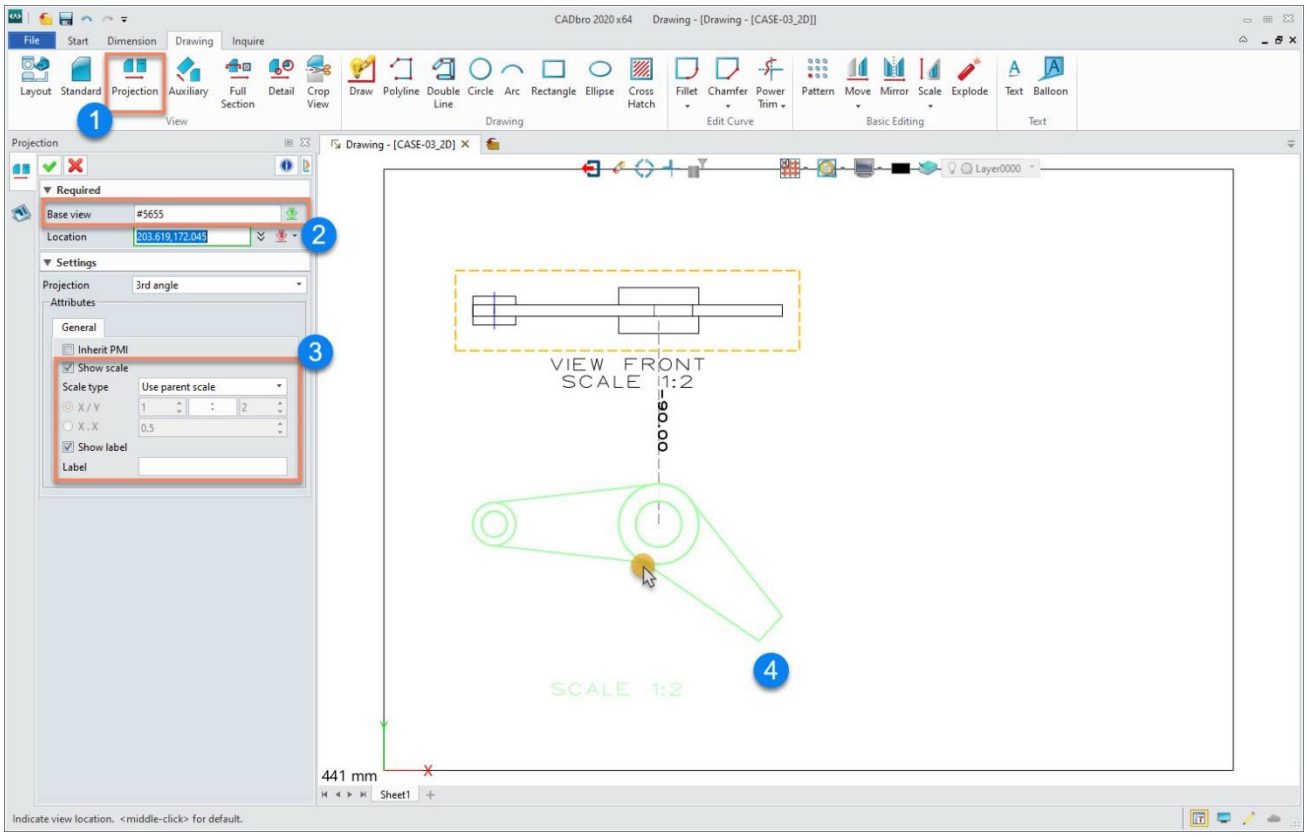
Use Projection View to create a view projected from another existing 3D layout view. Steps are as below:

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.



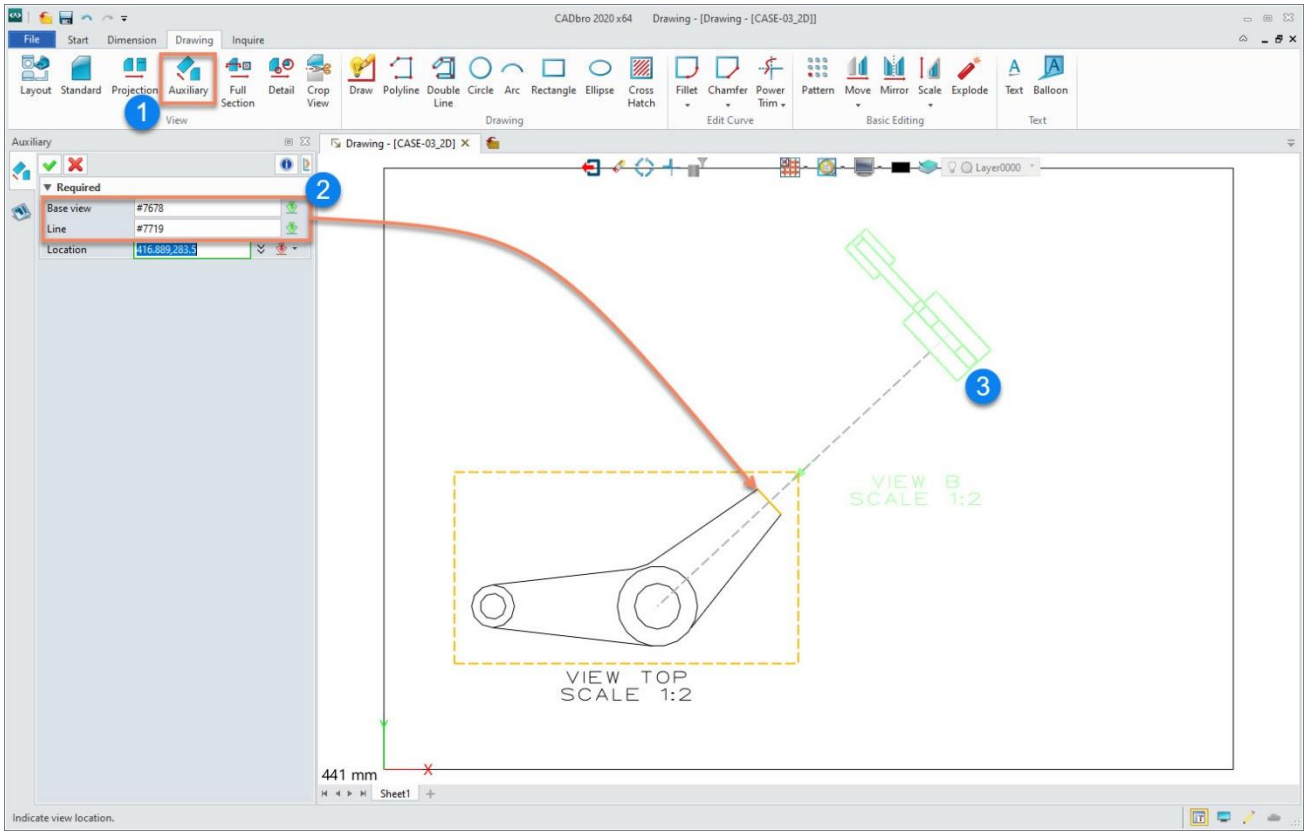
Auxiliary View

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.



Full Section View

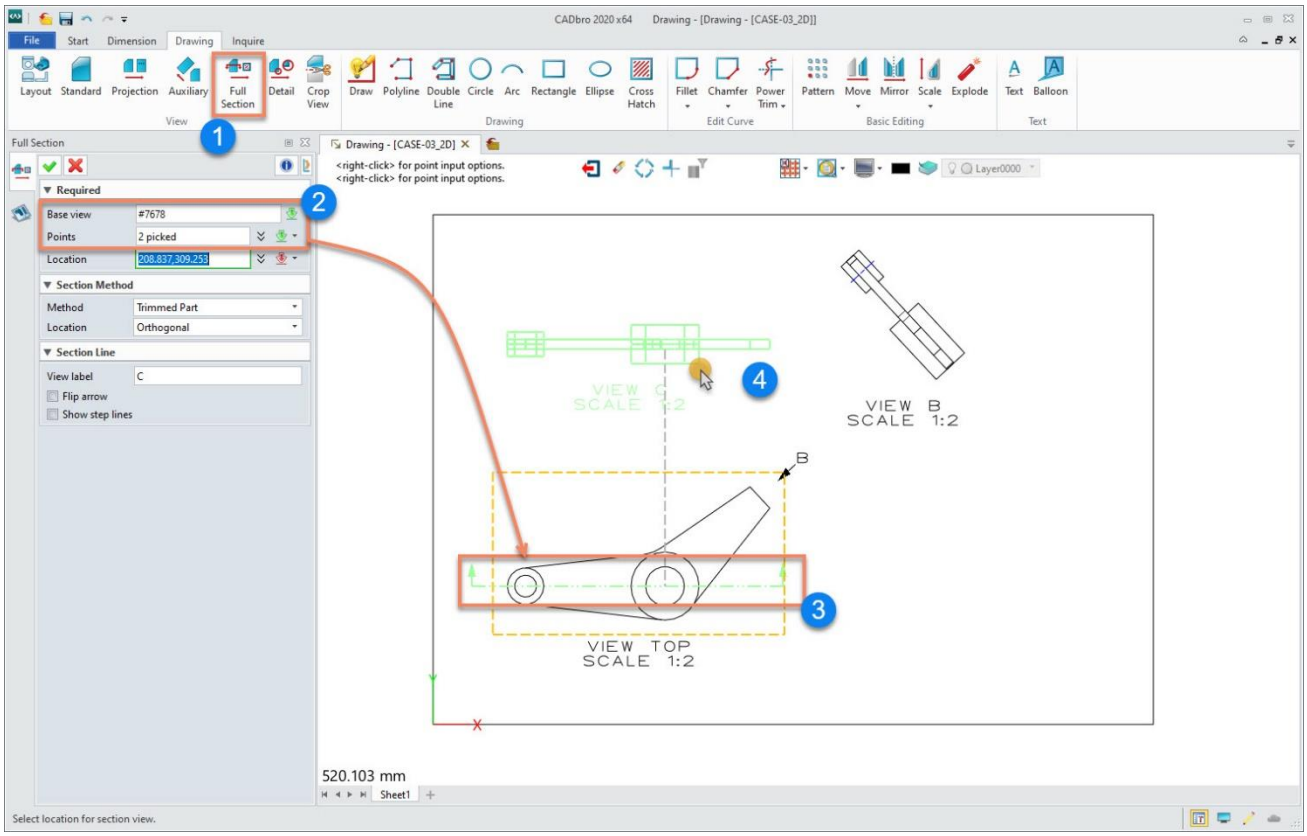
Use Full Section View to create a fully sectioned views base on a 3D layout view. Steps are as below:

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.

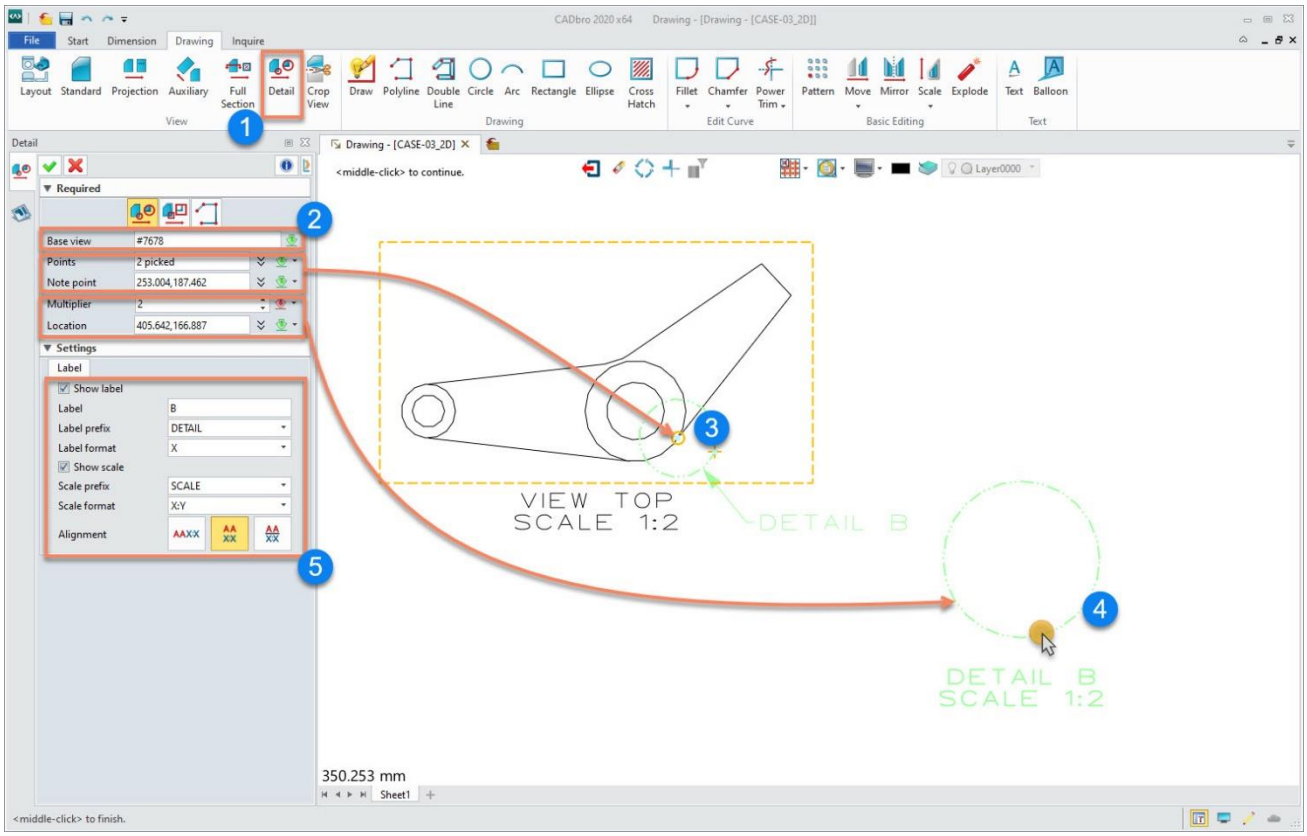
Step1: Pick the Detail View command.

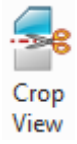
Step2: Select an existing view to generate a detail view.

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

Step4: Place the detail view on the work area.

Step5: Set the detailed parameters.

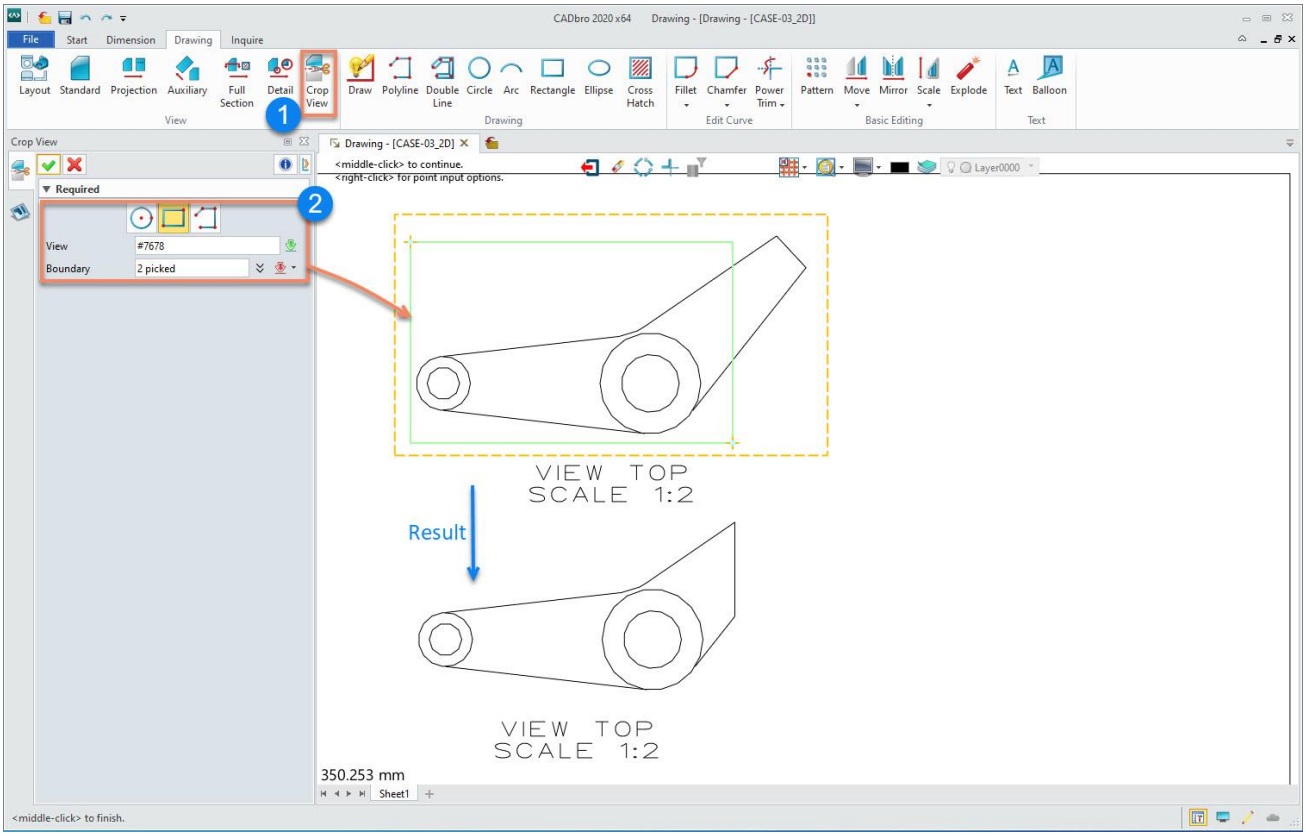




Crop View

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.

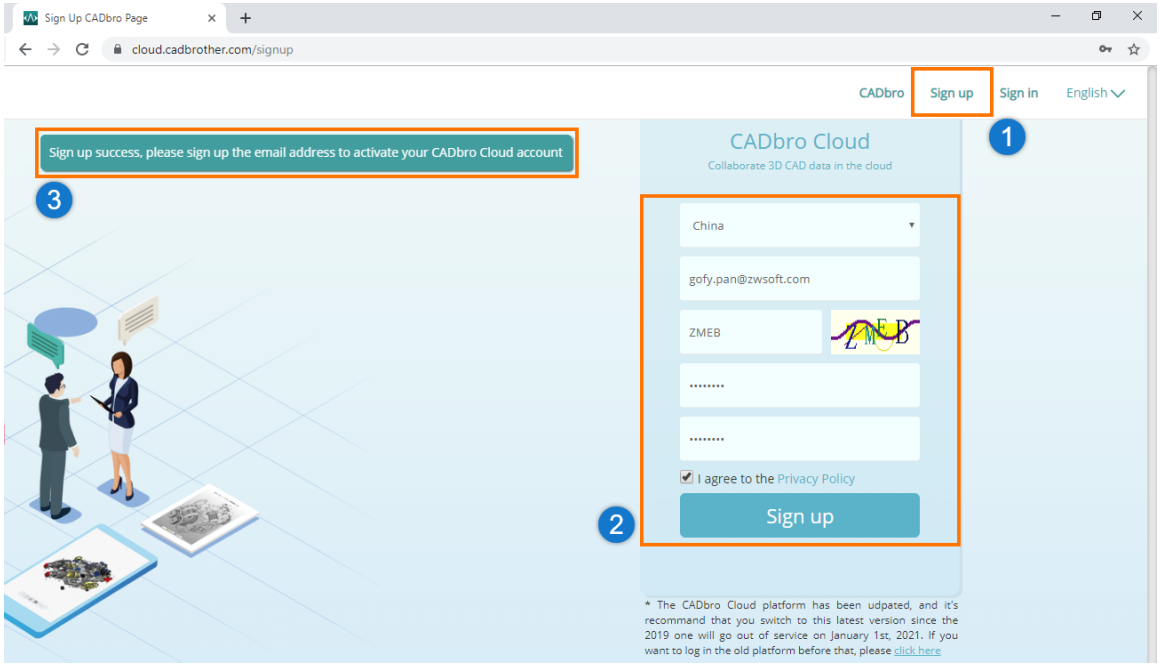


16. Cloud service

16.1 Create a cloud service account

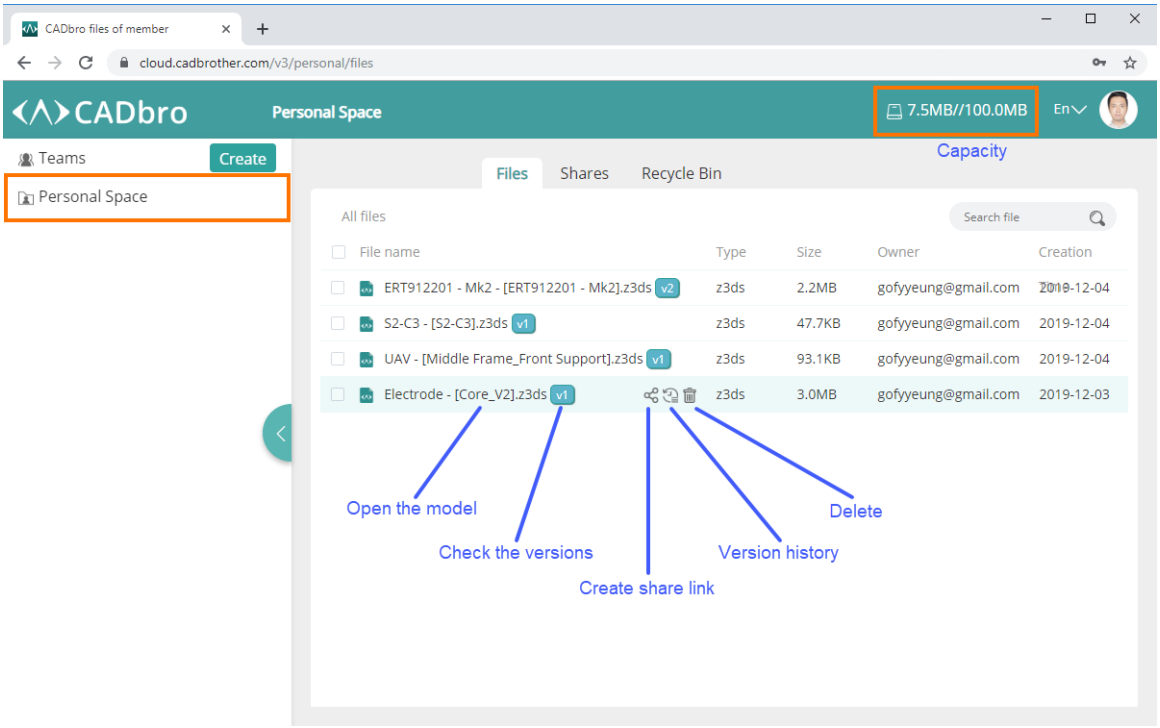
Before using cloud service, you need to sign up an account for CADbro cloud service. Please follow the below steps to create a CADbro could service account.

1) Go to CADbro Cloud website (https://cloud.cadbrother.com).
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.



16.2 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.



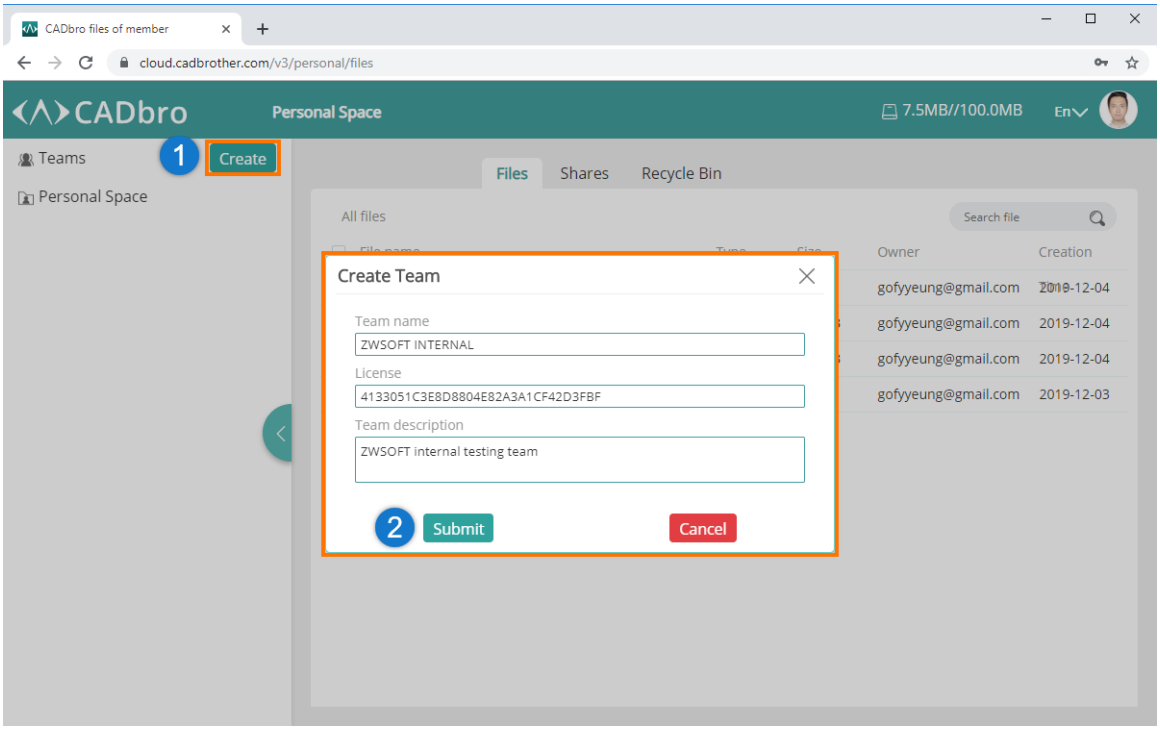
16.3 Team space management

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.

16.3.1 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.



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:

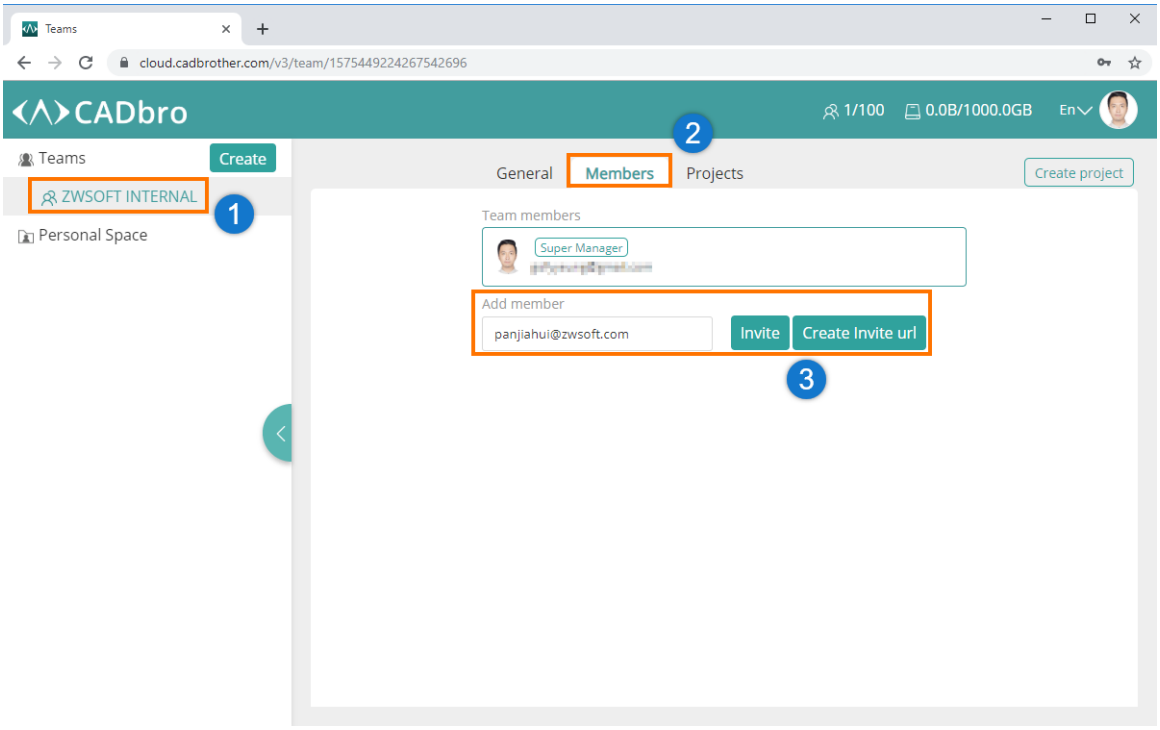
- change team settings such as the team name, description and reset the team
- invite members to join the team or remove members from the team
- create projects
- assign roles (manager, member and visitor) to members
- manage all the files in the projects

16.3.2 Invite members to a team

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

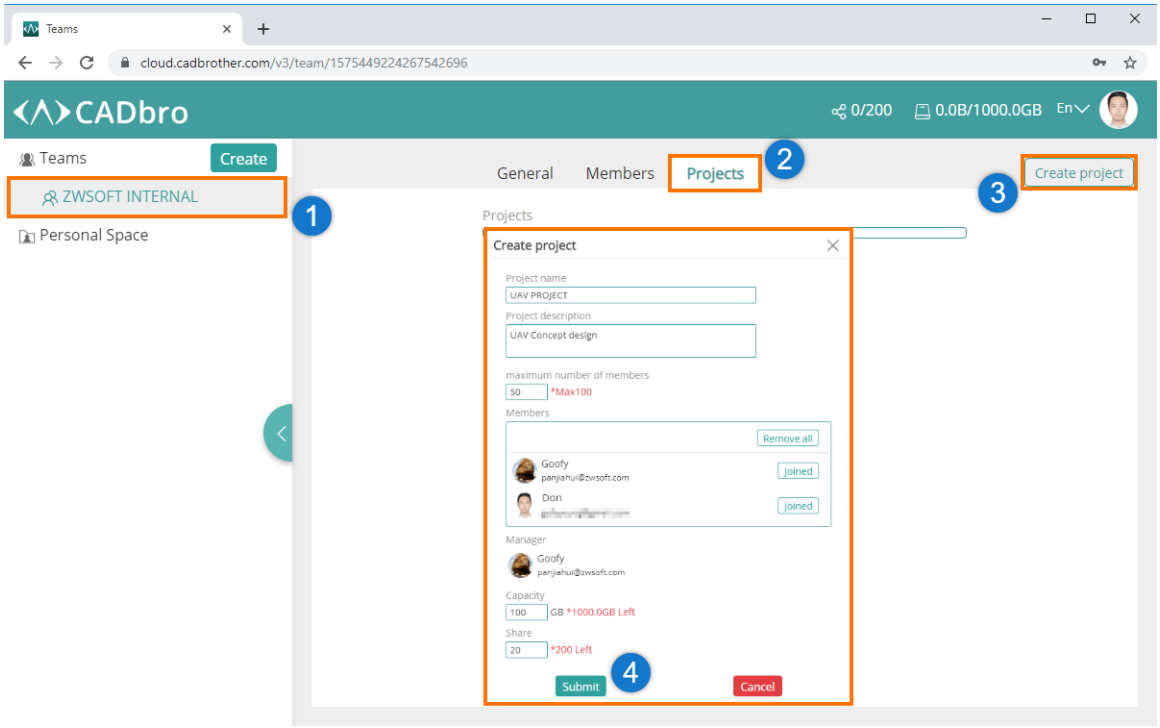
Email: Send an invitation email to members. The members who are invited need to accept the invitation via email.

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.



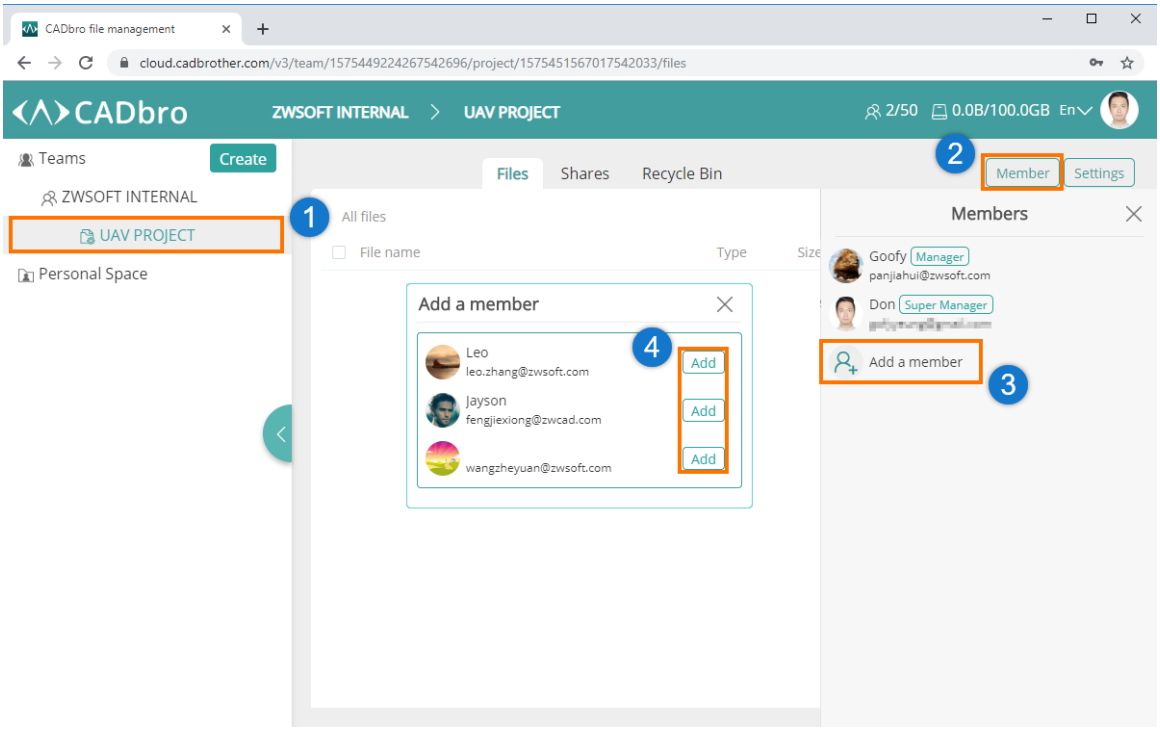
16.3.3 Create a project

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.



16.3.4 Add members to a project

Super manager or manager can add members to a project.



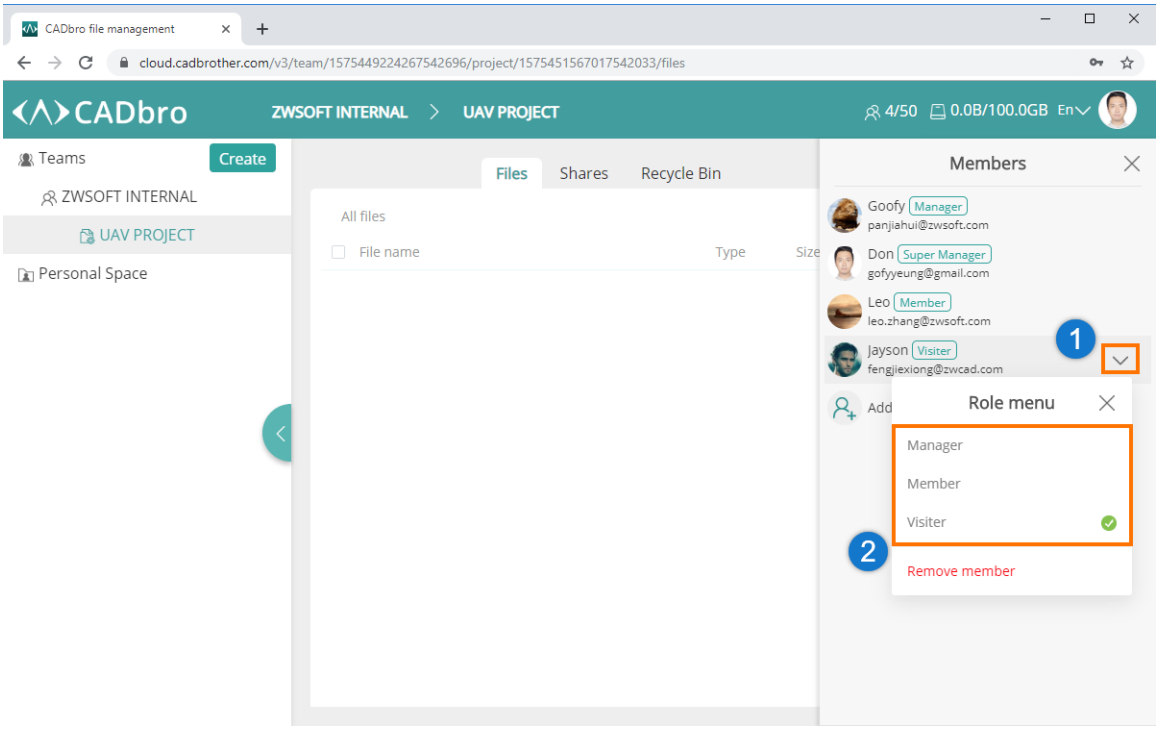
16.3.5 User roles

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.

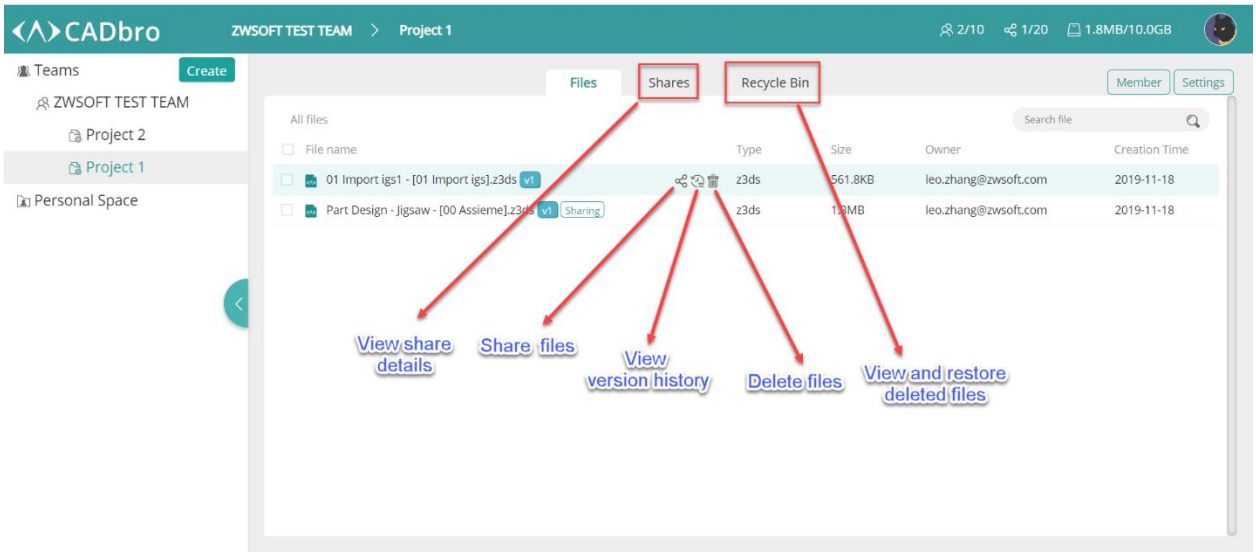
Visitor is only allowed to view models.



16.3.6 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.

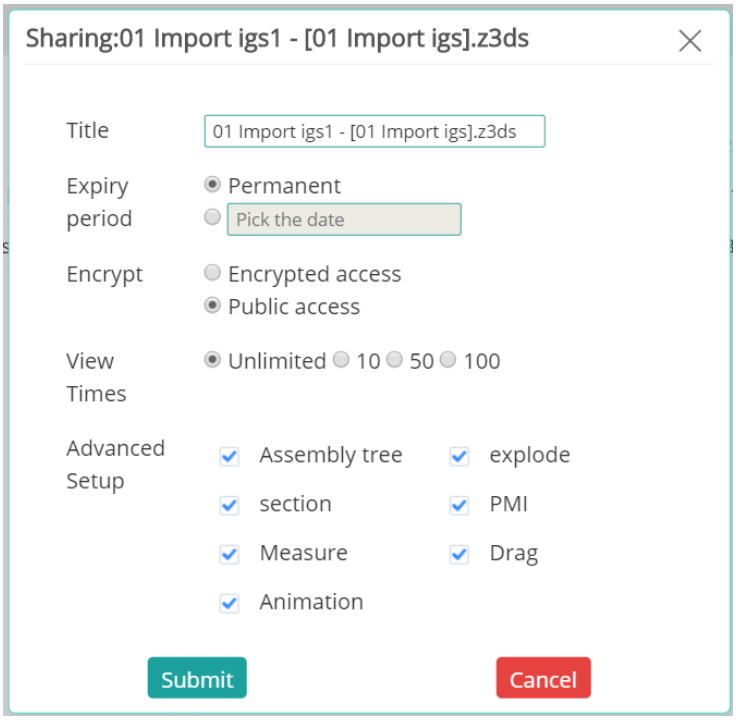
Note: Uploading is only supported in desktop client currently.



➢ **File share**

Steps for creating a share link.

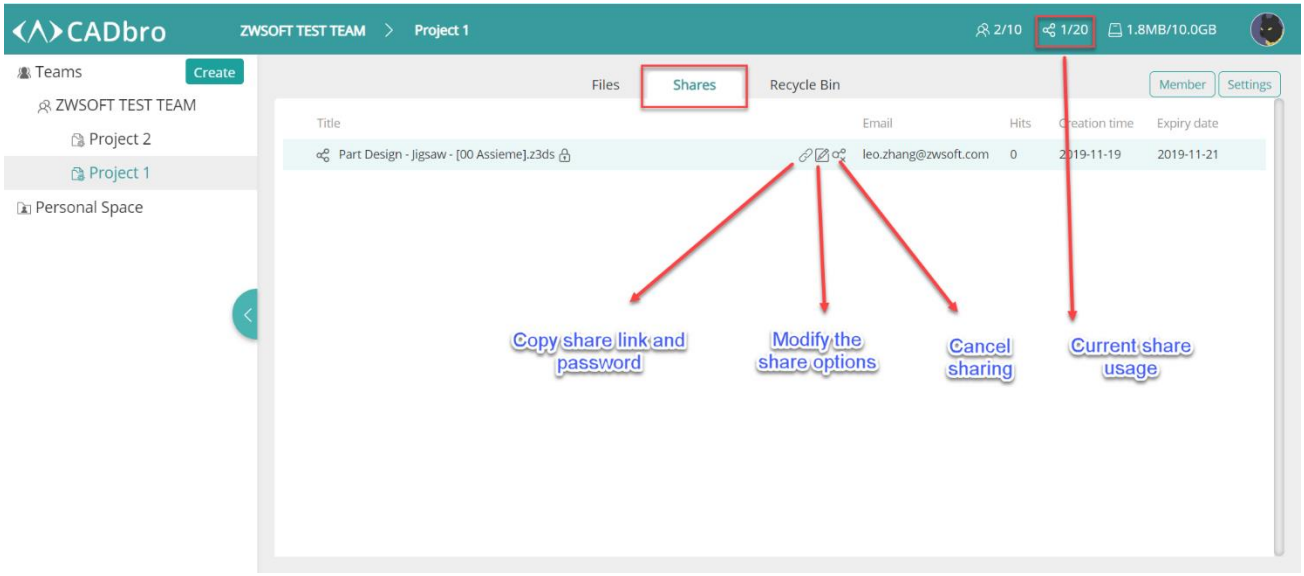
1) Click icon near the file to be shared.
2) Set the sharing options, such as expiry date, password, view times and so on.



3) Click “Submit” to finish.

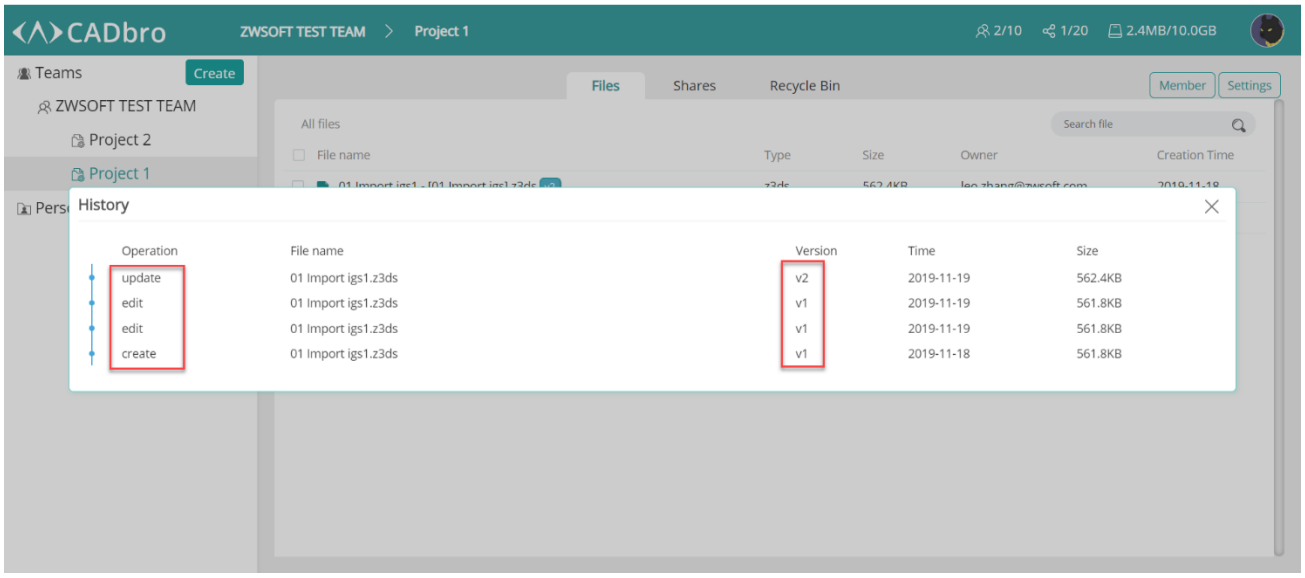
➢ **Management of shared files**

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.



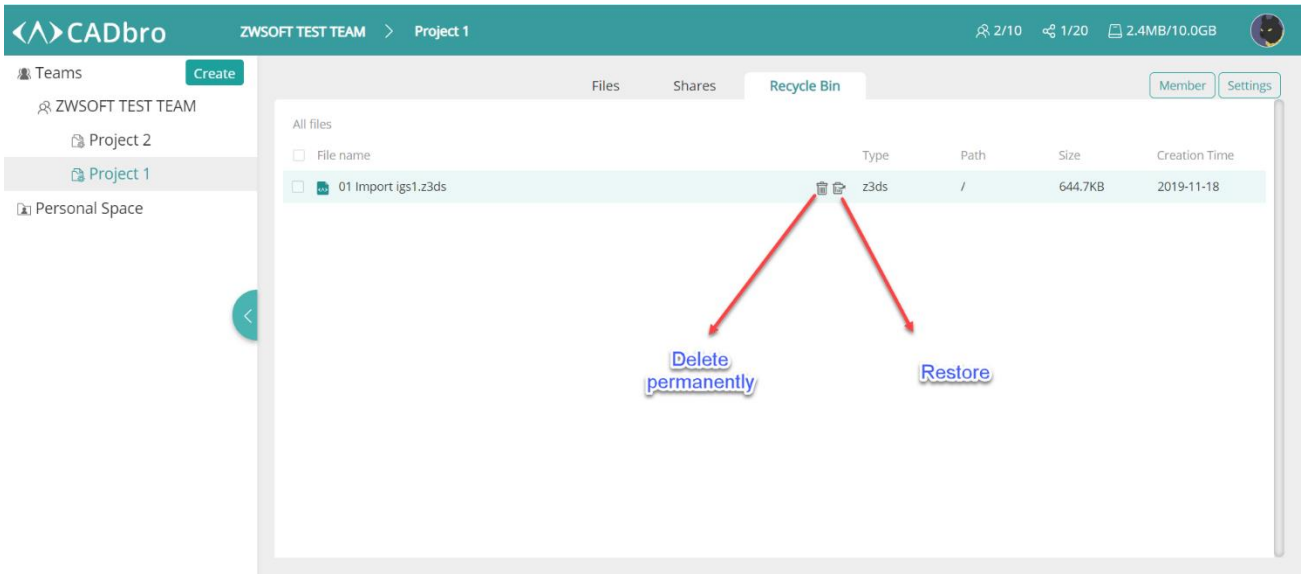
➢ **File history**

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.



➢ **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.



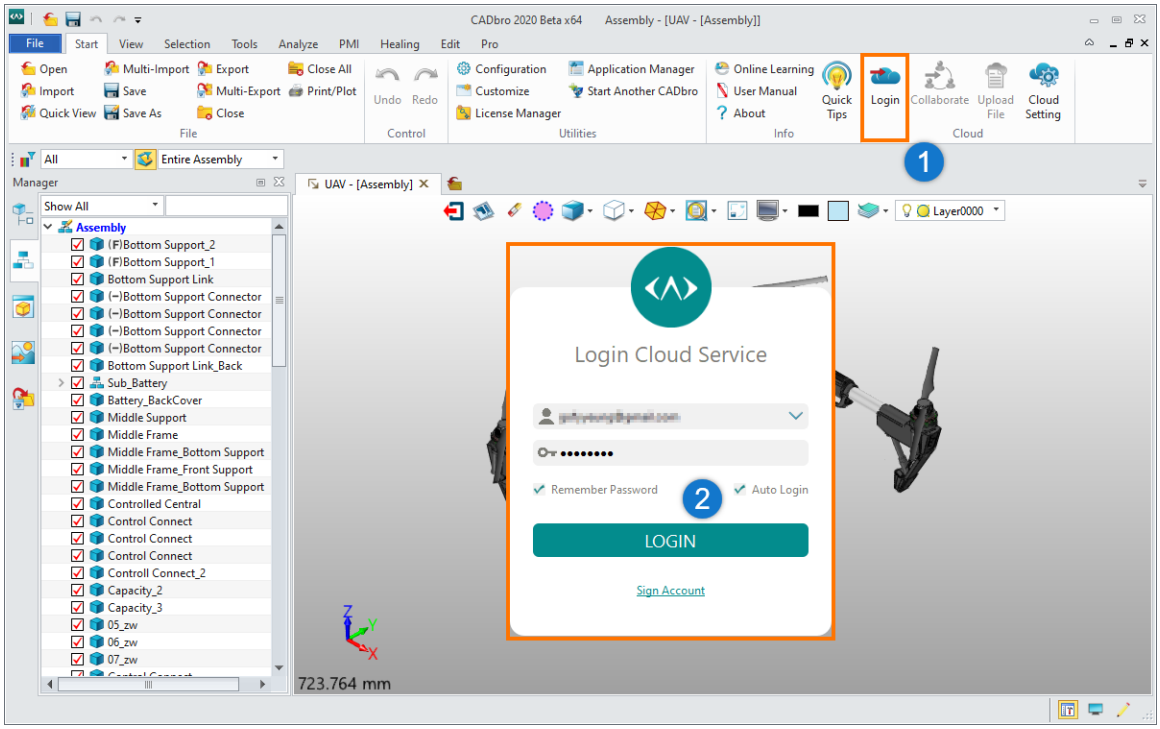
16.4 Collaboration

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.

16.4.1 Login CADbro

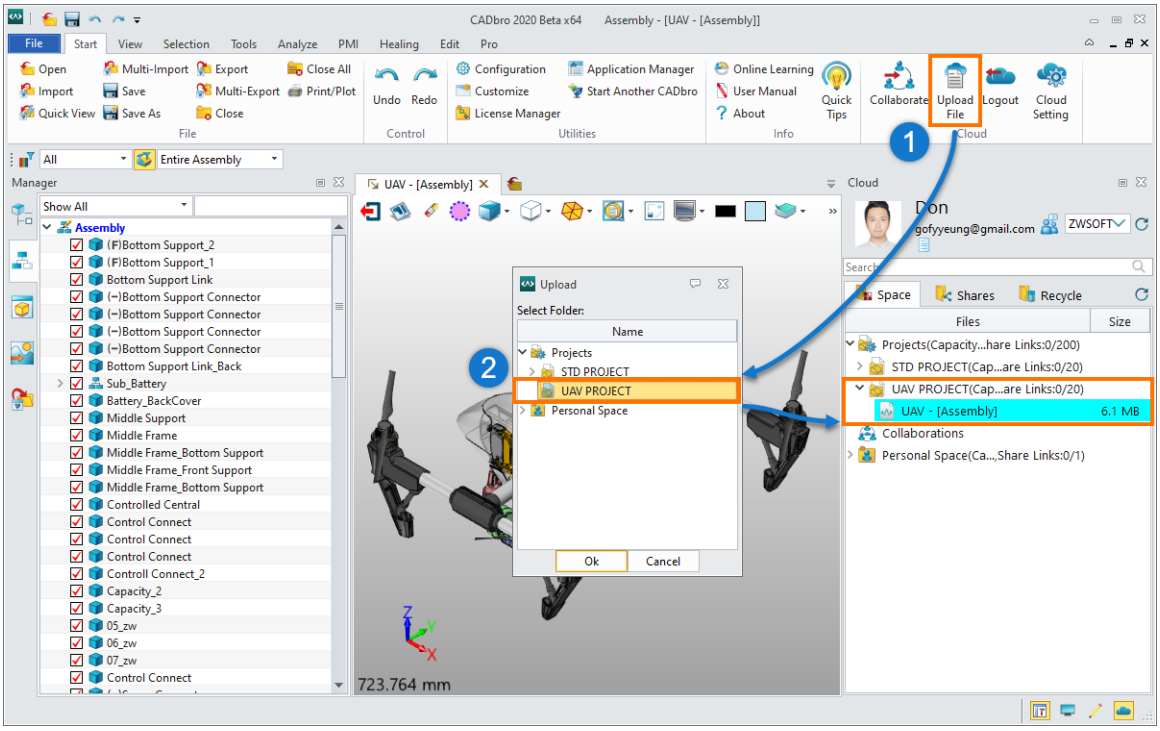
Steps for logging in CADbro Cloud in desktop client

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



16.4.2 Upload models

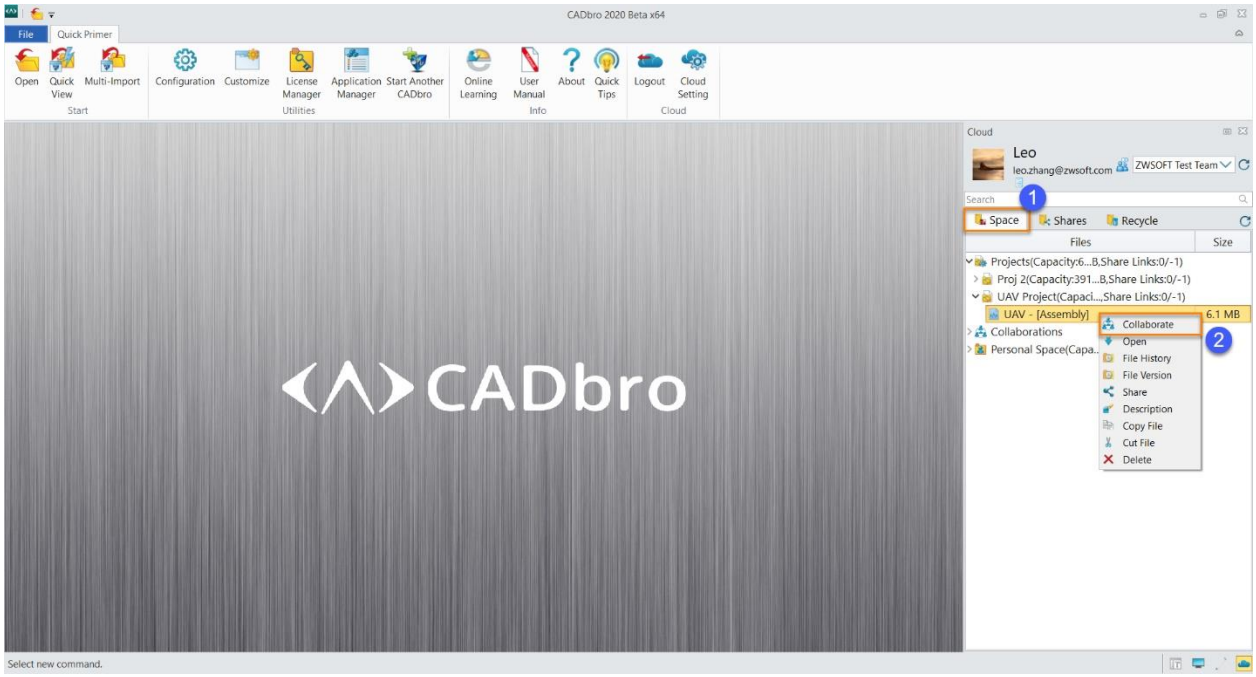
Open the file in CADbro, click Upload File command and select a project it will be uploaded to. Note: CADbro will convert the model into light-weight format (.z3ds) and upload it to the Cloud server.



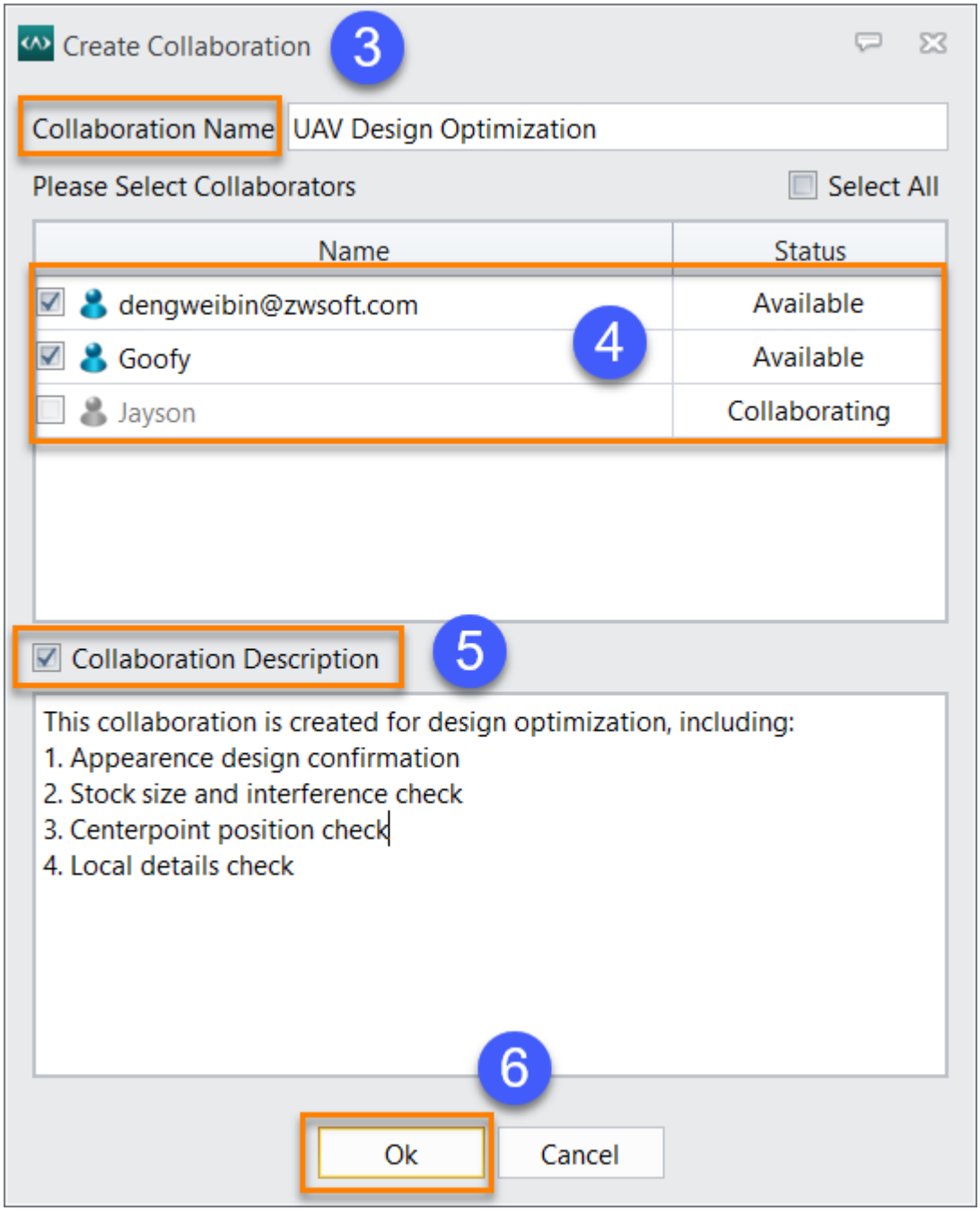
16.4.3 Start collaboration

Steps for starting collaboration:

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

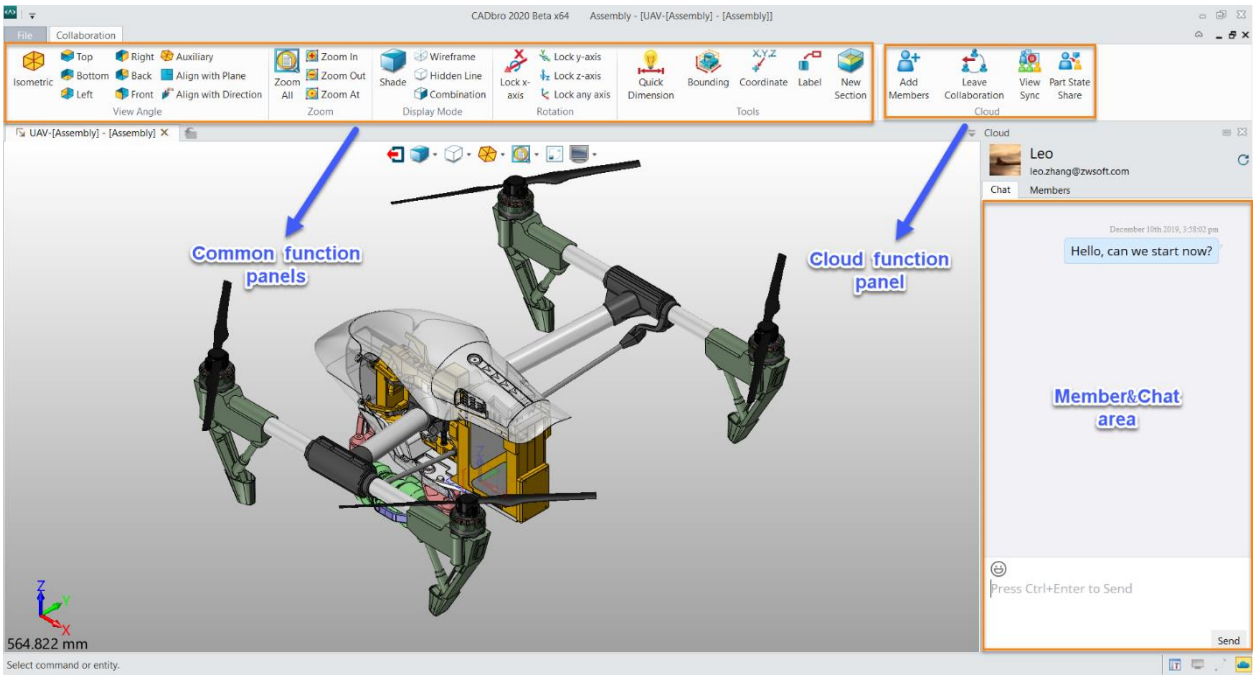


3) Input a name for the collaboration.
4) Select members to join the collaboration
5) Describe the collaboration if needed.
6) Click “OK” to start the collaboration



16.4.4 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.

3) **Part State Share**

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.

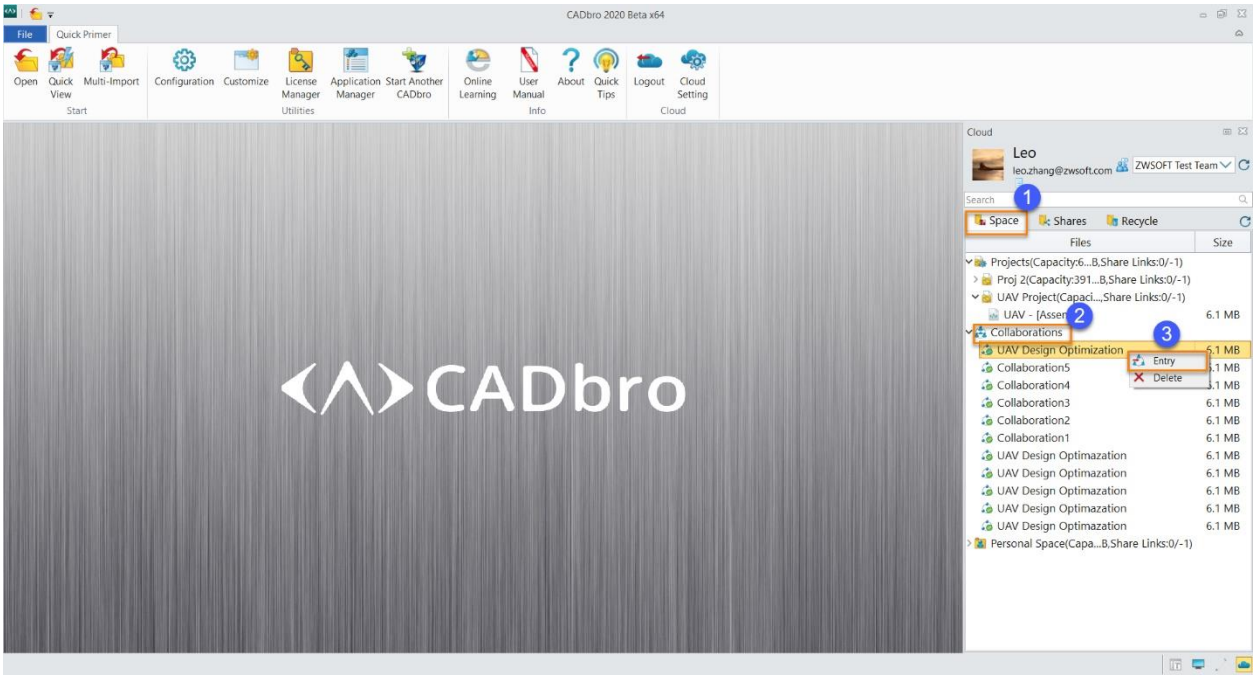
16.4.5 Playback

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.

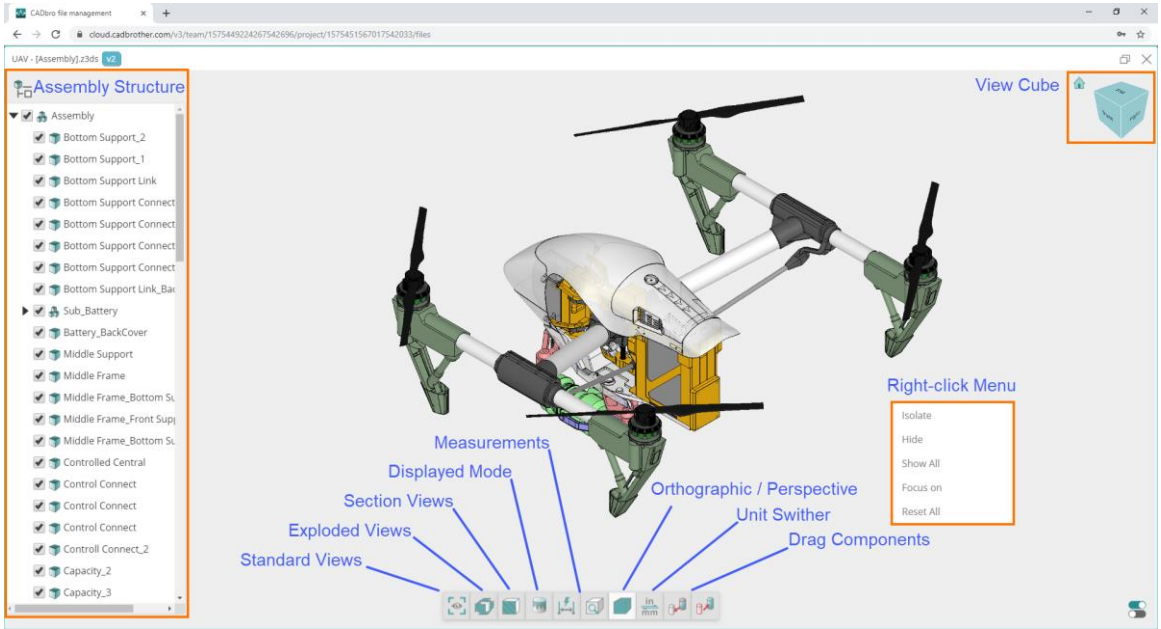
Step3: Review the playback record. There are many options for users' better experience.



16.4.6 View 3D models in browser

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

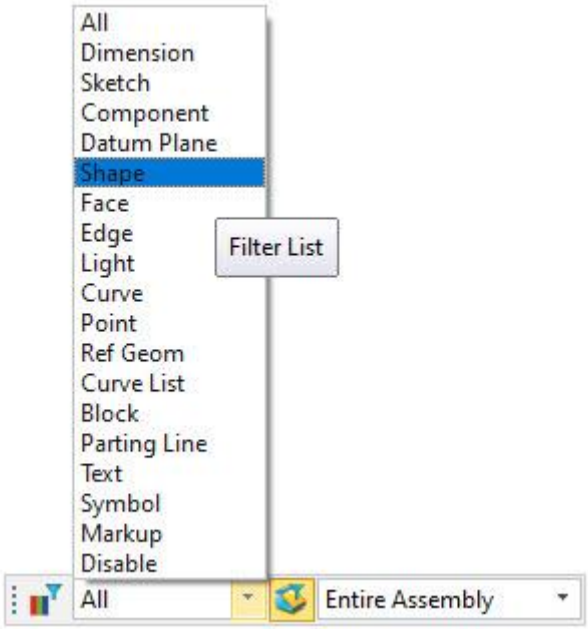
- View 3D models
- Check assembly structures
- Show or hide components
- View explosive view
- View by sections
- Do measurement
- Move components
- ……



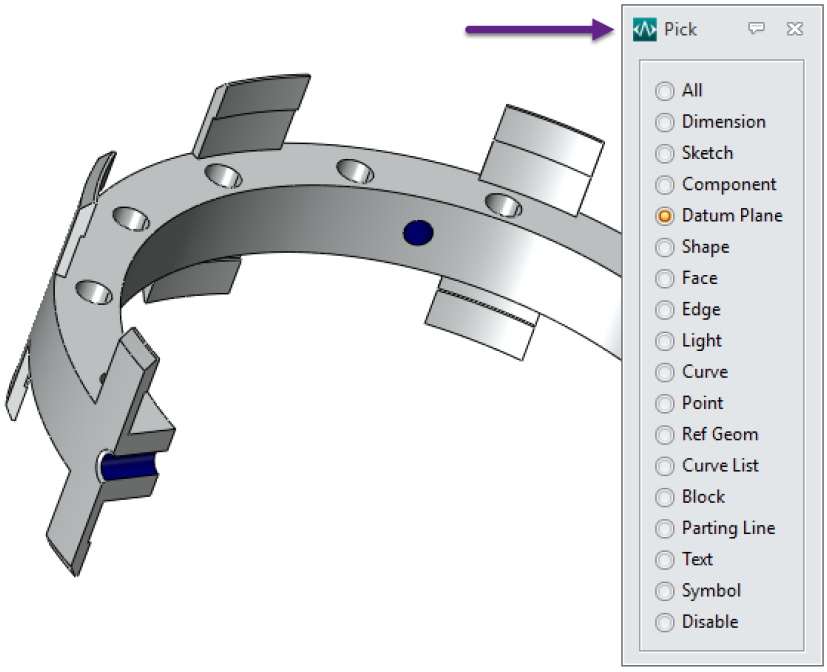
17. Filter

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.

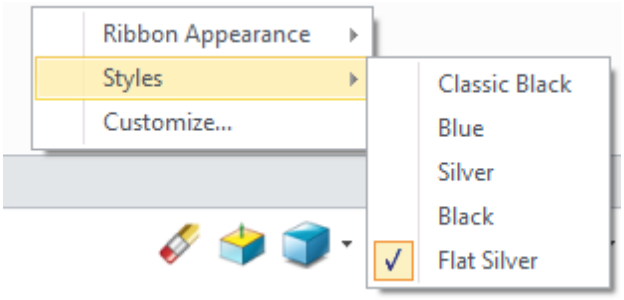


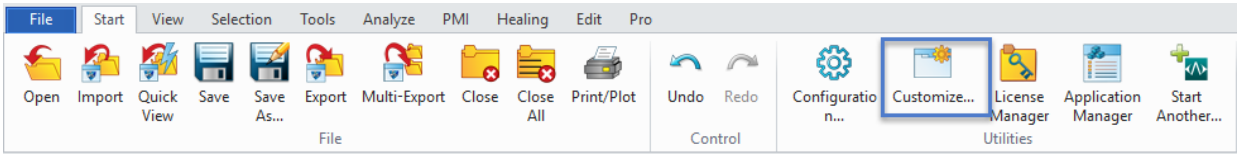
Holding the Shift key + Right mouse button, the filter list will display on your mouse pointer position.



18. Interface customization

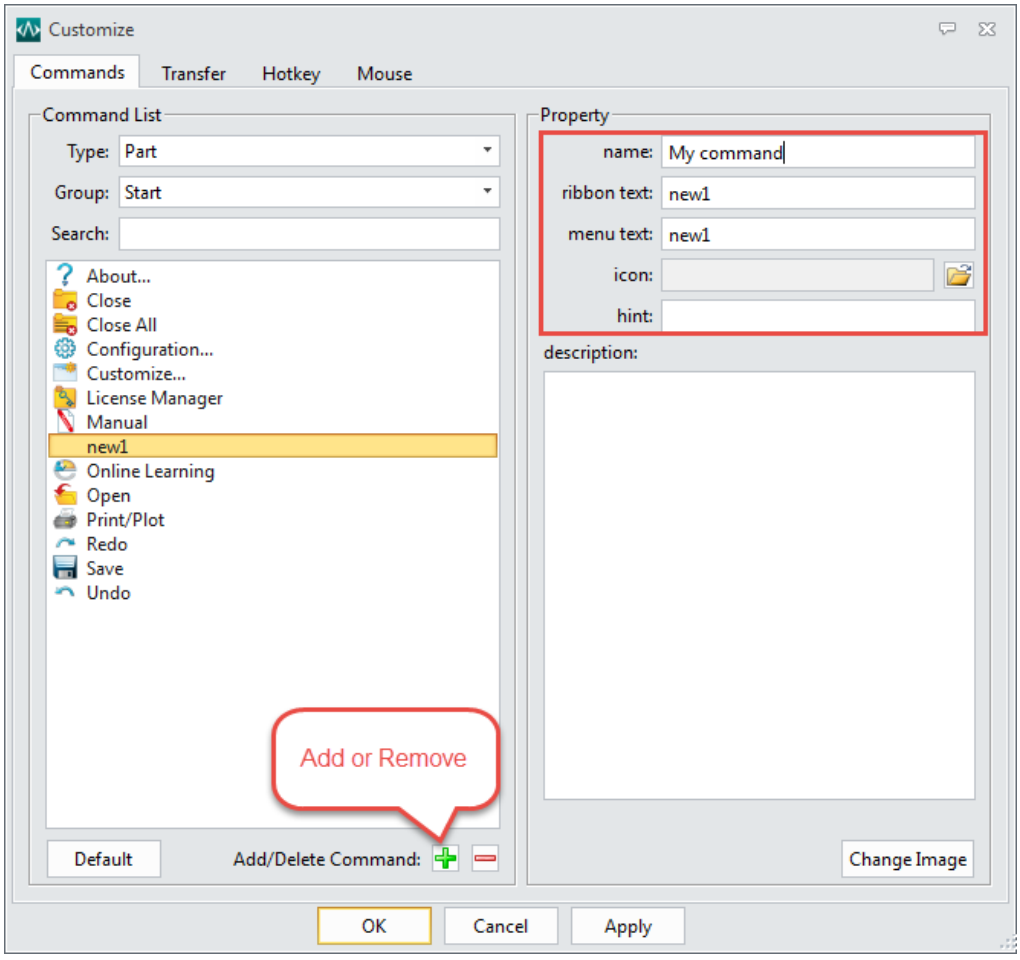
You can right mouse button click on the ribbon empty area, then you can see interface customization command. Or you can directly click the Customize… command from the Ribbon tab.



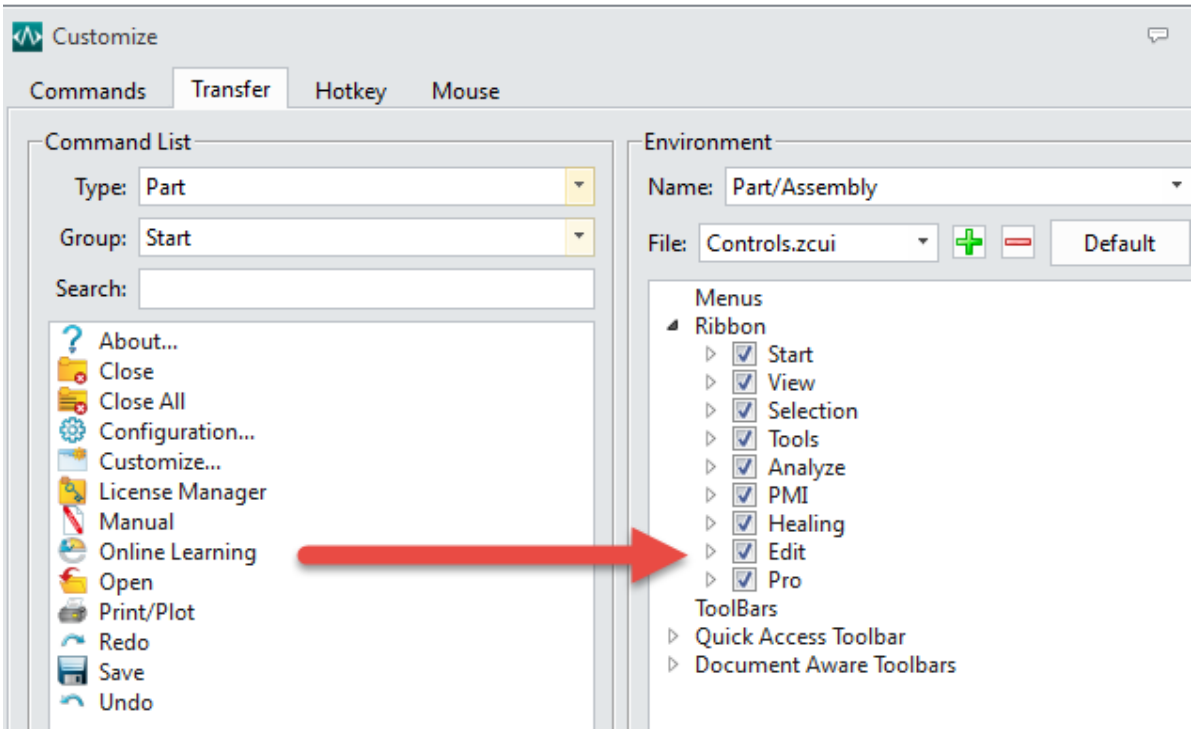


What you can customize?

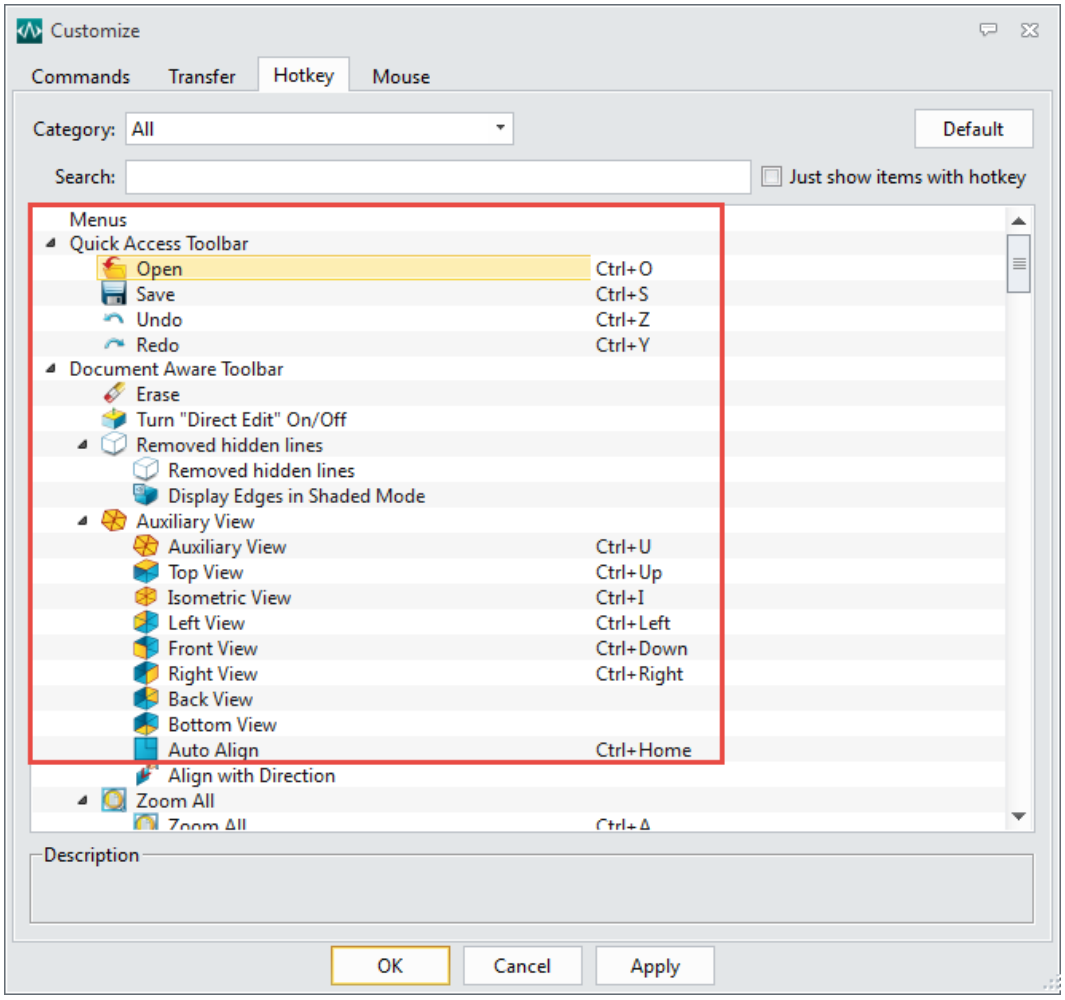
You can add your own command.



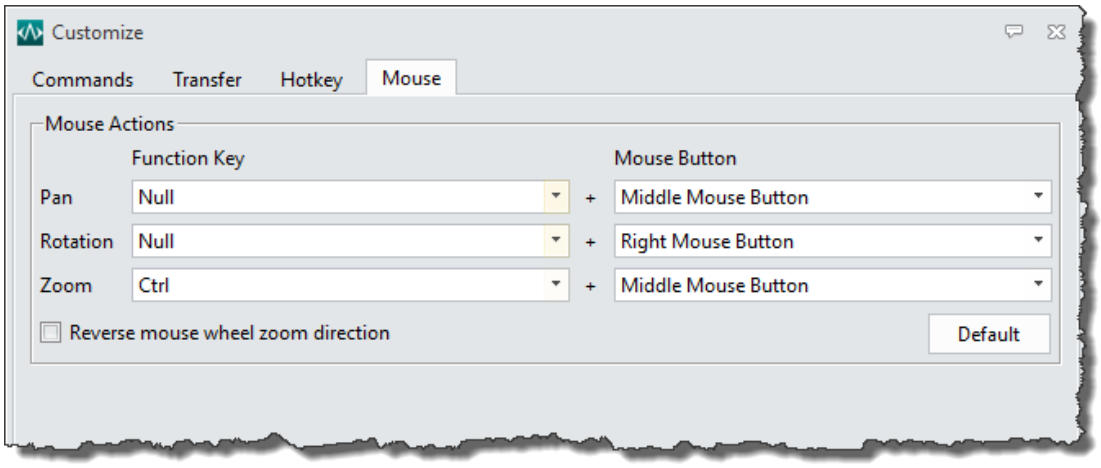
Transfer the command between different ribbon tab, you can also create a new ribbon tab.



Customize the hotkey for CADbro.

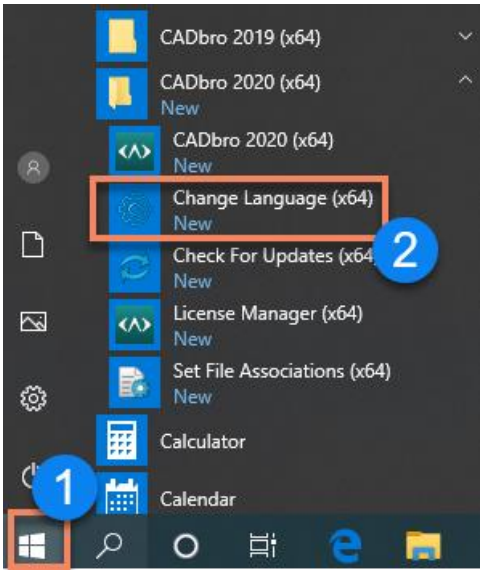


Customize your mouse action, with this function below, you can customize CADbro mouse actions to be the same as other 3D systems, this will make you easy to get used to working with CADbro.



19. Changing language

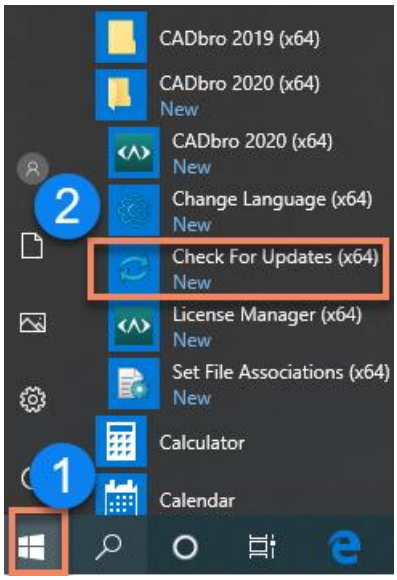
Click the Windows button, find the ZWSOFT folder and click the button Change Language.



Then you can choose one to change the installation language

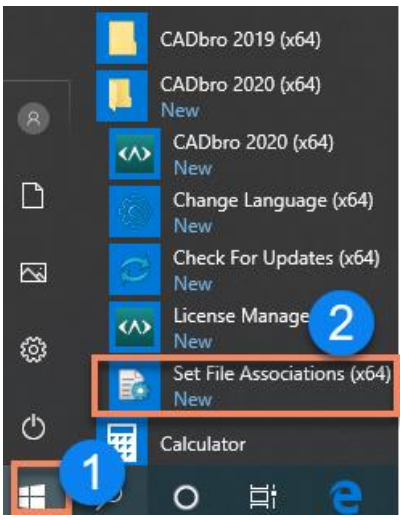
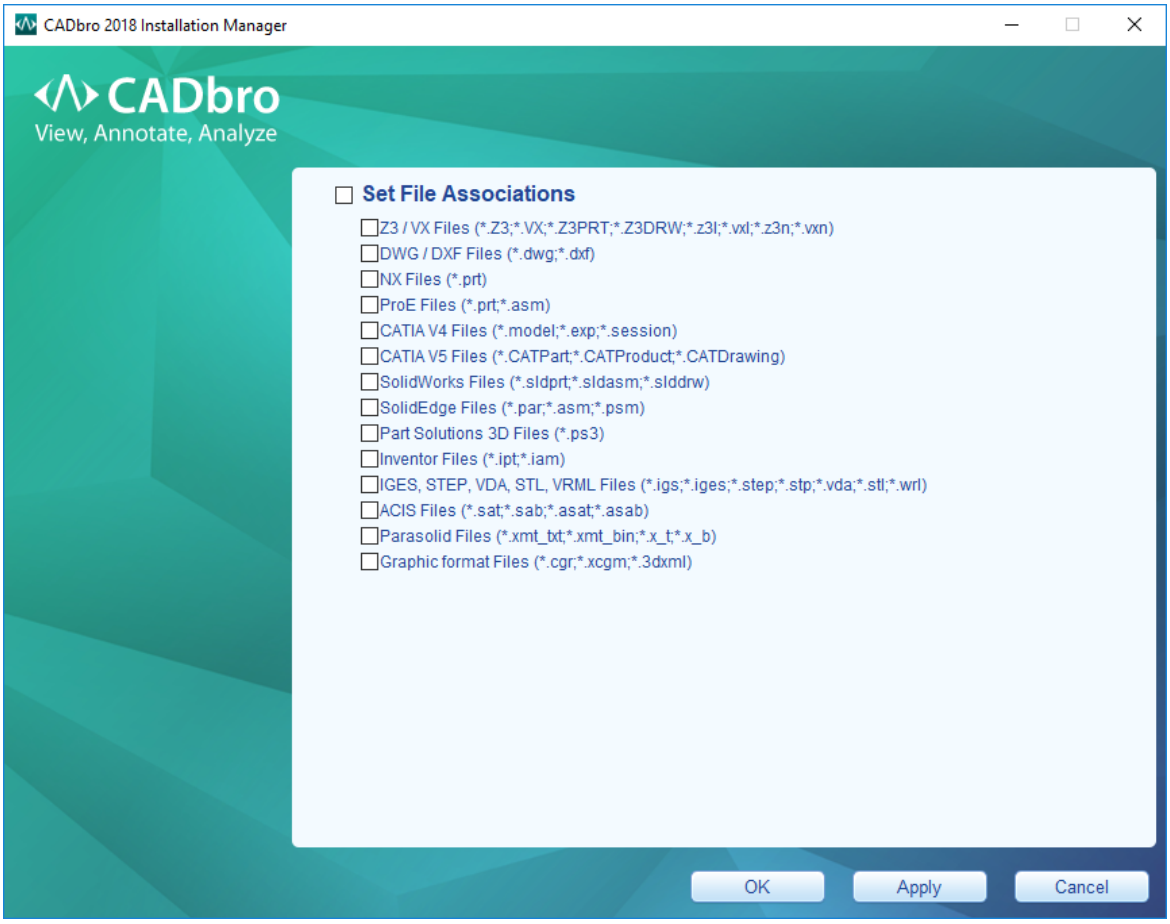
20. Online update

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



21.Set File associations

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 CAD bro.