



CAXA Draft

User's Manual

<http://www.caxa.com>

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Chapter 1 Preface

1.1 About CAXA

CAXA is the market leader and largest Chinese provider of PLM solutions and services. Based on the mission of "software service manufacturing industry", CAXA has developed a suite of products in design, process planning, manufacturing and collaborative management that tailor domestic practices of Chinese industry. CAXA has received the highest "Software Gold Award" from China Software Industry Association. CAXA has more than 280,000 installations in China through well-established channels including 35 sales and service branches, 600 training centers and 300 business partners.

1.2 About CAXA Draft

As the first enterprise specialized in CAD software development in China, CAXA devotes to popularizing and applying this design software for years, and extricating designing engineer from complicated drawing, which help enterprise to innovating constantly. CAXA Draft is a tool that helps to develop new products and design drawings within short time.

1.2.1 Function

The features of CAXA Draft is shown as follows:

- (1) Novel interface with alternation taste.

In Mechanical version of CAXA Draft, the previous menu mode for "file", "edit", "view" is changed, and popular Fluent/Ribbon interface is adopted, the new interface is more simple and direct, user can finish CAD operation more easier, furthermore, original CAXA style interface is kept, press F9 key to switch to the former interface. Parallel alternation technique, dynamic navigating and double-click editing are optimized in CAXA Draft, then user can operate more flexible.

- (2) Compatible with AutoCAD all-around.

Mechanical version of CAXA Draft is re-developed on the base of Unicode, compatibility with AutoCAD data is enforced, in order to switch data cross platform and cross language, and EXB file in CAXA Draft can be switched to DWG file directly, then historic data in enterprises can be compatible. CAXA Draft supports mainstream platform system, operation performance is improved.

(3) Professional drawing tool with international dimension style.

CAXA Draft has complete drawing create functions, including basic point, line, polyline, arc, rectangle, ellipse, formula curve etc. In addition, the system has edit functions of cut, vary, stretch, array, transit, paste, modifications of characters and size etc. The system offers powerful engineering annotations in dimension, coordinate, text, dimensional tolerance, tolerance, roughness etc, all dimension detail will be finished automatically in the system.

(4) Parametric library setting and software design aids

As per the request of mechanical design, Mechanical version of CAXA Draft CAXA supply parametric library that meets the novelist international standard. There are altogether 20 large categories, over 1000 categories, and 30000 pieces of dimension icon. User can enlarge library in complete open library management, what's more, aids tool will be supplied when query, calculate or switch. It meets drawing request under different circumstance.

1.2.2 Running environment

Running environment: Windows XP/Vista; P4 2.0G CPU; 512M RAM.

1.3 About the user's manual

This Manual explains the usage of every command in CAXA Draft together with appropriate examples to demonstrate operation procedures, and to achieve design objectives.

Each command and its corresponding operations are introduced in relative chapters with simple examples when necessary. Command and function can also be found

according to index in the appendices. A few real applications in latter chapter will illustrate the complete steps from scratch to finish using CAXA Draft smoothly.

User can press F1 key to refer to Online Help in CAXA Draft software. Press F1 key when different command or dialog box is being run, and the pop up online help will be positioned to corresponding section. Several option card for query is supplied on the left side of the online help window.

(1) **【catalog】**

- Usable file introduction is shown in the form of listed topic and sub-topic.
- Allow user to browser via select and spread topic.
- User can know his position, and step into other topic quickly through a structure in the help system.

(2) **【Index】**

- Key word related to topic in the Catalog option card is listed in alphabetical order.
- If a certain function, command or operation name is understood, or user needs to know what it executed, please user INDEX option card to visit its information quickly.

(3) **【Search】**

- Key word searching for all topics is listed in the Content option card .
- It will display the input word topic list included in key word field.
- Click title on title list and position list, the arrange result will be listed in alphabetical order accordingly.

User can visit our website to know CAXA product in detail.

Chapter 2

Quick start

This Chapter will explain how to start working with CAXA Draft, enabling you to master all the basic operations. The chapter guides you through installing and uninstalling the software, explains the user interface, basic operations, file operations, and view manipulation.

2.1 Install, uninstall and starting CAXA Draft

2.1.1 Notes

Please read the following notes before installing or uninstalling the software:

- (1) Check your system for the minimum system requirements mentioned in section 1.2.3 and be sure that you have Windows Local Administrator rights.
- (2) Please disable the UAC while installing CAXA Draft on any Microsoft Windows Vista system.
- (3) We strongly advise you to close all other applications while installing or uninstalling CAXA Draft.
- (4) It is also strongly recommended to close all other CAXA programs while installing or uninstalling CAXA Draft.

2.1.2 Installing CAXA Draft software

Insert the CAXA Draft CDROM into the drive. The Welcome dialog box should pop up automatically. Click the corresponding button to run CAXA Draft installation. If the Welcome dialog box doesn't pop up, please open the CD drive using Windows Explorer, find Autorun.exe and double click it to start the Welcome window.

Install process:

- (1) Select the language to install, and click OK to continue, or click Cancel to exit.
- (2) The Welcome dialog box appears, Click Next, Continue to the setup, or click Cancel to exit.
- (3) The Agreement dialog box appears, if agreed click Next to continue setup, otherwise click No to exit
- (4) The CAXA Draft special explanation for setup appears. Please read and click Next to continue setup.
- (5) The user information dialog box appears; input your name and your company.
- (6) Set the folder to install the software to, the default path is C:\Program Files\Caxa\CAXA DRAFT \2009. Click Browse to specify a different location.
- (7) Select the components to be installed; check the corresponding check box to select a component and click Next to continue.
- (8) Set CAXA Draft icon folder in the Start menu if required and click next.
- (9) The installation confirmation window appears, when the above operations are confirmed, click Next to install.
- (10) Click Finish when the installation is completed.

2.1.3 Uninstall CAXA Draft

Follow these steps to uninstall CAXA Draft :

- (1) Open the Windows 'Control Panel', double click 'Add/Remove Programs'.
- (2) In the 'Add/Remove Programs' dialog box, select CAXA Draft.
- (3) Click 'Remove'.
- (4) The CAXA Draft's maintenance wizard appears, click 'Remove'.
- (5) Continue to remove the program as per the wizard instructions.

2.1.4 Run CAXA Draft

There are three methods to run CAXA Draft.

- (1) CAXA Draft shortcut icon will appear on the Windows Desktop once it is installed correctly, double click it.
- (2) Click Start at left bottom corner of your screen, then select program -> CAXA Draft2009 -> CAXA Draft.
- (3) Double-click Caxa.exe from the CAXA\CAXADraft\bin\ folder (or the location where it is installed).

2.2 User Interface

The User Interface (UI) is the intermediary graphics that facilitates communication between you and the software. The CAXA Draft UI has two different styles; the latest fluent style and the classic style. In the classic UI, you will be able to reach common commands only through the main menu and toolbar. In the Fluent UI, you can reach common commands through the function area, quick start toolbar and also the menu buttons.

Additionally, the fluent UI includes a status bar, immediate menu, drawing area, tool option panel, command row, etc. The two interfaces of CAXA Draft are shown below.

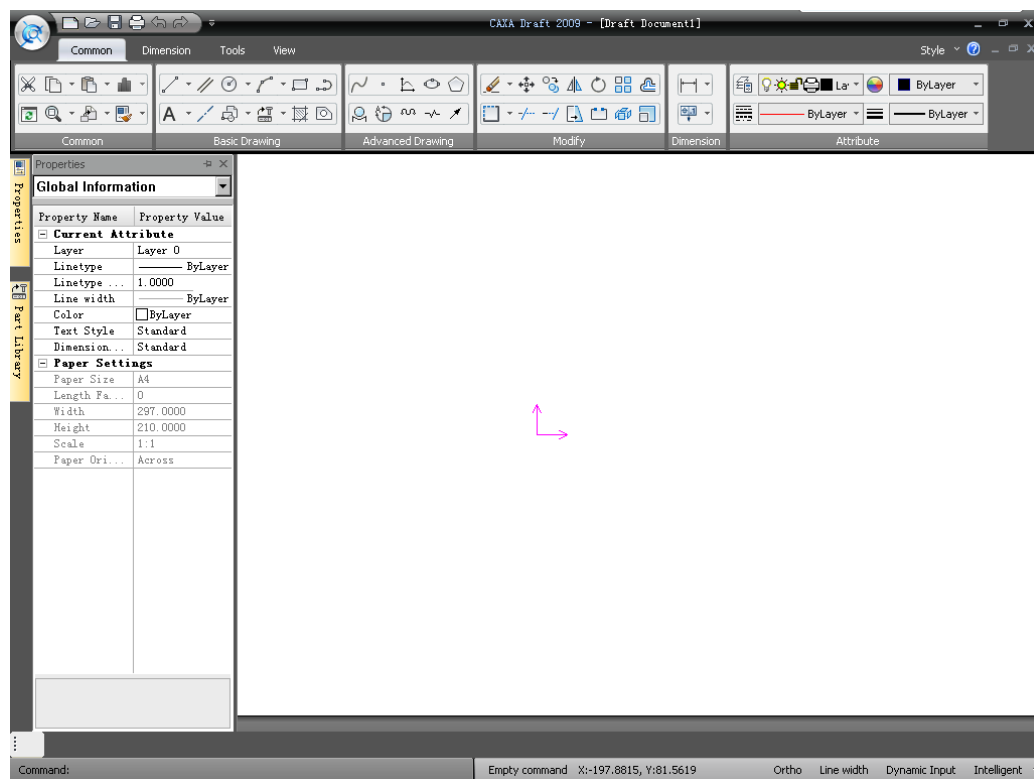


Fig.2.1 The CAXA Draft Fluent interface

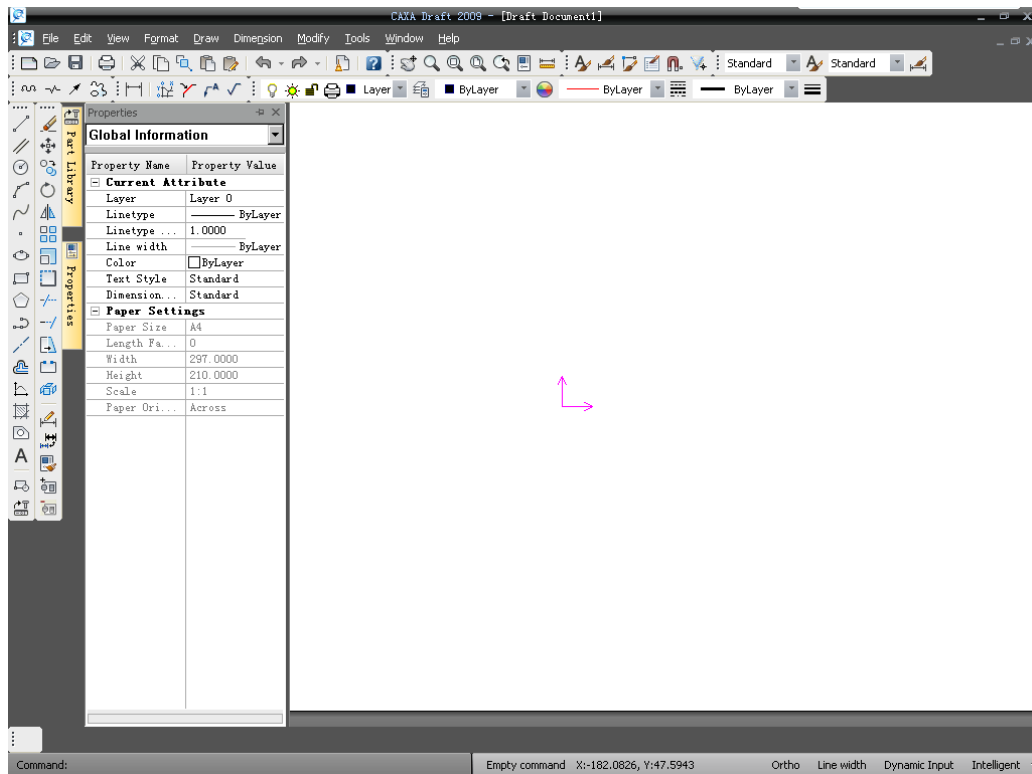


Fig.2.2 The CAXA Draft classic interface

The two interface styles can meet different user needs, and you can switch between the two as needed. Both the interfaces are explained below.

2.2.1 The Fluent Interface

The most important part of the fluent interface is the function area. The function area contains the command buttons, arranged in a compact and orderly manner to maximize the drawing area. You can also use the menu buttons in the 'quick start toolbar'.

Go to the "View" tab, "Interface Operations" panel, click "Switch Style" to switch from the fluent interface to the classic interface. Also, the F9 key can also be used as a toggle to switch between the fluent and classic interfaces.

2.2.1.1 Function area

The Function area contains various function tabs; such as Common, Dimension, Paper, Tools, View etc. All function tabs have command buttons grouped per their functions. These groups are called "option panels" or "panels". The following figure shows a typical view of a function area with many panels.

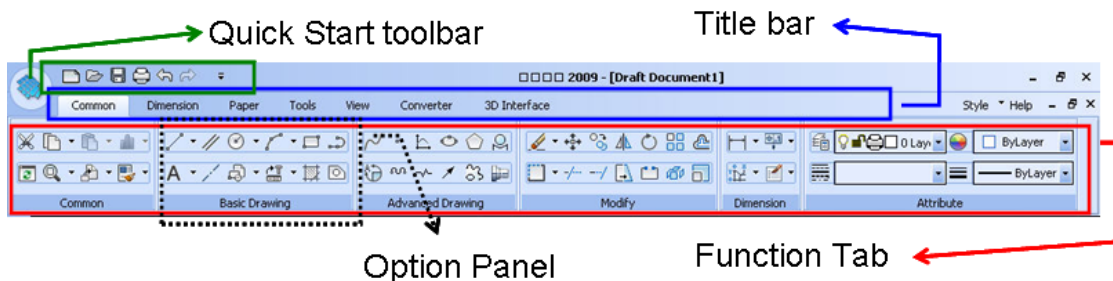


Fig.2.3 Function Area

The figure above shows the “Common” tab containing several panels “Common”, “Basic Drawing”, “Advanced Drawing”, “Modify” etc. Command buttons are there on each panel based on their functions.

The Following are few guidelines to use the function area:

- Every Function tab has a title. The tabs are arranged with their titles placed in a row above the panels. You can switch between tabs by just clicking on the tab title.
- You can hide or minimize the function tab as needed by double clicking on the title bar, double-click again to restore the tab.
- Right clicking on any of the interface elements will pop up a menu. You can select to open or close the element by selecting the corresponding option from the popup menu.
- Using the command buttons in the panels is the same as using the commands in the main menu.
- You can change the interface color by selecting the style in the ‘Style’ option menu in the upper-right corner of the function area.

2.2.1.2 Menu button

You can reach the classic main menu through ‘Menu Button’ available in the upper-left corner of the function area in the Fluent interface. As shown in below figure.

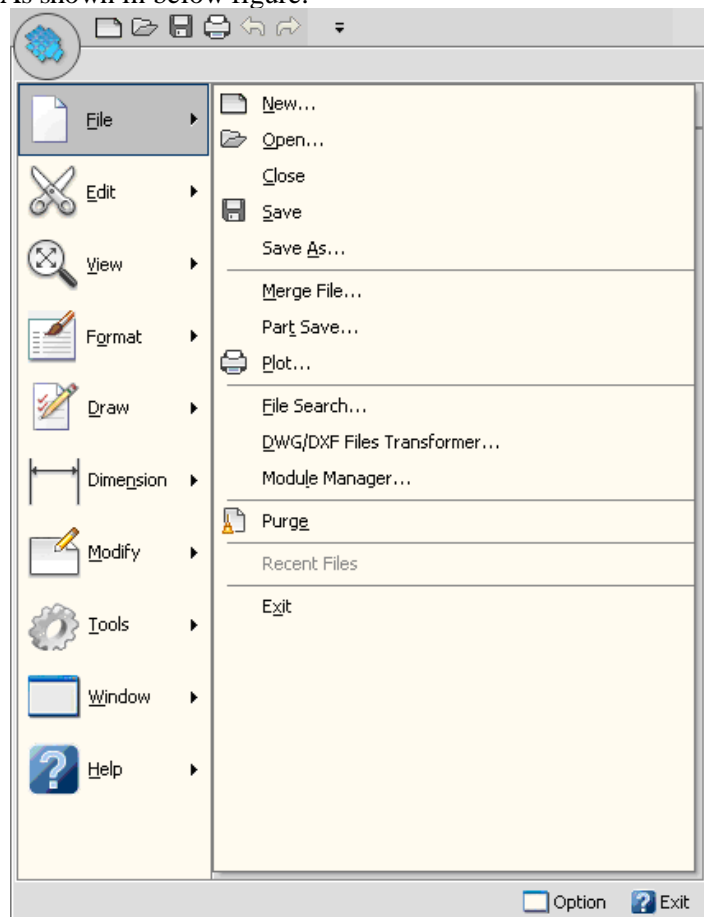


Fig.2.4 CAXA Main menu button

Usage of the menu button:

- Left click on the main menu icon on the left corner as shown in the above figure to access the main menu.
- The main menu contains a few submenus as “File”, “Edit”, “Tools”, etc., through which you can access any command. The main menu also lists the recently accessed files, from which you can easily choose one to

open.

- You can learn the function of a command from the tool tip text that appears when you hover the mouse over an icon. You have to left click a menu option to open it.

2.2.1.3 Quick start toolbar

The Quick start toolbar appears as shown below. It is available on the task bar just adjacent to the main menu icon. This menu contains commonly used command icons to enable easy access. This toolbar can also be customized as needed.



Fig 2.5 Quick Start Toolbar

Usage of the quick start toolbar:

- Left click on any icon on the toolbar to execute its corresponding operation.
- Right clicking on an icon, a menu as shown below will popup.

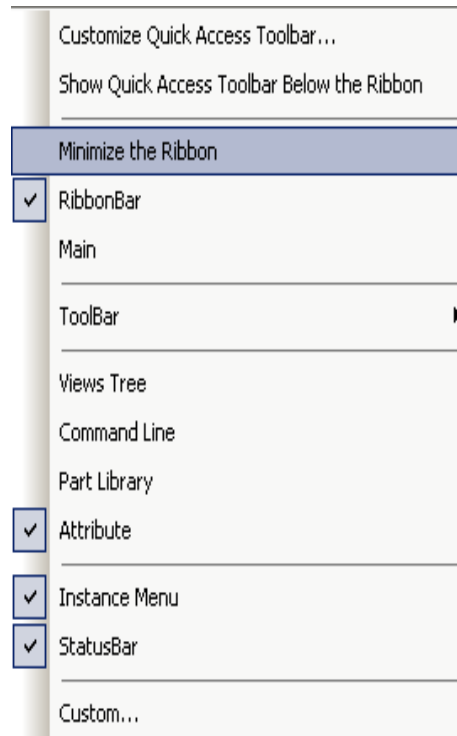



Fig.2.6 Quick start toolbar menu

- This popup menu helps you to define / customize the quick start toolbar. Using this popup menu you can select to remove a command from the quick start toolbar or can opt to add more commands to the quick start toolbar. In addition, you can open or close other interface elements in the popup menu, such as main menu, toolbar, status bar etc.
 - You can right click on a command in the function area and select 'add to quick start toolbar' to add the particular command to the quick start toolbar.
 - Click on the pull down button  on the right end of the quick start toolbar to define the quick start toolbar.

2.2.2 Classic interface

You can organize the main menu and toolbar in the CAXA Draft classic interface, as shown below.

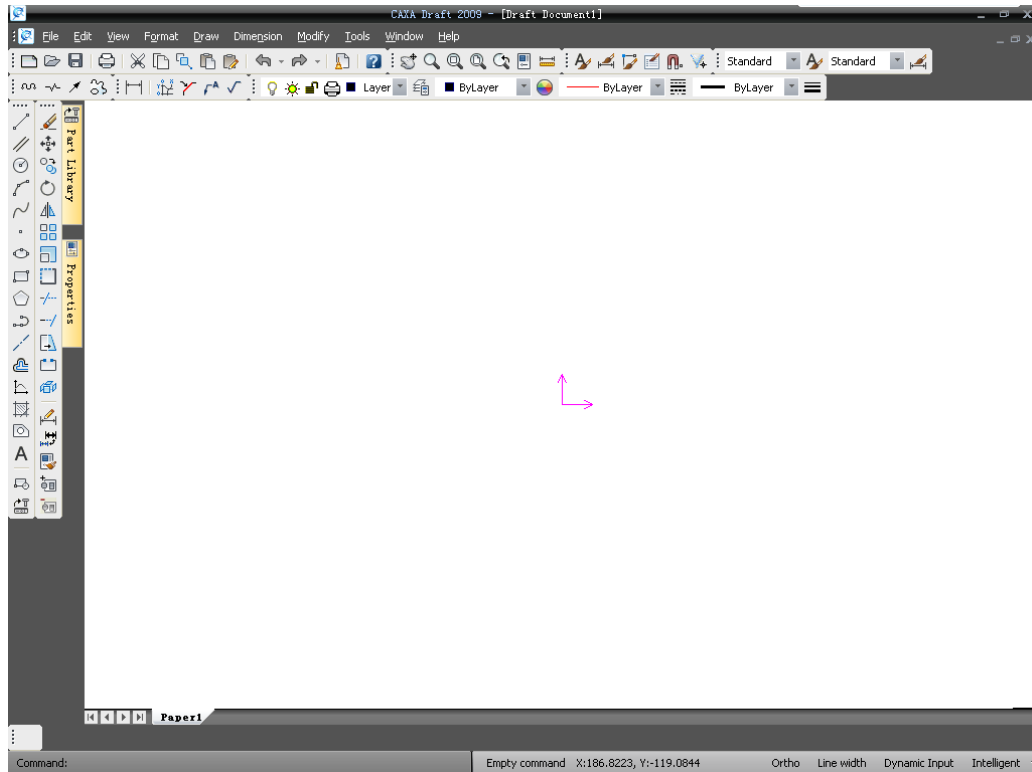


Fig 2.7 Classic Interface

The Classic interface has a menu bar at the top of the window. The menu bar contains the menus named “File”, “Edit”, “View”, “Format”, “Paper”, “Dimension”, “Modify”, “Tools”, “Window”, and “Help”. Left clicking on any of the menus opens a drop down list of functions / commands; you can click the required function from this menu to execute the command.

Next are the toolbars that contain icons to represent various commands. There are a number of toolbars with icons categorized based on the function. The toolbars are categorized as “Standard”, “Draw Tool”, “Modify Tool”, “Common Tool”, “Settings Tool”, “Dimension”, etc. Click an icon to execute the associated command. The various toolbars are as shown in the figure below.

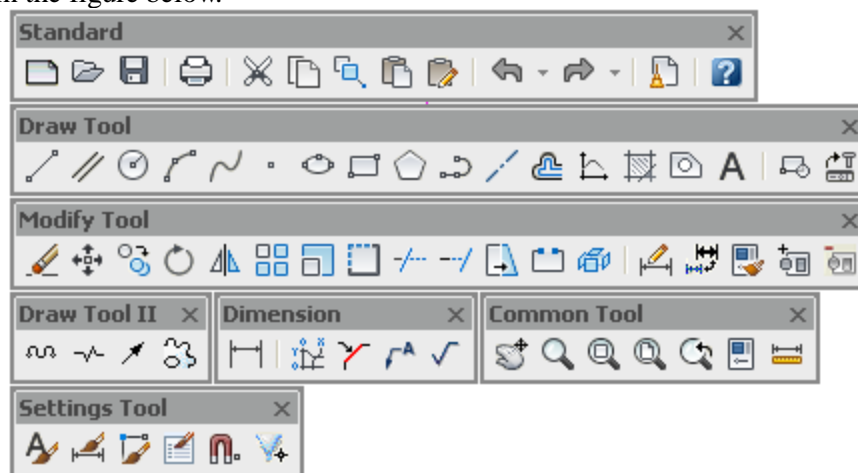


Fig 2.8 Toolbars

2.2.3 Immediate menu

CAXA Draft also has an immediate menu. The immediate menu provides command conditions / options in the

form of lists that can be selected as required. As soon as you initialize a command, it's corresponding immediate menu will appear below the drawing area.

Eg. 1, Let's take an example of the 'Line' command


Type 'Line' or click the line icon  in the drawing toolbar to draw a line. Instantly the Line command's immediate menu and corresponding operation instructions will popup as shown below.



Fig 2-9 Immediate menu

The above shown menu indicates that the line is to be drawn by defining two points, should be non-orthogonal and continuous, meanwhile, the first point's coordinate is displayed below the immediate menu. In case of absence of this coordinate information while entering the point, the tangency point or perpendicular point cannot be input. So input the first point as requested, and the second point (tangency point or Perpendicular point) instruction will be shown. A line is complete after defining the second point.

The main purpose for immediate menu is to select different functions of a particular command. To list a pull down menu in an immediate menu, click the respective pull down arrow or press combination key ALT + the corresponding number that will list it. When there are several options in the pull down menu, use combination key ALT + successive number key to select the desired option. E.g. For drawing an orthogonal line, click 3 to select non-orthogonal in immediate menu or use combination key ALT+3 to select it.

• Eg.2, Lets take example of the Coordinate dimension command

Execute the Coordinate dimension command and the immediate menu as shown below will appear instantly.

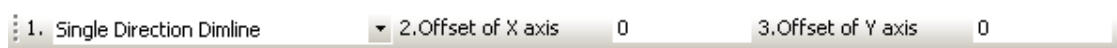


Fig 2.10 Immediate menu

In the immediate menu, click the first item, eg. **【1.origin dimension】** or press combination keys of **【Alt + numeric key】**, then select the option required.

Notice the calculator button on the right side of the numeral input frame, click that, the following dialog box (calculator) will pop up, which will help you to do calculations.



Fig. 2-11 Calculator

Input numeral and calculate, then press enter key, the result will be filled in to the numeral input frame automatically.

Under that circumstance, press the space key and another immediate menu called 'tool point' menu will popup, you can select a feature point as per drawing requirement to snap to, as shown in following figure.

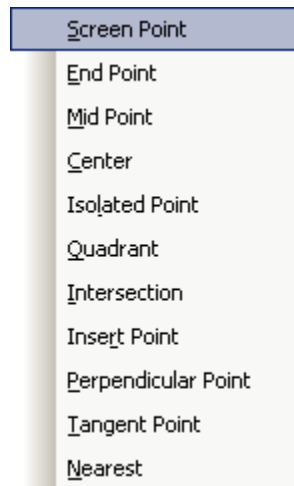


Fig. 2.12 Tool Point menu

2.2.4 Drawing area

The blank area in the center of the window is the drawing area. This blank area is where you create our drawings.

A 2D right angle coordinate system that is seen at the center of the drawing area is called the world coordinate origin (0.0000,0.0000).

With respect to the origin point the horizontal axis is the X axis, and the vertical axis is the Y axis. The right side of origin is positive X, and left side is negative X. Similarly above the origin is positive Y and below is negative Y.

Any point selected by mouse or keys in the drawing area are based on the current coordinate system.

2.2.5 Status bar

The status bar is present at the bottom of the window. It provides information about the current status of the window like screen view, operation status view, current tool point setting, selecting status view etc. The status bar is as shown below.

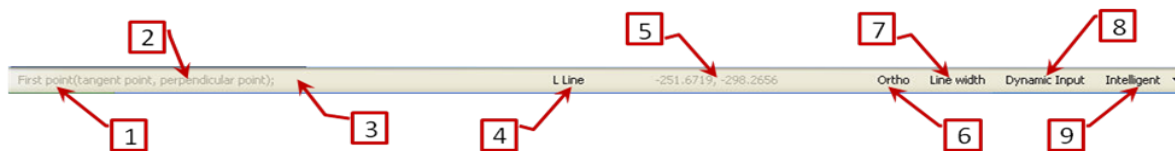


Fig. 2.13 Status bar

- (1) Operation information. The system prompts you as to what is to be done to execute the current command that is initialized or the remaining details to be input for a command.
- (2) Tool menu state. This is the supportive information to the operation information. This information gives hint about the current point or the action (Screen point / tangential / end point etc.) and the selection mode (adding or moving state)
- (3) Command and data input area. This is the area where you enter your command or input data via keyboard.
- (4) Command hint area. This is to hint the current command.
- (5) Coordinate view area for current point. The coordinate details of the current pointer position (X, Y) are displayed here. The coordinate value keeps changing instantly as you keep moving the mouse.
- (6) Switch Orthogonal. You can click here to toggle orthogonal or non-orthogonal status.
- (7) Switch Line width. By clicking here you can instruct CAXA to display as per line width or not.
- (8) Dynamic input tool switch. You can click this button to switch on/off dynamic input mode.
- (9) Point picking state setting area. You can set the point picking status in this area (free point, intellectual point, navigating point and grid point)

2.2.6 Tool option panel

Tool option panel is a special alternate tool in CAXA, which contains the library, attribute box, command line, and the tree.

Right click blank area on the tool option panel, a menu will popup. The popup menu will allow you to select the entities that you require to be displayed in it.

- Thus selected windows will appear hidden on the panel as shown in below figure. You can click on the required window, for it to appear as shown below.

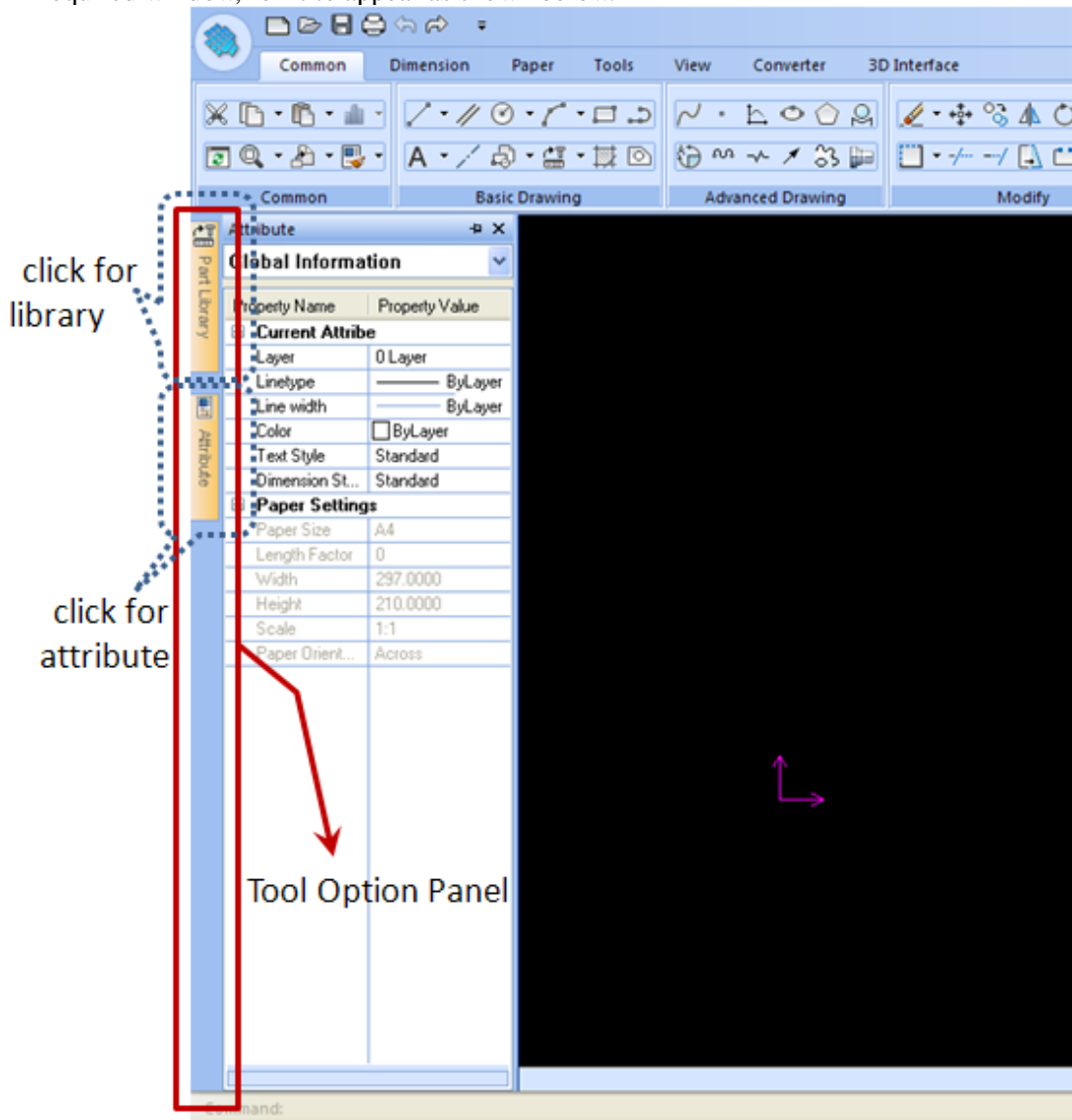
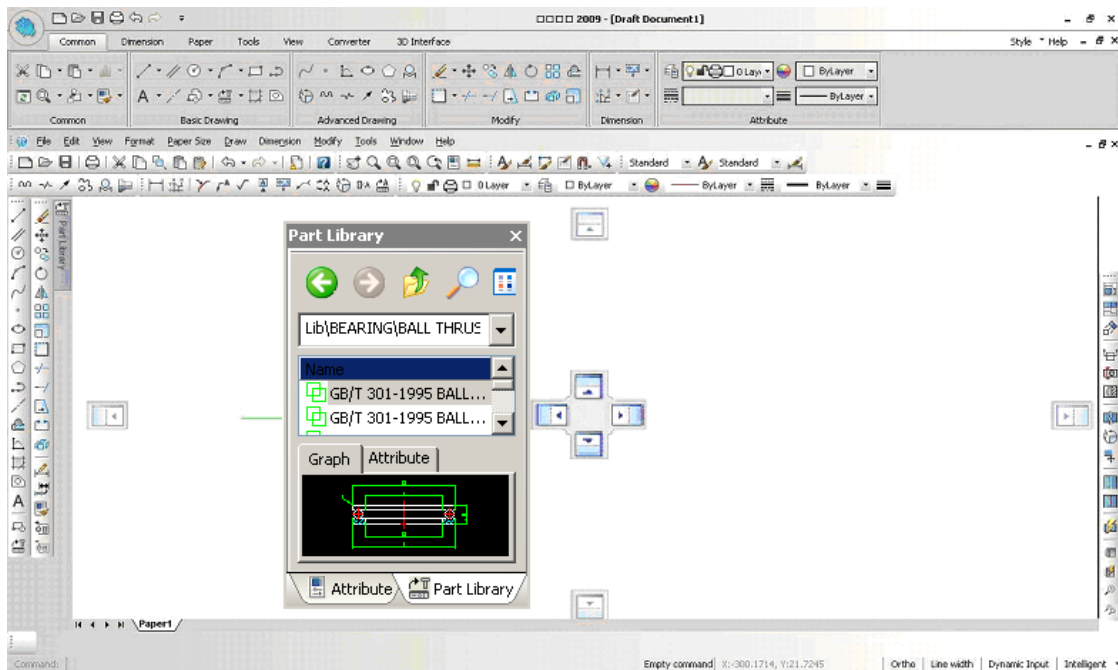



Fig 2.14 Tool option panel



- You can also move the window and place it where ever required by dragging the window title.
- Click the  button on the top-right side of the panel, to toggle auto-hide On and Off.

2.3 Basic operations

This section is to explain you how to execute a command, input values required, use right click menu, dynamic input option etc.

2.3.1 Execute command

In CAXA you can execute a command in many ways. You can initialize the command by typing it, clicking on the corresponding icon in the function tab, or by clicking on the command from the respective main menu. Every command performs its unique function my getting input from you. You can input the data for a command either using the mouse, or type in the data input area of the status bar or input the data in dynamic mode.

Any of the above methods can be adopted by you at all levels. Mouse selection might be more suitable for beginners or those who are comfortable with mouse operation. In this mode, you have to input data by clicking on the drawing area using the mouse, according to the prompt shown on the status bar and your requirements.

The other mode is input through Keyboard. It is directly typing the command or data using the keyboard. This method requires you to be familiar with each command and its corresponding function.

After execution of the command, when the operation hint says Command, right clicking or selecting enter will repeat the command again.

2.3.2 Input value of a Point

Point is the most basic drawing element, and entering the point value is the basis of all drawings.

In addition to mouse selection and keyboard input, CAXA Draft also provides several point picking modes, such as intellectual point picking and tool point picking.

Input coordinates points using keyboard

There are two types of coordinates systems, absolute coordinates and relative coordinates. Data input method for either system is completely different.

The input method for absolute coordinates is simple, one can input X and Y coordinates via the keyboard directly, with a comma between X coordinate and Y coordinate, e.g. 30,40.

In relative coordinates, the coordinates are related to the current point in the system, and not concerned with origin of coordinates. To input relative coordinates you have to include a '@' mark before the coordinate values. For eg. @60,84 will be taken as reference coordinate values, with 60 as X coordinate value, and 84 as Y coordinate value. In addition, relative coordinates can also be denoted by polar coordinates. For example, @60<84, will be taken as relative coordinate and the values in polar coordinate system. It represents that the distance is 60, and counterclockwise angle from the X axis is 84 degrees.

Reference point explained in more detail: A reference basis relating to coordinates set by the system automatically. It is usually the last operation point. Press the F4 key in the course of current command interaction to define the desired reference point.

Input point coordinates by mouse

You can drag the mouse pointer to the desired location; left clicking at the location will take the corresponding coordinates as the input point value. All input by mouse will be in absolute coordinates. While picking points using mouse you can easily pick points like End point, Tangency point, etc., without much difficulty.

Tool point picking

You can pick points of geometric features in the course of drawing, like center of a circle, tangency point, end point etc. using the Tool point snapping.

You can press the space bar when you need the tool point menu to popup. Following are listed the various options in the tool point menu

Screen point(S):	Any point on the screen
End point(E):	End point of curve
Midpoint(M):	Midpoint of curve
Center (C):	Center of Circle or circular arc
Isolated Point(I):	Existing point on the screen
Quadrant (Q):	Quadrant point on circle or circular arc
Intersection point(I):	Intersection point of two curves
Tangent point(T):	Tangent point
Perpendicular point(P):	Perpendicular point of curve
Nearest(N):	The point is the most nearest to cursor picking point

The default state of tool point is the screen point. When you select other points, the picking state of current tool point will be displayed at the right bottom corner of the status bar. But this picking is effective only for one pick, it will return to screen point when a pick is done.

You can use shortcut keys to select pick point options as shown in the above table.

When using tool point picking, the picking box can be pre-set using the command Tools, Snap Settings in the main menu. All other picking modes will be cancelled temporarily.

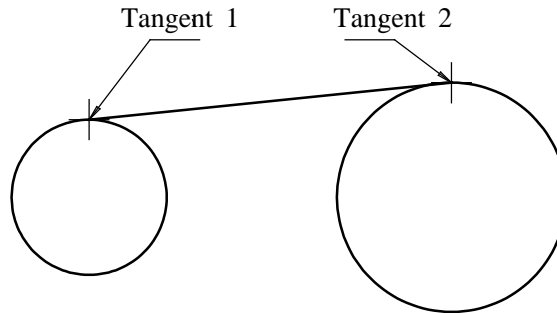


Fig. 2-15 Tangent Points option

The above Figure is an example of using line and tool point picking to draw a common tangent, proceed as follows:

- 1) Select or type the line command
- 2) For the first point press the space bar a menu will pop up, select T and the first circle then select the point on the circle.
- 3) For the second point press the space bar, select T from the popup menu that appears and then the second circle to pick the second point.

When the dynamic input tool is started, you can input point coordinates in the frame directly.

2.3.3 Select object

In any drawing there are lines, arcs, circles, blocks, etc., which are called the objects of the drawing. Every object in a drawing has its own command set. CAXA Draft has many objects like lines, circles, circular arcs, points, ellipses, blocks, section lines, dimensions, etc. To edit or delete an object, you need to select that object first. Hence it is important to learn how to select an object. Several selected objects are called selection collection.

Below are few guidelines to select an object.

- You can select an object or objects in a specified area by using the mouse to specify two diagonal points of a rectangle. The object or objects within the defined rectangle will remain selected.
- When you do not want an object to be selected (deselect) you can lock the layer where the object is located.
- You can select a desired object using 'Pick setting' also.
- You can either select the object and execute the 'Edit' command or vice versa as per system requirement for different edit commands.
- Press the Esc key to deselect all objects.

The selected object will be highlighted in different color and line style. As default the selected object will be shown in red colored dashed line, to distinguish a selected object from the others. Small handles will be shown on each object, which are called "grips", as shown in following figure.

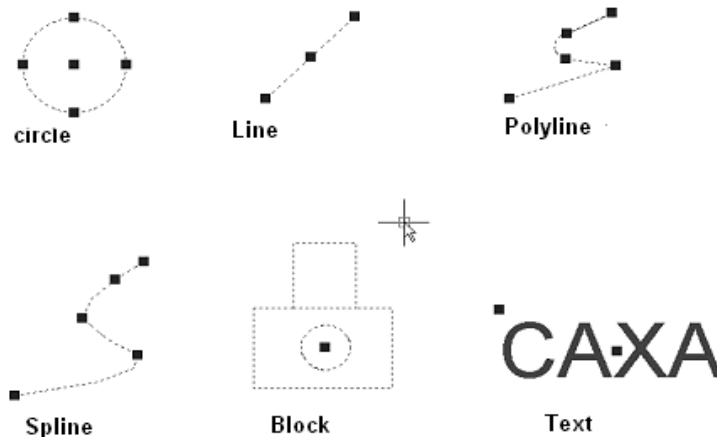


Fig. 2-16 Entity editing with grips

Select grip and drag, you can edit it at that time.

2.3.4 Right click menu

In CAXA Draft, you can access the right click menu to perform various operations based on the area where clicked. A right click menu is as shown in following figure.

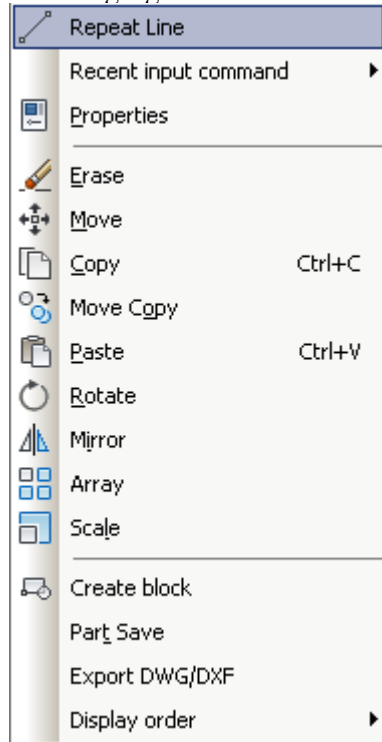


Fig. 2-17 Right click menu

There are different types of right-click menus that appear based on the area clicked or the operation status. A few different cases of right-click menus are as follows:

- In the drawing area, with an object selected or with nothing selected and no command executed, the right-click menu will be different.
- In the function area, quick start toolbar or other object.
- In the input area of text editor.
- In the blank area of status bar, on each kind of button.
- In some dialog boxes.

As shown in figure 2-17, the right-click menu generally includes:

- Repeat option for the previously used command.
- Display the recently input command list.
- Copy, paste, or other edit operations.
- Display order arrangement, block edit etc.

Right-click menu behavior can be set in Tools, Option, Selection as shown in figure 2-18.

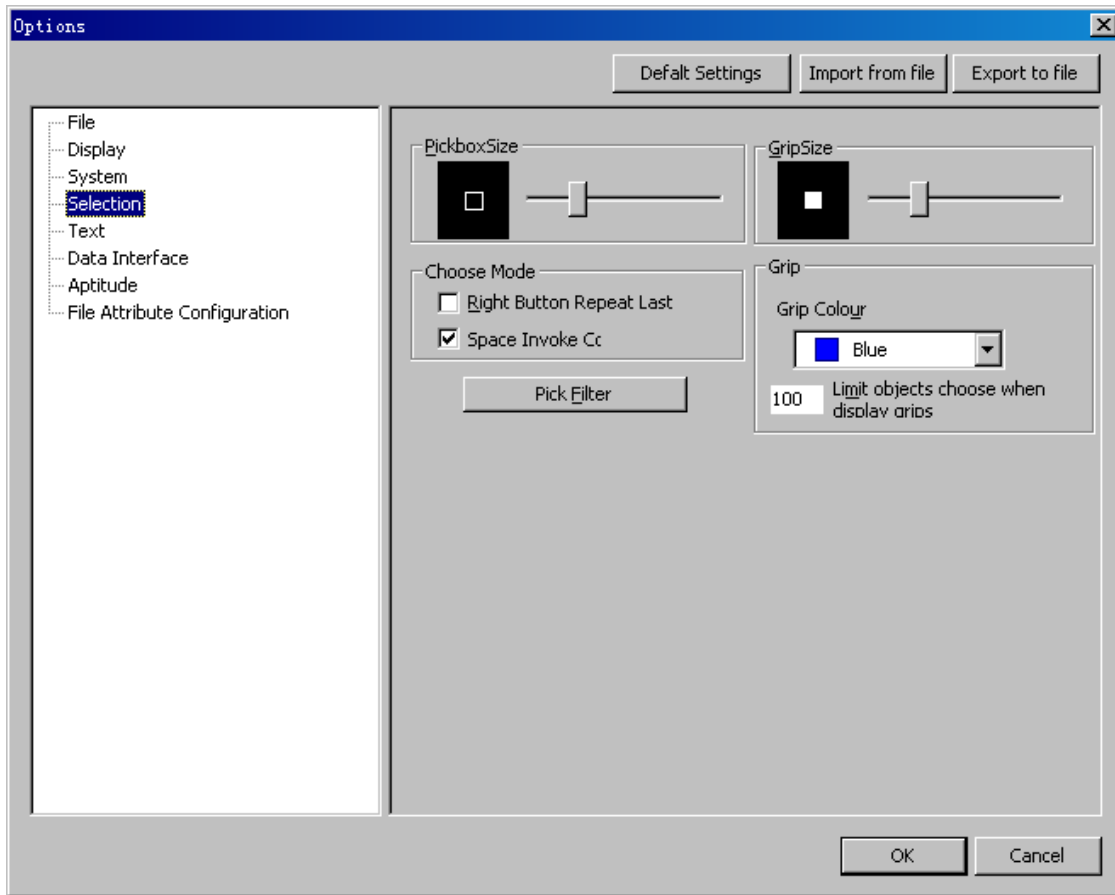


Fig. 2-18 Options /System Configuration

2.3.5 Dynamic input

In CAXA Draft, you can input command and point coordinates in the input area of status bar or the command line. Meanwhile, there is a special alternate method called 'Dynamic input'. The Dynamic input option is displayed near the mouse pointer itself where, you input command or input parameter.

Dynamic prompt

While starting Dynamic input, the command prompt will be displayed near the cursor. If required to specify the coordinates, it will prompt accordingly near the cursor. As shown in figure 2-19.



Fig. 2-19 Dynamic Input

Input coordinate

When required to specify a coordinate of a point, select the point using mouse or enter the values using keyboard in the dynamic input prompt. You need not go back to the command line to do this. While entering, press TAB key to switch to different input boxes.

Input dimension

In dynamic input method when required to enter the second point, tool prompt will be the distance and angle value. The values will be displayed in tool prompt which keeps changing with the mouse position. Press the TAB key, now you can modify the value in the tool prompt. Dimension input can be done in arc, circle, ellipse, line, and polyline as shown in following figure, specify distance, angle, radius or other parameter by Dynamic input.

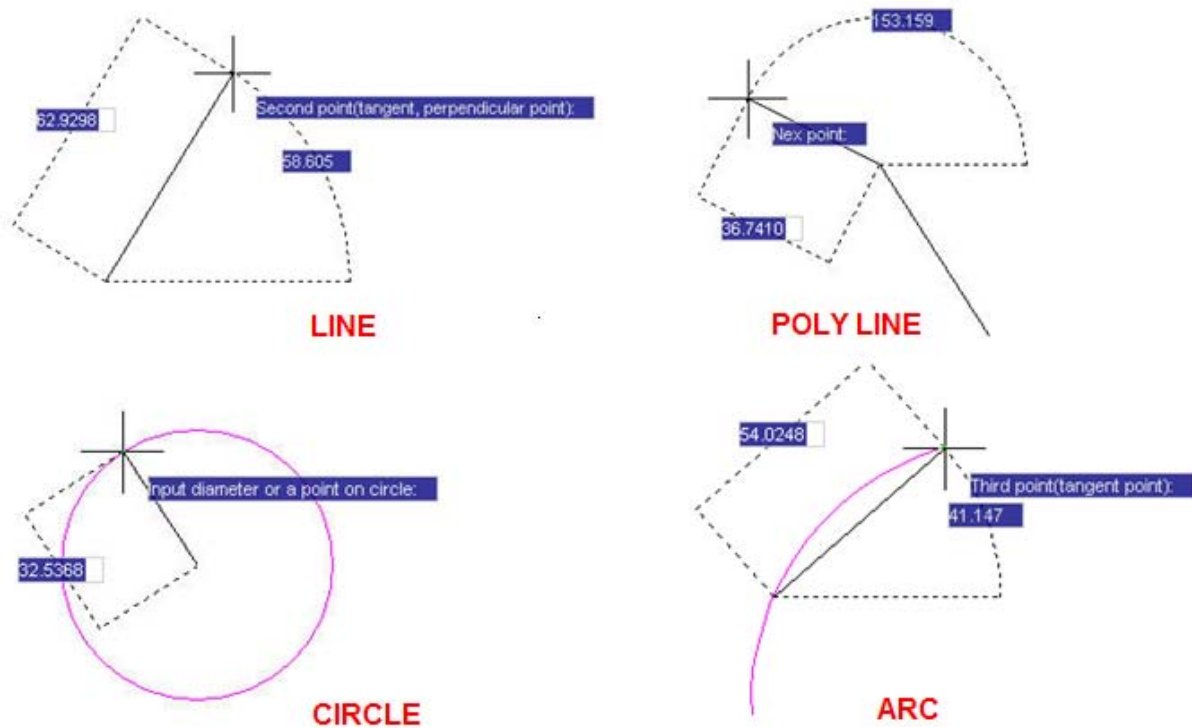


Fig. 2-20 Dynamic Input

Open dynamic input to edit grip point, the related information will be prompted as follows:

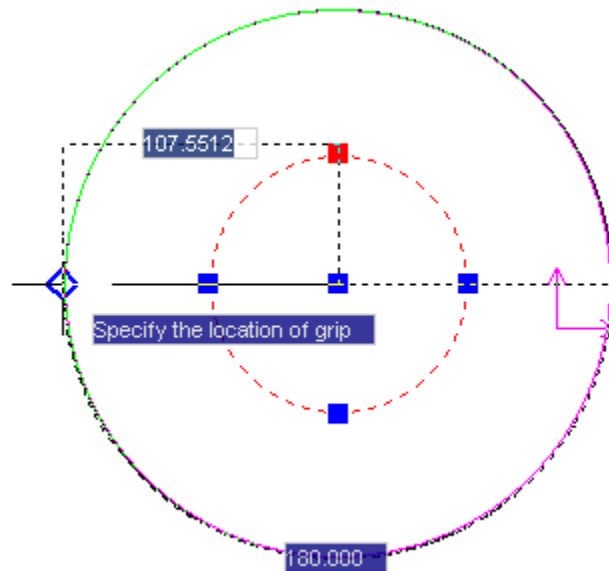


Fig. 2-21 Grip Point Dynamic edit

Following are guidelines for using Dynamic input

- Start 'Dynamic input', the information will be shown in the tool prompt near the cursor, meanwhile, the information will update dynamically along with the mouse pointer movement. When a command is executed, the tool prompt will provide input space.
- Enter the value required to be specified in the 'input field' and press TAB key, a locked icon will appear in the field. Then enter the value in the second 'input field'. If you press the enter key after

entering the first value, the second value input will be neglected and will be considered for direct input distance.

- The procedure for executing a command in case of using grip point or the command prompt is all the same. Except that you have to pay attention only around the cursor while using command prompt.
- In dynamic input mode, there are command prompts in the status bar or command line. You can close the command line to get a large drawing area.
- You can click 'Dynamic input' in the status bar to open or close it.
- Dynamic input can also use grip point edit.

2.3.6 Command line

You can use the 'Command line' tab in CAXA Draft to enter a command, and refer the previous operations records.

You can open the Command line window by right clicking the Ribbon bar and selecting "Command Line".

The 'Command line' tab shown in the figure below:



Fig. 2-21 Fig. Command line

2.3.7

Following are few guidelines for using command line:

- Type the complete command in the command line, or input abbreviated command. Eg. If you want to execute line command, input 'line' or 'l'.
- Press the Enter key or spacebar key to confirm command typed. If it is required to repeat the previous command, just press the enter key or space key directly.
- Drag the scroll bar on the right side of command line window or use mouse wheels to browse up and down.
- The position of command line, feature or other tabs and library on the screen, can be adjusted.

2.4 File operations

File operations like New, Open, Save, Parallel, Part Save, etc. will be discussed in this section.

You can perform file operations using the 'File' menu in the main menu or in the quick start toolbar. Below figure shows the File menu.

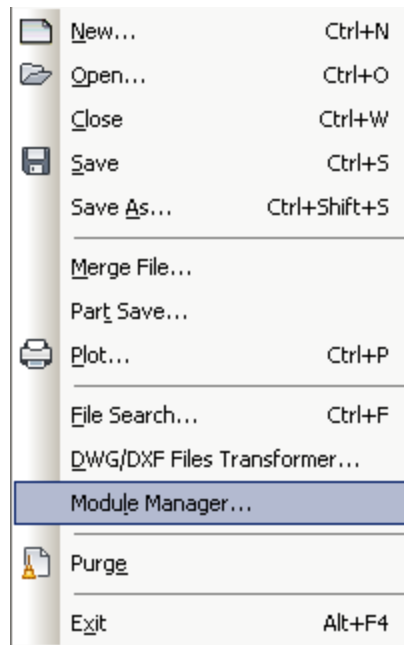


Fig. 2-22 File menu.

Click the required menu option to manage file.


2.4.1 New

【Command】 new

【Icon】 

【Definition】 Create a new drawing file, Select template to create a file.

【Process】

You can create a new file by selecting ‘new’ in the ‘File’ menu or clicking the  button on the quick start toolbar or press ‘CTRL + N’.

Once any of the above is done, the dialog shown below will popup:

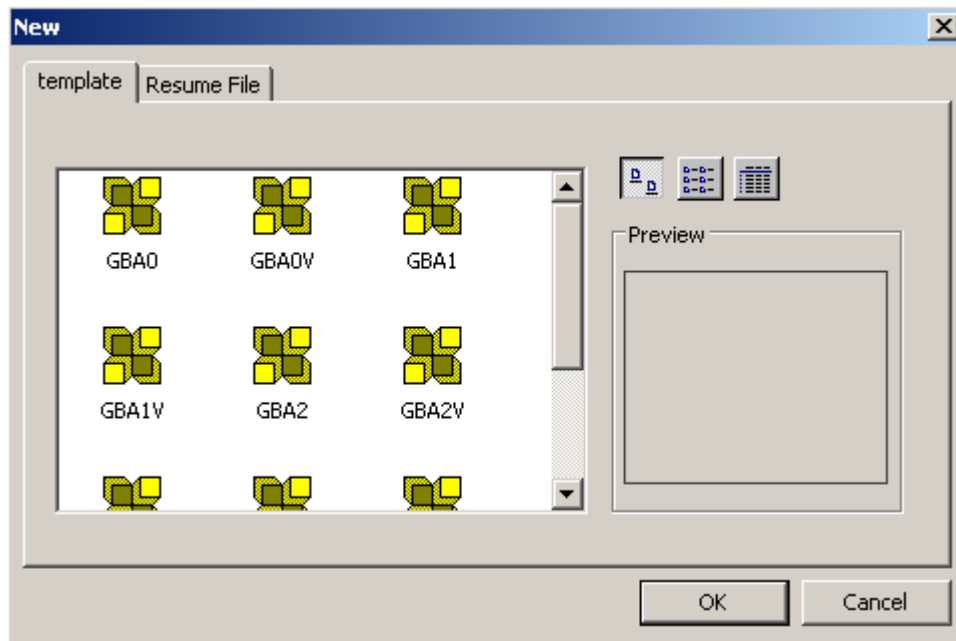


Fig. 2-23 New file dialog

There are several templates listed in the dialog box, which include A0-A4 paper, frame, title bar template and a blank template. You can select one of them to load it to start drawing on it as a blank paper.

After selecting the required template, you shall click the 'ok' button in the dialog box. A new drawing file will be loaded where you can start to draw, edit, or mark etc.


2.4.2 Open file

【Command】 open

【Icon】 

【Definition】 Open an existing drawing file

【Process】

You can open a file by selecting 'Open' in the 'file' menu or click the  button on the quick start toolbar or press 'CTRL + O'.

Once any of the above is done, the dialog shown below will popup:

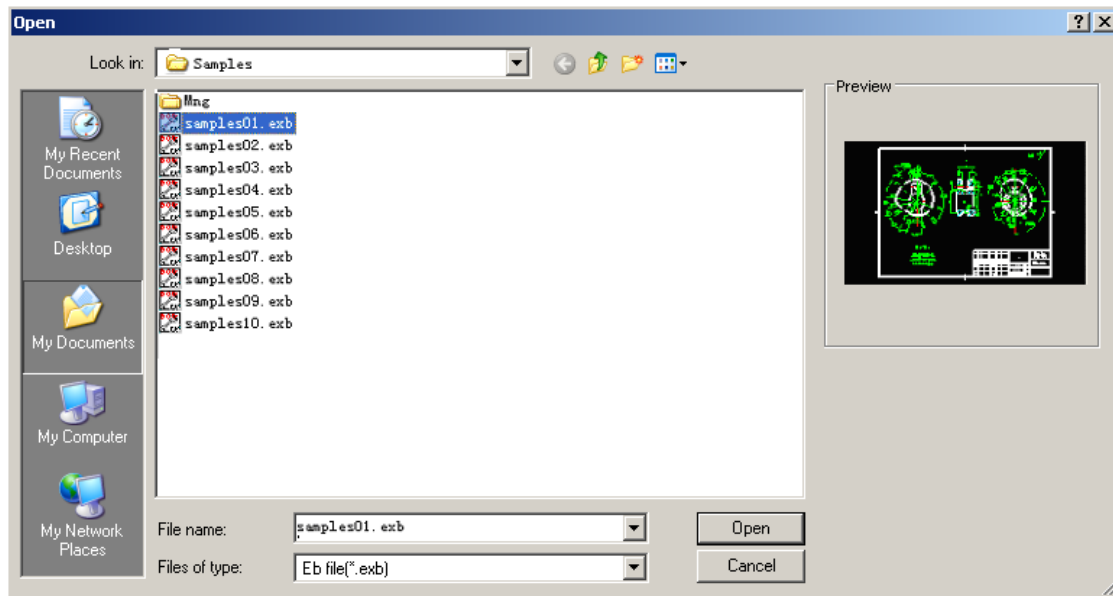


Fig. 2-24 Open file option

A list of all existing files that can be opened in CAXA Draft are available in this dialog box. You can select a file and click the 'Open' button to open it. Meanwhile you can also see preview of the file in the window on the right side. You can change the type of file to be opened by using the list box in the bottom of the dialog box as shown in below figure. This list contains the various types of file extensions sorted alphabetically. Select the file extension that is associated with the type of file you want to open.

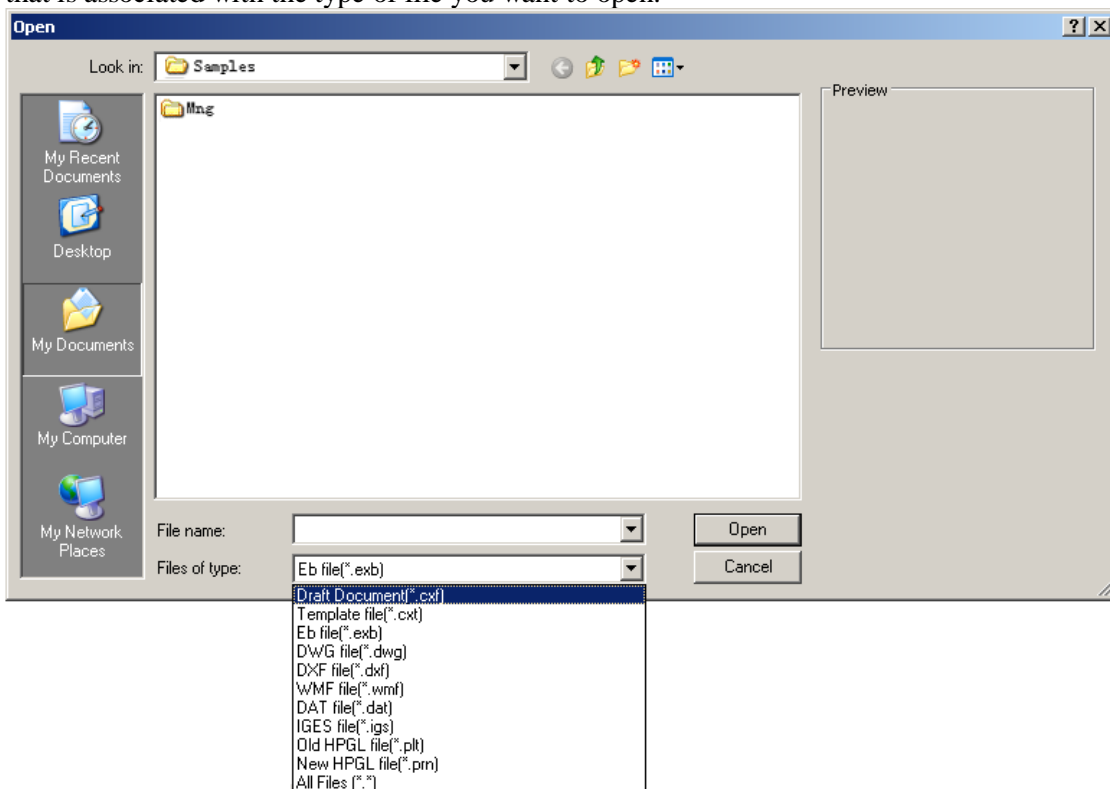


Fig. 2-25 Open file dialog with file type extensions

The various file types that can be opened in CAXA Draft are, EXB file, TPL file, DWG file, DXF file, WMF

file, DAT file, IGES file, PLT and PRN file in HPGL etc.

In the latter sections of Data interface, we will explain how to open DWG/DXF file in detail.

2.4.3 Save file


【Command】 save

【Icon】 

【Definition】 Save current drawing in the form of a file to the disk.

In the course of drawing, you should save the file frequently, in order to avoid data loss. If the file is not saved, the Save as dialog box will pop up when saving. But if it is saved already, or open one already-saved file, you should save it after being edited, then the modified result will be saved directly without any prompt.

【Process】

You can save a file by selecting 'SAVE' in the 'file' menu or click the  button on the quick start toolbar or press 'CTRL + S'.

Once any of the above is done, the dialog shown below will popup:

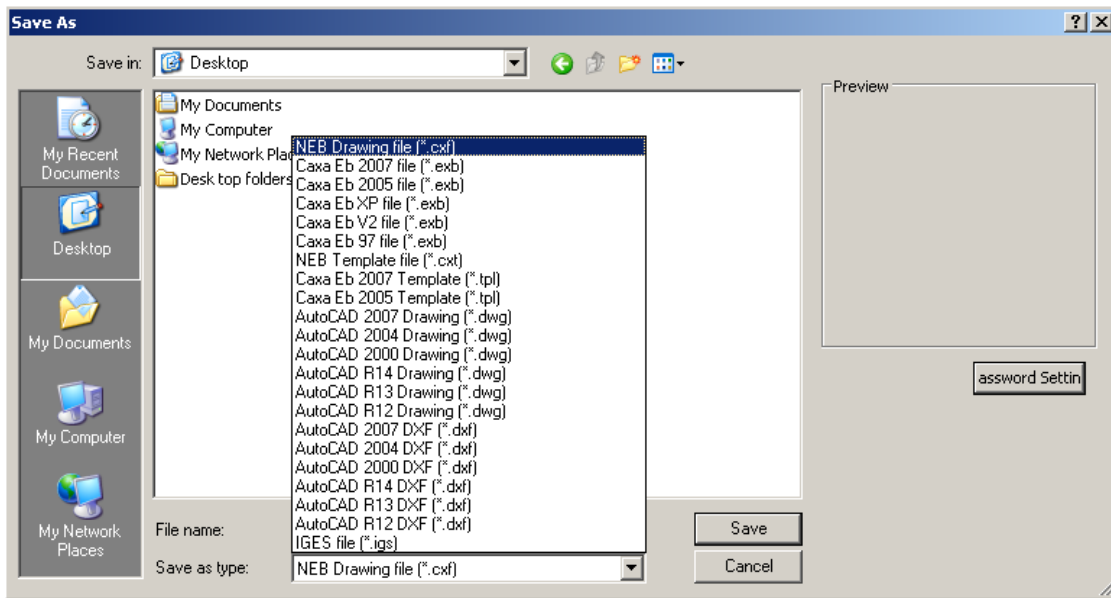


Fig. 2-26 Save file

Following are guidelines for saving a file:

- Select the folder location to save, enter the file name in the dialog box and click 'Save'.
- If the file already exists, the system will prompt 'Do you want to overwrite the existing file?' Select 'yes' or 'No' as required.
- To set a password for a saved file, click the 'Password' button and input the password twice. A password protected file will prompt the user for the password when trying to open it.
- You can choose to save a file to a different file format. At the bottom of the save dialog box there is a 'Save as type' list box which displays the various file formats supported by CAXA Draft. You can select the required file format from the list before clicking 'save'.
- You can also save files in formats of previous versions of CAXA Draft such as, CAXA Draft 2007, CAXA Draft 2005, CAXA Draft XP, CAXA Draft V2 etc.
- CAXA Draft 2009 can also save files in IGES, HPGL and BMP formats.
- To save a copy of an existing file, use the 'Save As' command.


2.4.4 Merge file

【Command】 merge

【Icon】 

【Definition】 You can merge a file to an existing file using this function.

【Process】

You can merge a file by selecting 'merge' in the 'File' menu or click the  button on the quick start toolbar. Once any of the above is done, the dialog shown below will popup:

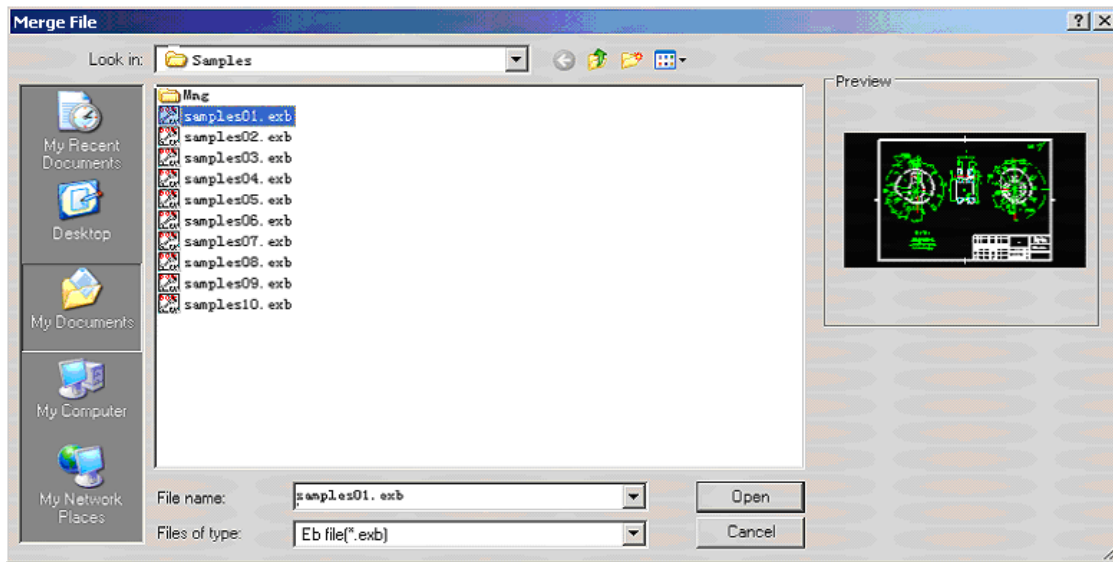


Fig. 2-27 Merge File Dialog

Guidelines to using the merge operation:

Select the file to be merged, and click Open, the dialog shown above will pop up.

If the selected file contains multiple drawings they are listed in the dialog box as shown above. You need to select the drawing which is to be merged from the list. Preview of the selected drawing will be displayed in the space at the right side.

When you select 'Merge' in the option, either of the two actions below will take place.

- Merge into current drawing: Merge the selected drawing as a part into the current drawing. In the immediate menu that appears, there are few options as 'Point' or 'Specify area', 'set zoom scale', 'keep original', 'paste as block' from which one is to be selected as per requirement. When selecting 'merge into current drawing', you can select only one drawing.

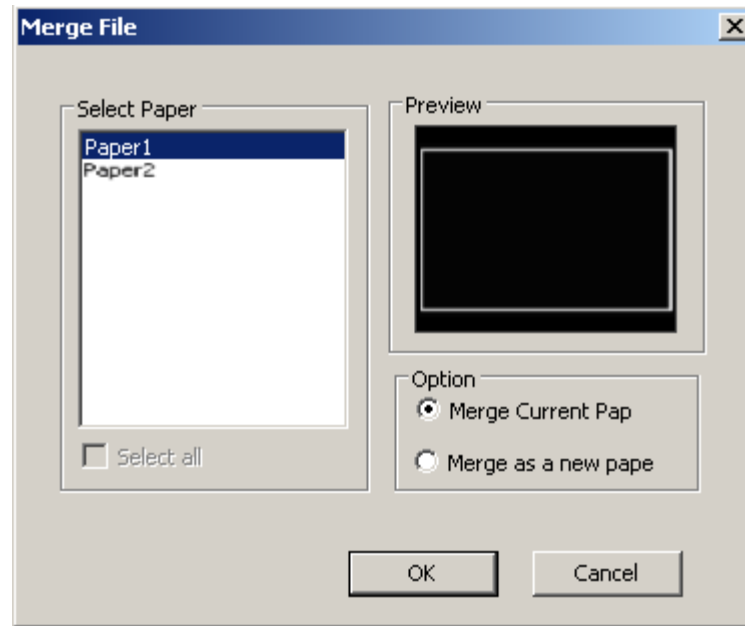


Fig. 2-28 Merge File Option

- Merge as a new drawing: Merge the selected drawing into the current file. Select one or more drawings. If the drawing name to be merged is the same as that of the current drawing, you will be prompted 'Please modify drawing name', as shown in following figure.

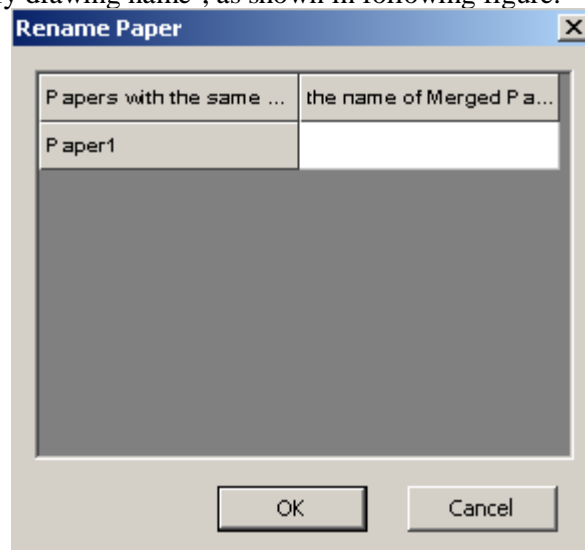


Fig. 2-29 Fig. Rename Paper

2.4.5 Part save

【Command】 Part save

【Definition】 Save part of figure as a file.

【Process】

You can click 'Part save' in the file menu, or click 'Part save' option in the right click menu, or execute Part save directly.

Select the object to be saved, then click 'Part save', or execute command 'Part save', then select the object and right click to confirm selection. Once the base point is specified, the below shown dialog box will popup.

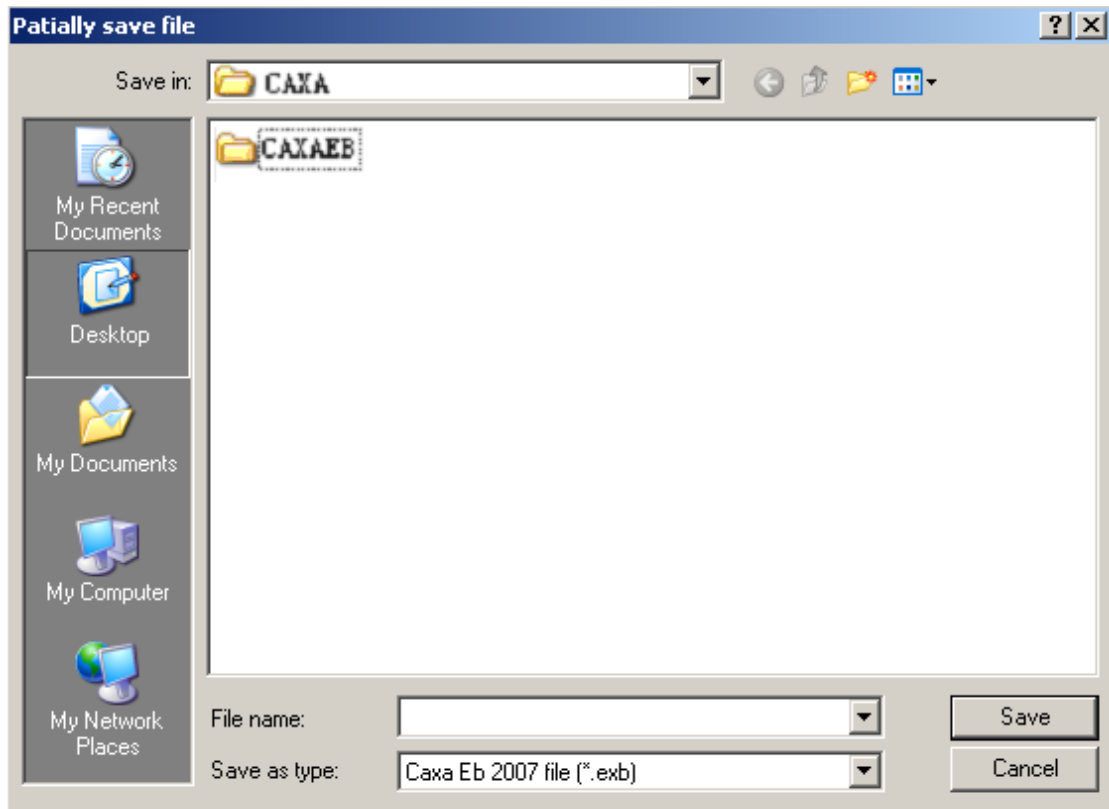


Fig. 2-30 Part Save option

The further operations are the same as that in section 2.4.3.

2.4.6 Multiple document operation

CAXA draft can handle multiple, drawing files simultaneously and also setup multiple drawings in a single file. Meanwhile you can switch between several drawings in a file, following this method:

(1) Open multiple documents simultaneously

Multiple documents can be opened simultaneously; each document can be used and saved individually. To switch between documents, you can press CTRL+TAB. In the Classic interface, you can click the main menu Window to switch, as shown in the below figure.

- Arrangement mode for multiple documents can be selected, such as layer, horizontal tile, vertical tile, or you can click the file name directly to switch as current window.

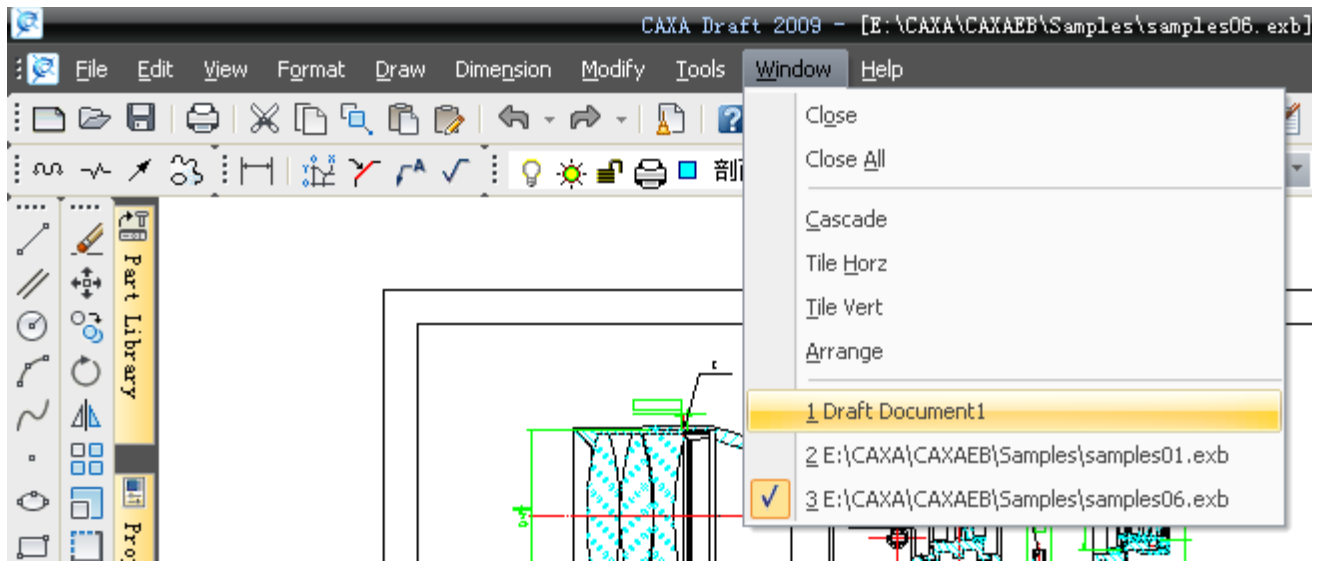


Fig. 2-31 switch between documents in classic interface

- In the Fluent interface, you can click View to use corresponding function in Window panel, as shown in following figure.

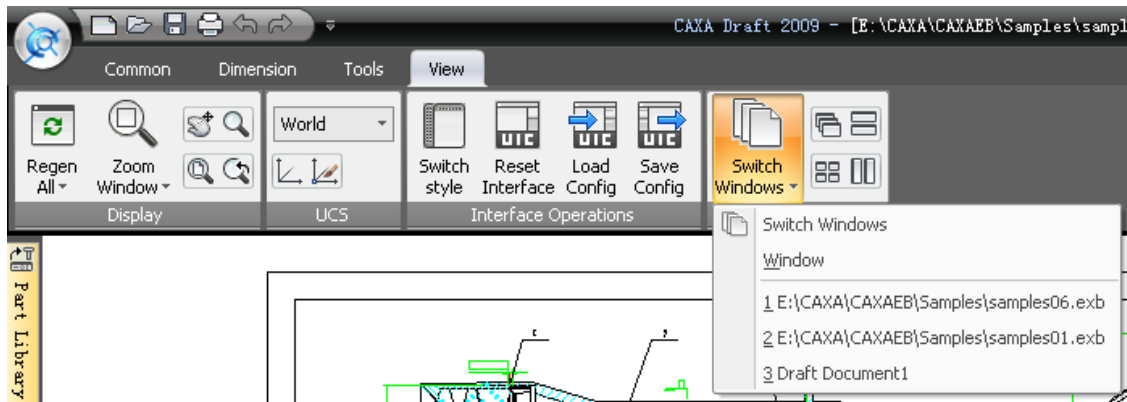


Fig. 2-32 switch between documents in fluent interface

Click either of the buttons, layer, horizontal tile, or vertical tile, to select Window arrangement mode. You can select the file to be switched to in the “Switch Windows” pull down menu.

(2) Work on multiple drawing in one file

You can work with multiple drawings simultaneously in one file in CAXA Draft, as shown in below figure.

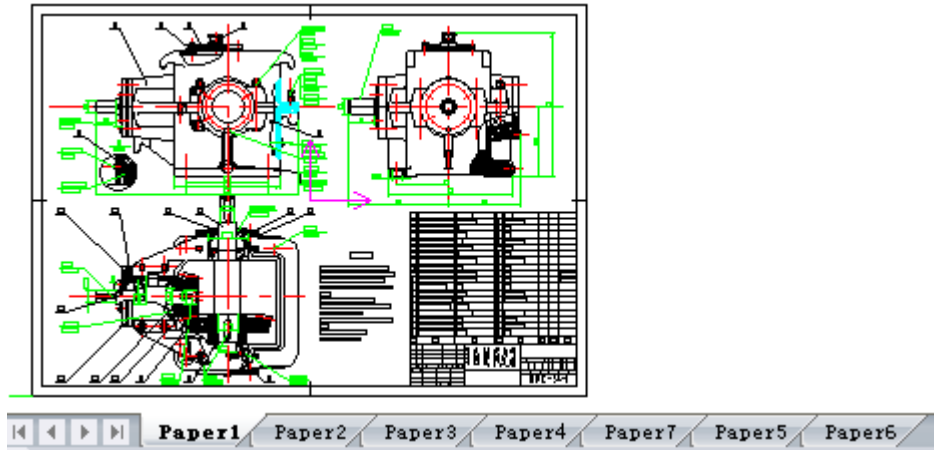


Fig. 2-33. Drawing Sheets

- Click the drawing sheet tab at the bottom of drawing area to switch between different drawings.
- When you right click on one of the sheet tabs, you get options to, insert a sheet, delete a sheet, rename a sheet and save it as another file.

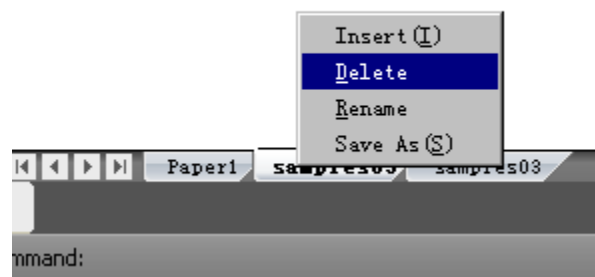



Fig. 2-34 Fig. sheet options

2.5 Control view

2.5.1 Summary

While drawing and editing, Pan or Zoom are frequently used, CAXA Draft provides such view commands to control the display. The main function of a view command is to change the subjective view of a drawing.

All view control commands can be executed from the main view menu, or the  button in the Common function area, or the display panel in function area. All view commands are listed and discussed below

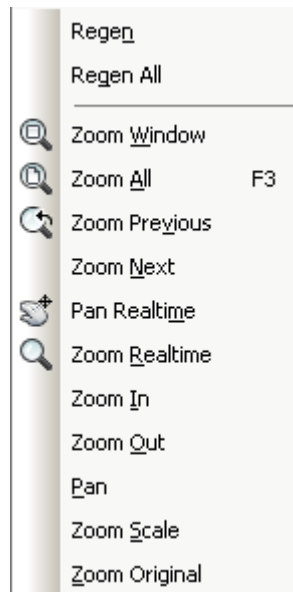


Fig. 3-35 Fig. View Control commands

Following are few view commands in CAXA draft.

2.5.2 Regen

【Command】 regen

【Definition】 Regenerates a distorted drawing.

Circles and arcs are combination of line segments, when they are enlarged to a certain scale, display distortion may occur. The distorted display can be regenerated in the current window display, as shown in the figure below.

Using the Regen command, you can regenerate the distorted drawing that is displayed.

【Process】

The Regen command can be executed in various ways. You can go to the main 'view' menu and click Regen, or Regen can be clicked from the common tab, it can be selected from the view option tab or regen can be entered directly at the command prompt.

Following figure shows a comparison before and after refreshing:

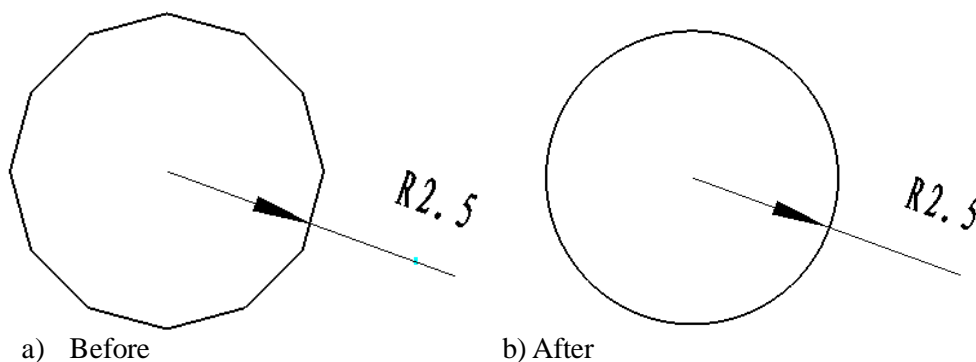


Fig. 3-36 Regen results

2.5.3 Regen all

【Command】 regenall

【Icon】

【Definition】 Regenerates all distorted objects.

【Process】

The Regen all command can be executed in various ways. You can go to the main 'view' menu and click

‘Regen all’, it can be clicked from the common tab, it can be selected from the view option tab.

Using the ‘Regen all’ command, you can regenerate all the distorted objects that are displayed.


2.5.4 Zoom Window

【Command】 zoom

【Icon】 

【Definition】 You can specify two diagonal points of a rectangle by dragging the mouse pointer. The figures within the rectangle will be zoomed to the window size.

【Process】

The zoom command can be executed in various ways. You can go to the main ‘view’ menu and click zoom, or zoom  can be clicked from the common tab, it can be selected from the view option tab or zoom can be entered directly at the command prompt.

To execute the zoom window command, click the first diagonal point of the rectangle at a desired position, the cursor will disappear immediately, continue to drag the mouse pointer towards selecting the other diagonal point. You can see a rectangular box while you are dragging. The area defined within the rectangle will be scaled to fit the window. In this case, no scale factor needs to be given; objects in the selected area will be displayed as large as possible on the screen again.

The figure below is an example of Zoom Window in operation:


2.5.5 Zoom All

【Command】 zoomall

【Icon】 

【Definition】 Rescales the view to include all objects in the window.

【Process】

The zoom all command can be executed in various methods. You can go to the main ‘view’ menu and click zoom all, or zoom all  can be clicked from the common tab, or zoom all can be selected from the view option tab, or zoomall can be entered directly at the command prompt

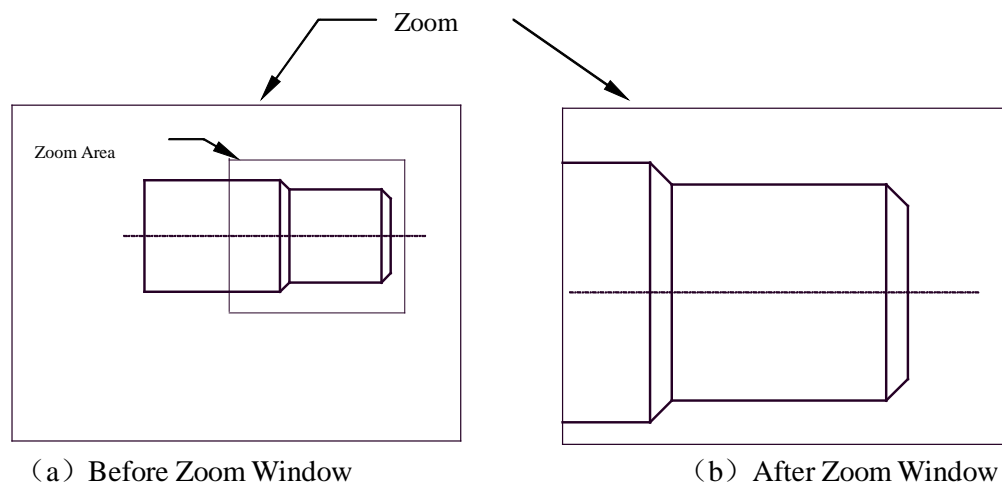


Fig. 3-36 View Window


2.5.6 Zoom Previous

【Command】 prev

【Icon】 

【Definition】 Cancel current view and return to previous view status.

【Process】

The previous view command can be executed in various ways. You can go to the main ‘view’ menu and click previous view, or it can be  clicked from the common tab, or it can be selected from the view option tab, or prev can be entered directly at the command prompt.

Execute the view previous command and the window will show the previous view status.

2.5.7 Next

【Command】 next

【Definition】 Return to a next view that is compatible with Previous view.

【Process】

You can execute the Next command in various ways, click next in the view menu, or click the Next button in the Common toolbar, or click next button under the View tab, or next can be entered directly at the command prompt.



2.5.8 Pan Realtime


【Command】 dyntrans

【Icon】 

【Definition】 Drag mouse to pan the drawing.

【Process】

You can execute the Pan Realtime command in various ways; click Pan Realtime in the view menu, or click the  button in the Common toolbar, or click the  button under the View Option tab, or Dyntrans can be entered directly at the command prompt.

Execute command Dyntrans, the cursor icon will change to , move the mouse pointer with the left button down and right click or press ‘ESC’ key to finish the operation.

In addition to this, you can access Pan Realtime by holding the scroll button down while moving the mouse.



2.5.9 Zoom Realtime


【Command】 dynscale

【Icon】 

【Definition】 Drag mouse to zoom-in or zoom out the graph.

【Process】

You can execute the dynscale command in various ways; click dynscale in the view menu, or click the  button in the Common toolbar, or click the  button under the View Option tab, or dynscale can be entered directly at the command prompt.

Execute the Dynscale command, the mouse pointer icon will change to , move the mouse up or down with the left button down. Moving the mouse pointer upward will enlarge the view and downward will reduce the view in the drawing window. Right click the mouse or press the ‘ESC’ key when the operation is done.

In addition, you can roll the mouse wheel to zoom in or zoom out.



2.5.10 Zoom in


【Command】 zoomin

【Icon】 

【Definition】 Enlarge current / required view as per fixed scale.

【Process】

You can execute the Zoom in command in various ways; click zoomin in the view menu, or click the  button in the Common toolbar, or click the  button under the View Option tab, or zoomin can be entered directly at the command prompt.

When the zoomin command is executed, the mouse pointer icon will changed to , left click the mouse to enlarge and when done right click mouse or press 'esc' key.

In addition, you can press Page up key to enlarge view.



2.5.11 Zoom out


【Command】 zoomout

【Icon】 

【Definition】 Zoom out the view as per fixed scale.

【Step】

You can execute the zoomout command in various ways; click zoomout in the view menu, or click the  button in the Common toolbar, or click the  button under the View Option tab, or zoomout can be entered directly at the command prompt.

When the zoomout command is executed, the mouse pointer icon will changed to , left click the mouse to enlarge and when done right click mouse or press 'esc' key.

In addition, you can press Page down key to enlarge view.



2.5.12 Pan

【Command】 pan

【Icon】 

【Definition】 Specify a center point for display, that point will be center of screen when panning drawing.

【Process】

You can execute the pan command in various ways; click pan in the view menu, or click the  button in the Common toolbar, or click the  button under the View Option tab, or pan can be entered directly at the command prompt.

Execute the pan command; specify a point as center point of the view, and move the mouse pointer with the left button down. The view will be panned. When done right click mouse or press 'esc' key. In addition, you can use the direction keys (up, down, left, right) to move screen center.

2.5.13 View scale



【Command】 vscale

【Icon】 

【Definition】 Input scale modulus for zoom in or zoom out current view.

It will zoom out or zoom in as per fixed scale.

【Process】

You can execute the vscale command various ways; click vscale in the view menu, or click the  button in the Common toolbar, or click the  button under the View Option tab, or vscale can be entered directly at the command prompt.

Execute the Vscale command, Input a scale value (0~1000) through keyboard as per prompt, then press Enter key. The view will be zoomed and displayed again as per input scale factor.

2.5.14 Home

【Command】 home

【Definition】 Resume drawing to original one when opened.

In the course of creating the drawing, you may change the views display as per different requirement, then use the home command to return to original standard drawing status.

【Process】

You can execute the home command in various ways; click home in the view menu, or click button home in the Common toolbar, or click button home under the View Option tab, or home can be entered directly at the command prompt.

Once command Home is clicked, the view will be displayed as per standard drawing.

In addition, you can press Home key via keyboard to execute this command.

Chapter 3 Drawing

3.1 Summary

Drawing is the foundation that CAD software forms, CAXA Draft replace manual drawing with advanced technology and direct operation. The drawing efficiency will be highly raised.

It provides multiple function mapping way for every complicated engineer drawing. Drawing mainly includes: basic shape, advanced shape, block, picture etc.

3.2 Basic shape

3.2.1 Introduction

Base shape includes straight line, parallel line, circle, arc, center line, rectangle, polyline, Equidistant Line, section line, filling.

User can execute each function in the form of corresponding keyboard command or short-cut key, click relative button in the main menu of Drawing, click relative button in the basic drawing panel, or click corresponding button in the toolbar.

3.2.2 Line



【Command】 line

【Icon】 

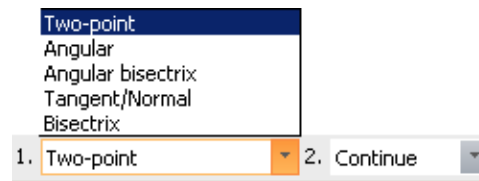
【Definition】 Create straight line.

Line is the foundation that drawing forms, and selecting point is the key for correct and shortcut line drawing, this software provides **tool point**, **intellectual point**, **navigating point** and **grid point** for point selection. The absolute coordinates is often used for point input, relative coordinates and polar coordinates are used if necessary (refer to Basic operation for point input).

【Step】

User can implement command Line in the forms of : click Line button in the main menu of Drawing, click the button of  in the commonly-used Basic drawing panel from the option card , click the button of  in the toolbar of Drawing, implement command of line.

Implement the command Line, the following immediate menu will pop up.



Pic 3-1 implement command of line

It also provides 5 methods to draw line: two point line , angle line, offset line, tangent line/normal line in CAXA Draft, user can select different mode and parameter from the immediate menu to create line individually.

3.2.2.1 Draw two point line

【Command】 lpp

【Icon】

【Definition】 Create two point line

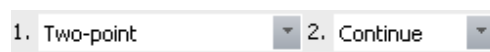
Draw a line or continuous line segment as per the given option. When it is non-orthogonal, the first point and the second point can be: tangent point, perpendicular point, or other point, tangent line, perpendicular line, perpendicular tangent line or random two point line. When it is orthogonal, the line to be produced is parallel or perpendicular to current coordinate.

Note: Press F7 key to switch to orthogonal, or click button orthogonal in the status bar down-right .

【Step】

User can implement command Lpp in the forms of : click Lpp button in the sub-menu of Line, execute Line command, then select two point line in the immediate menu, implement command of Lpp.

Click command line, the immediate menu will pop up as follows, in which select two point line:



Pic3-2 two point line

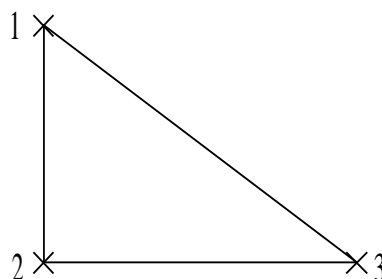
Click **2:in immediate menu**, **continuous** will be **single**, every line will be joined when it is at **continuous** option, the first end point will be the start point for next line. **Single** means the line to be drawn is separately each time.

According to the condition in immediate menu, select two points to draw a line. User can input coordinates or distance for two points to draw accurate line ,or input realtime coordinate and angle dynamically. This command can be operated repeatedly, Click right key or press ESC key to terminate

such command.

【Example】

Example one: drawing Right-angled triangle



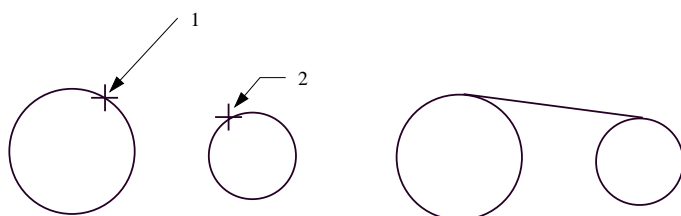
Pic3-3 two point line

Specify one point, a green line segment will pop up while moving mouse, when converting to orthogonal, input coordinate value or input distance to define the position of point 2 and point 3.

Example 2: Common tangent line of circle

Draw multiple special lines by making full use of tool point menu. See the following example of drawing tangent line.

Click Command line, the system will hint " input first point", press spacebar, tool point menu will pop up, click T , and select a circle according to the hint. When point 1 is selected, user can choose point 2 by the same method. Then a line is drawn, as show in figure (b).



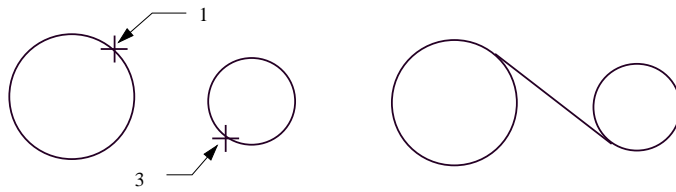
(a)Before operation

(b)after operation

Pic3-4 external common tangent of circle

Note: the position of common tangent line is different if the selected point is different.

As shown in figure, if the second point is at the position of 3, the line will be internal common tangent line.

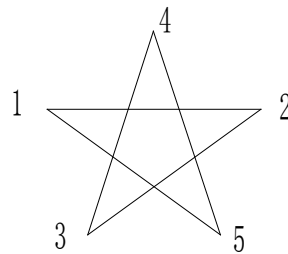


(a)Before operation (b)after operation

Pic3-5 inner common tangent of circle

Eg. three: Draw a pentagram will length 20 by relative coordinate and polar coordinate

Click command Lpp, (0,0) for the first point, input "@20,0" for the second point, it is a coordinate related to **1**, input @20<-144 for point three, it is a polar coordinate related to **2**, beginning from X axis positive side, anticlockwise is positive, clockwise is negative, input point four@20<72 and point five @20<-72 by the same method, input (0,0) at last to turn to **1**, click mouse right key to finish such operation, figure is produced.



Pic3-6 pentacle

3.2.2.2 Angle line

【Command】 la

【Icon】

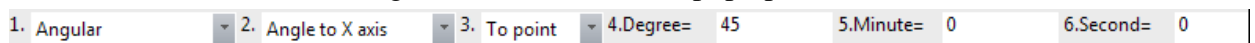
【Definition】 Draw angle line

Draw a line as per the give angle and give length., the given angle means inclination between target line and known line, or X axis, or Y axis.

【Step】

User can implement command La in the forms of : click La button in the sub-menu of Line, execute Line command, then select angle line in the immediate menu, implement command of La.

Click command La, the following immediate menu will pop up.



Pic3-7 immediate menu of angle line

(1) Click **2: in immediate menu**, the figure as below will pop up, choose the **inclination type, angle with line** is selected, that means a line will be drawn as per the given angle to the given line, then select line and input the first point and the second point to complete.

(2) Click **3: to point**, it will be **to line**, then the pointed end point is on the selected line, it will hint "select to

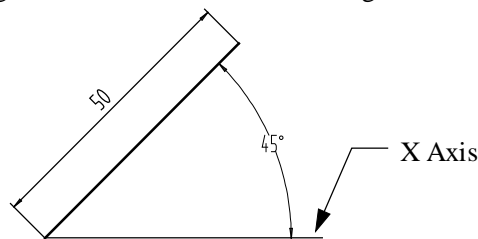
line" instead of **input second point**.

(3) Click **degree**, minute , sec , then **input real number** in the right edit box accordingly, the value in the edit box is default angle degree in immediate menu.

(4) Input the first point as per instruction requirement, there will be a mark for such point on the screen, and the hint will " input length or point". If input a number and press **enter** key, a desired line will be produced, when moving mouse, a green line will occur, define the cursor position and press left key to draw a given length and angle line.

【Example】

The angle between a line with length of 50mm and X-axis is 45 degree , as shown in figure.



Pic3-8 draw a angle line

3.2.2.3 Draw angular bisector

【Command】 lia

【Icon】

【Definition】 Draw angular bisector as per given parameter.

【Step】

User can implement command Lia in the forms of : click Lia button in the sub-menu of Line, execute Line command, then select divide angle evenly in the immediate menu, implement command of Lia.

Click command Lia, the following immediate menu will pop up.

1. Angular bisector	2.Number	2	3.Length	100
---------------------	----------	---	----------	-----

Pic3-9 immediate menu for angular bisector

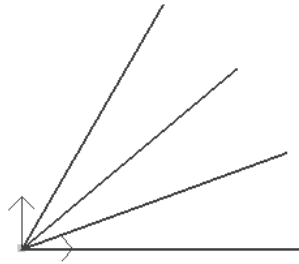
(1) Click **2: number** and input a real number in the hint area as per requirement, the given number is default in the immediate menu.

(2) Click **3: length** and input a real number in the hint area as per requirement, the given number is default in the immediate menu.

(3) When the value is input in the immediate, it will prompt “Select first line”, and confirm, select second line as per prompt, then angular bisector will pop up for the known angle.

【Example】

A 60 degree angle is divided into 3 even angle by two 100mm bisectors, as shown in following figure.



Pic3-10 Draw angular bisector

3.2.2.4 Draw tangent line / normal

【Command】 ltn

【Icon】

【Definition】 Draw tangent line / normal for given line from the given point

【Step】

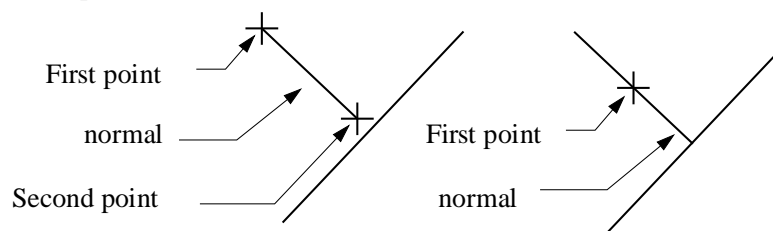
User can implement command Ltn in the forms of : click Ltn button in the sub-menu of Line, execute Line command, then select tangent line/normal in the immediate menu, implement command of Ltn.

Click command Ltn, the following immediate menu will pop up.



Pic3-11 immediate menu for tangent line / normal

(1) Click **2: tangent**, it will be normal, then a line, perpendicular to the given line, is produced as per the modified immediate menu, as shown in figure 4.12. if Tangent is selected, a line that is parallel to the given line will be produced.



Pic 3-12 normal of line

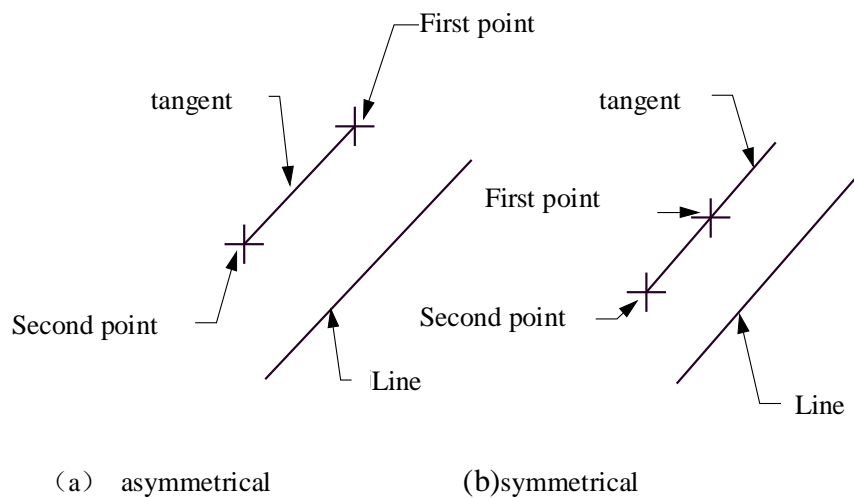
(2) Click **3: dissymmetry**, it will be symmetry, then the first point selected is mid point for the to be drawn line, and the second point selected is end point, as shown in figure 4.13 (b) and 4.14 (b).

(3) Click **4: to point**, it will be **to line**, that means the end point of tangent line or normal is on the given line .

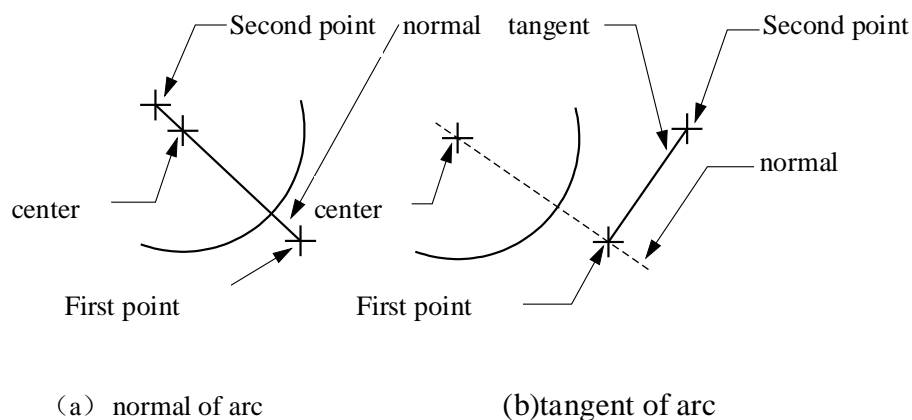
(4) Select a given line as per current instruction, the hint is "**input one point**", define a point by mouse, the hint will be "**input next point or length**", move the cursor a line segment passing the first point and parallel to the given line is produced, the mouse and number input by keyboard can determine its length .As shown in figure (c).

(5) If a circle or an arc is selected, the above operation is also applicable, the arc normal must be on the

line with first point and centre of circle, and tangent is perpendicular to normal.



Pic3-13 line tangent



Pic3-14 tangent line / normal of Arc

3.2.2.5 Bisector of two line segments

【Command】

【Icon】

【Definition】 To produce a series of lines between two disjoint lines in equal distance.

To product bisector of two line segment, it must meet one of the following conditions:

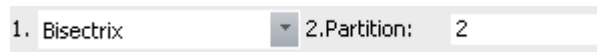
- (1) Two parallel lines
- (2) Neither parallel, nor intersected, extension line at any orientation of any line won't be intersected to the other line itself.
- (3) Unparallel, one end point of a line is coincided with end point of the other line, what's more, angle between these two lines is not 180 degree

Note: The definition of Bisector of two line segments and Angular bisector is different, angular bisector is divided as per angle degree, but Bisector of two line segments is divided as per distance of end point connected line.

【Step】

User can implement command in the forms of : click button in the sub-menu of Line, execute Line command, then select divide equally in the immediate menu, implement command of .

Click command , the following immediate menu will pop up.

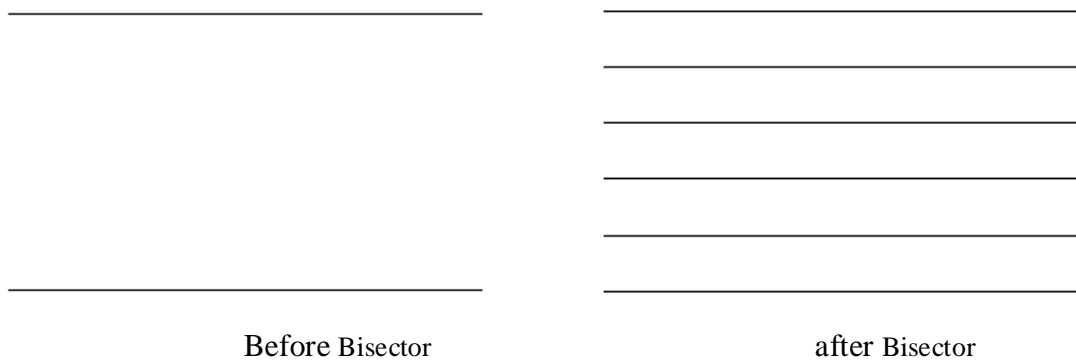


Pic3-15 immediate menu for Bisector

When Bisector of two line segments command is implemented, selected two line segments that meet above mentioned condition, then a series of lines between two disjoint lines in equal distance will be produced.

【Example】

select two parallel lines, the number is 5.the result will be shown in following figure.



Pic3-16 example of Bisector

3.2.3 Parallel line

【Command】 ll

【Icon】

【Definition】 Draw parallel line to the existing line.

【Step】

User can implement command ll in the forms of : Click command ll in the main menu of Drawing, click ll button in the tool bar of Drawing, Click button in the basic drawing panel from the common option card, execute ll command.

Click command ll, the following immediate menu will pop up.

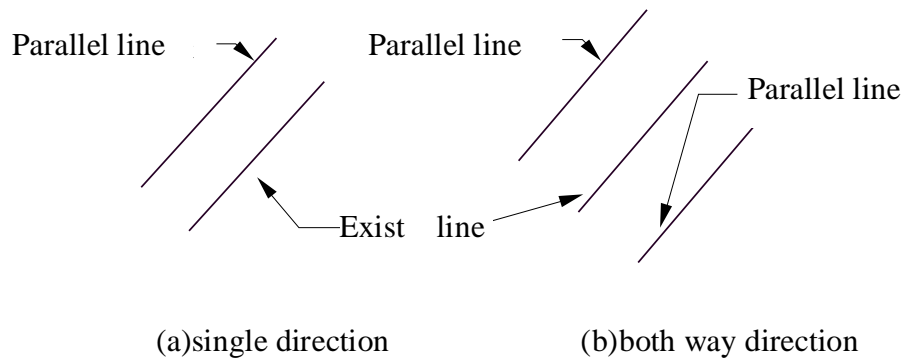


Pic3-17 immediate menu for Parallel line

- (1) In **1: of immediate menu**, choose offset mode or two points mode .
- (2) Click **2: single direction**, it will be **bi-direction**, then two parallel line as long as the given line will be produced. For single direction, input distance via keyboard, the system will judge the line position according to the position of + cursor..
- (3) When two points mode is selected, user can select point or distance in 2: of immediate menu, then draw corresponding line.

- (4) If offset mode is selected, user can select a given line, input distance or point as per hint, move mouse, a parallel line has the same length as the given line is being dragged. Once the position is determined, click left key to draw. The distance can also be inputted via keyboard.

Example: single direction parallel line (a), bi-direction parallel line(b)



Pic3-18 Draw Parallel line

3.2.4 Circle

【Command】 circle

【Icon】

【Definition】 Draw circle as per different given parameter.

To create circle, user can specify center, radius, diameter, point in the circle and combine it with point on other object. If needed, circle center line can be selected in the immediate menu. By default, the circle has no center line.

【Step】

User can implement command circle in the forms of : Click command Circle in the main menu of Drawing, click button in the tool bar of Drawing, Click button in the basic drawing panel from the common option card, execute Circle command.

Click command circle, the following immediate menu will pop up.



Figure 3-19 Circle immediate menu

CAXA Draft supplies several kinds of methods to draw circle. In the immediate menu, user can choose different modes to draw circle. In the meantime, each kind of circle can be produced individually. Following is the detail.

3.2.4.1 Draw circle via given centre and radius

【Command】 cir

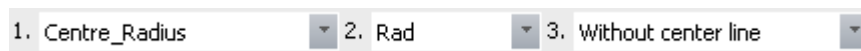
【Icon】

【Definition】 Draw circle through known circle center and radius

【Step】

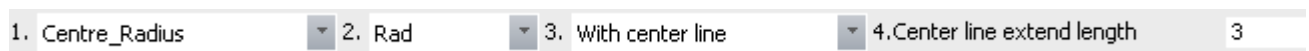
User can implement command cir in the forms of : click cir button in the sub-menu of Circle, execute Circle command, then select center radius in the immediate menu, implement command of cir.

Click command cir, the following immediate menu will pop up.



Pic3-20 Center radius circle immediate menu

- (1) Input centre as per instruction, then it hint " input radius or a point in circle", input radius value via keyboard and press **enter** key, or move cursor to define a point in circle and press left key.
- (2) Click 2: the content will changed from radius to **diameter**, after inputting centre, it will hint " input diameter or a point in circle", then input value for diameter via keyboard.
- (3) Click 3: then no central line will be changed to central line, following is the figure.



Pic3-21 Para of center line

- (4) The above command can be proceeded repeatedly, press right key to terminate.

3.2.4.2 Draw two point circle

【Command】 cppl

【Icon】

【Definition】 Draw a circle via given two point, the distance between these two points is diameter dimension.

【Step】

User can implement command cppl in the forms of : click cppl button in the sub-menu of Circle, execute Circle command, then select two point in the immediate menu, implement command of cppl.

Click command cppl, the following immediate menu will pop up.

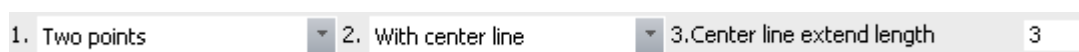


Figure 3-22 Two point circle immediate menu

Input point one and point two as per prompt, then a circle is drawn.

The above command can be proceeded repeatedly, press right key to terminate.

3.2.4.3 Draw circle via three point

【Command】 cPPP

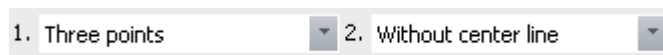
【Icon】

【Definition】 Draw circle via three point in the circle.

【Step】

User can implement command cPPP in the forms of : click cPPP button in the sub-menu of Circle, execute Circle command, then select three point in the immediate menu, implement command of cPPP.

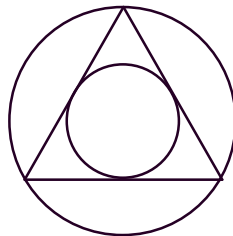
Click command cPPP, the following immediate menu will pop up.



Pic3-23 immediate menu for three point circle

Input point one, point two and point three as per instruction, a circle is drawn. Make full use of intellectual point, grid point, navigating point and tool point when inputting point. The above command can be proceeded repeatedly, press right key to terminate.

【Example】 Draw incircle and circumcircle by three point and toolbar menu



Circle 3-24 Three point circle

3.2.4.4 Draw circle via two point-radius

【Command】 cPPR

【Icon】

【Definition】 Draw circle via given radius and passing the given two point.

【Step】

User can implement command cPPR in the forms of : click cPPR button in the sub-menu of Circle, execute Circle command, then select two point radius in the immediate menu, implement command of cPPR.

Click command cPPR, the following immediate menu will pop up.



Figure 3-25 Two point radius immediate menu

Input point one and point two as per instruction, use mouse or keyboard to input point three or input a radius value via keyboard, a circle can be drawn then. The above command can be proceeded repeatedly, press right key to terminate.

3.2.5 Arc



【Command】 arc

【Icon】 

【Definition】 Draw arc as per given parameter

User can specify and combine center, end point, start point, radius, angle to create arc.

【Step】

User can implement command arc in the forms of : Click command arc in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the basic drawing panel from the common option card, execute Circle command.

Click command arc, the following immediate menu will pop up.



Figure 3-26 Arc immediate menu

CAXA Draft supplies all kinds of modes to draw arc, such as three point arc etc, user can select modes and parameter to create arc. In the meantime, each kind of arc can be produced individually. Following is detail.

3.2.5.1 Three point arc

【Command】 appp

【Icon】

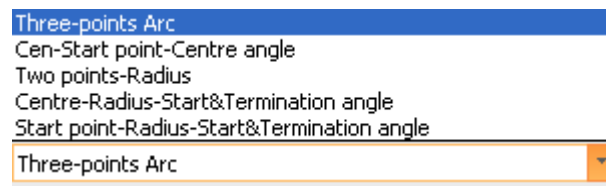
【Definition】 Draw arc through three points.

Three point arc, point one is start point, point three is end point, point two determines the position and direction of arc.

【Step】

User can implement command arcppp in the forms of : click arcppp button in the sub-menu of Arc, execute Arc command, then select three point in the immediate menu, implement command of appp.

Click command arcppp, the following immediate menu will pop up.



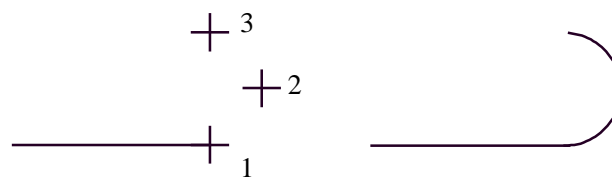
Pic3-27 immediate menu for three point arc

Define point one and point two, a three-point arc is shown on screen, move cursor and select the desired three points, press left key to draw an arc. When selecting point three, the intellectual point, navigating point or grid point can be use freely. Meanwhile, point coordinates are allowed to be inputted via keyboard.

【Example】

Select **three point arc**, when it hints "first point", press **spacebar**, a shortcut menu will pop up, in which user can click T, then select first arc as per instruction, input the second point of arc, when it hint **third point**, input it according to method of first point, select tangent of second arc, an new arc is drawn.

Example one, Eg.1: Draw arc tangent to line



(a)Select point

(b)finished

Pic3-28 Draw arc tangent to line

Eg.2: Draw arc tangent to arc of circle



(a)Select point

(b)finished

Pic3-29. Draw arc tangent to arc of circle

3.2.5.2 Centre , start point and centre angle for drawing arc

【Command】 acsa

【Icon】

【Definition】 Draw arc by known center, start point, center angle

【Step】

User can implement command acsa in the forms of : click acsa button in the sub-menu of Arc,

execute Arc command, then select centre , start point and centre angle in the immediate menu, implement command of acsa.

Click command acsa, the following immediate menu will pop up:



Pic3-30 immediate menu for 'Centre , start point and centre angle arc'

Input centre and arc start point as per instruction, when the hint is " centre angle or end point", input centre angle value or end point, the arc is drawn, it can also be selected by dragging mouse. The above command can be proceeded repeatedly, press right key to terminate.

3.2.5.3 Draw arc via given two point and radius

【Command】 appr

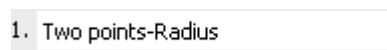
【Icon】

【Definition】 Draw arc via given two point and radius.

【Step】

User can implement command appr in the forms of : click appr button in the sub-menu of Arc, execute Arc command, then select two point and radius in the immediate menu, implement command of appr.

Click command appr, the following immediate menu will pop up:



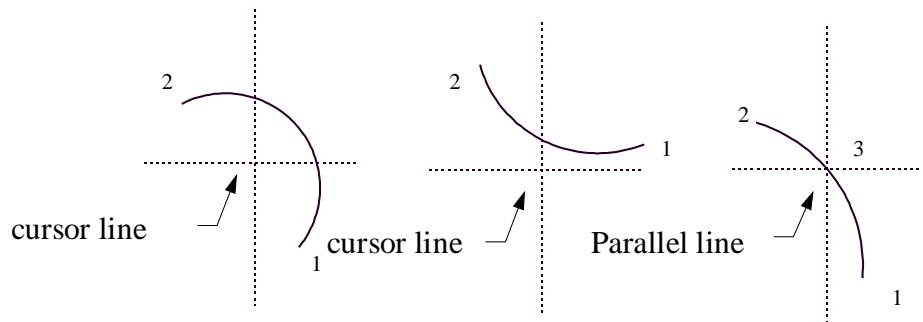
Pic3-31 immediate menu for two point and radius arc

Input **first point** and **second point** as per instruction, then it hints " third point or radius", input radius value, the system will judge arc direction as per current position of + cursor , the rule is: where the + cursor stays , where is the position of arc, as shown in figure (a),(b). Different cursor position will get arcs by different direction. If drag mouse when second point is input, an arc is formed by the inputted two point and the cursor position. The arc will change when the cursor drags. Press left key to finish this operation, as shown in figure (c).

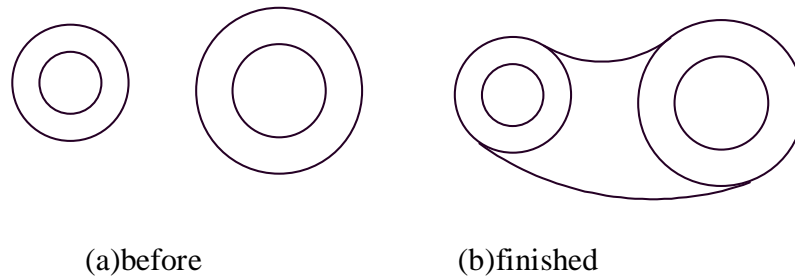
The above command can be proceeded repeatedly, press right key to finish.

【Example】

Eg.1: arc via given two point and radius



Eg.2: arc tangent to circle.



(a)before

(b)finished

Pic4-21 arc tangent to circle

3.2.5.4 Draw arc via given centre, radius , start and terminal angle

【Command】 acra

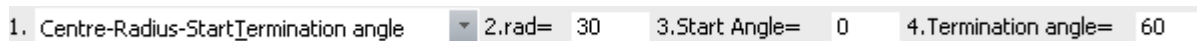
【Icon】

【Definition】 Draw arc via given centre, radius , start and terminal angle

【Step】

User can implement command acra in the forms of : click acra button in the sub-menu of Arc, execute Arc command, then select centre, radius , start and terminal angle in the immediate menu, implement command of acra.

Click command acra, the following immediate menu will pop up:



Pic3-34 immediate menu for 'centre, radius , start and terminal angle arc'

- (1) Click **2: radius**, the hint will be " input real number", the default is in the edit box, radius value can be put via keyboard.
- (2) Click **3:** or **4:** in immediate menu, input degree (-360,360) for start angle or terminal angle as per instruction. When new number is input, the due content will be changed in immediate menu. Start angle and terminal angle begin at X straight semi-axis positive side, counter clockwise is positive, clockwise is negative.
- (3) The condition for arc to be drawn is shown in immediate menu. Input centre point as per instruction, and an arc will move as the cursor moves. The value of radius, start/end angle are set, when the centre is selected, press left key ,an arc is produced on screen.

(4) The above command can be proceeded repeatedly, press right key to terminate.

3.2.5.5 Draw arc via given start point-end point-centre angle

【Command】 asear

【Icon】

【Definition】 Draw arc via given start point-end point-center angle.

【Step】

User can implement command asear in the forms of : click asear button in the sub-menu of Arc, execute Arc command, then select start point-end point-centre angle in the immediate menu, implement command of asear.

Click command asear, the following immediate menu will pop up:

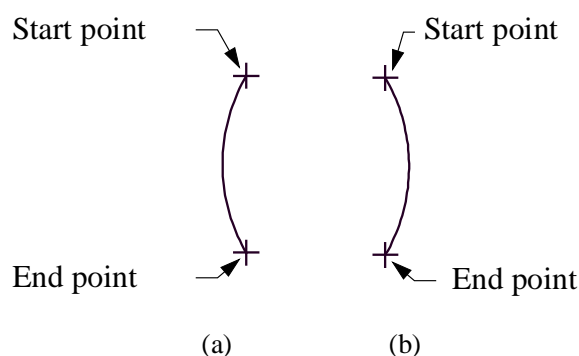


Pic3-35 immediate menu for 'start point-end point-centre angle arc'

- (1) Click **2: centre angle**, input centre angle degree (-360, 360) as per instruction, negative angle means draw arc clockwise from start point to end point, positive angle means draw arc counter clockwise from start point to end point. Press **enter** key when degree is input.
- (2) Input start point and end point as per instruction.
- (3) The above command can be proceeded repeatedly, press right key to terminate.

【Example】 :

Start point and end point are the same, but the symbol for centre angle is different, then the arc direction will be different, as shown in figure, centre angle for (a) is 60 degree, centre angle for (b) is -60 degree.



Pic3-36 draw arc of start point-end point-centre angle

3.2.5.6 Draw arc via given start point-radius-start/end angle

【Command】 asra

【Icon】**【Definition】** Draw arc via given start point-radius-start/end angle**【Step】**

User can implement command asra in the forms of : click asra button in the sub-menu of Arc, execute Arc command, then select start point-radius-start/end angle in the immediate menu, implement command of asra.

Click command asra, the following immediate menu will pop up:



Pic3-37 immediate menu for 'start point-radius-start/end angle arc'

- (1) Click **2:**, input radius value as per instruction.
- (2) Click **3:** or **4:** in immediate menu, as per system instruction, input value for start angle or terminal angle according drawing requirement. The condition in immediate menu will change accordingly.
- (3) The condition for the to be drawn arc is shown in the immediate menu. Input start point, radius value, start angle and end angle as per requirement to draw desired arc. The start point can be input via mouse or keyboard.

3.2.6 Draw rectangle

【Command】 rect**【Icon】** **【Definition】** Use closed polyline to form a rectangle

There are two modes to create rectangle, two points , width and length.

【Step】

User can implement command arc in the forms of : Click command arc in the main menu of Drawing, click button in the tool bar of Drawing, Click button in the basic drawing panel from the common option card, execute Rect command.

Click command Rect, the following immediate menu will pop up.



Pic3-38 draw rectangle with two points

Choose **two point** in **1:** of **immediate menu**, then determine the first angle point and the second angle point as per instruction. When deciding the second angle point, a changing rectangle occur, press left key if the position defined, and a desired rectangle is drawn. Or input absolute coordinates or relative coordinates for two angle point via keyboard directly. Eg. The first angle point coordinates is (20,15), the length is 36, the width is 18, then the absolute coordinates for the second angle point is (56,33), its relative coordinates is (@36,18). Using relative coordinate will be more simple when the length and width of rectangle are given , by choosing **two point**.

If click 1:, then it will change to Width and length, immediate menu is shown as follows:



Pic3-39 immediate menu for 'length and width rectangle'

- (1) Click 2: the centre localization will be changed to left corner localization, that is to say, the mid point in upper line is the positioning point for drawing a rectangle..
- (2) Click 3: angle , 4: length, 5: width in immediate menu, it will hint "input real number", then input obliquity ,length, width value to draw a new rectangle.
- (3) Above immediate menu shows: draw a rectangle with 200 length and 100 width , and centre localization, 0 degree obliquity. Define a localization point to draw a rectangle. Before point definition, a rectangle occur and will move with the moving cursor, if the center is localized, a desired rectangle is drawn.

3.2.7 Polyline

【Command】pline

【Icon】

【Definition】Polyline is combined by a series of continued individual line segment

By the command of Pline, user can create straight line segment, arc line segment or the combination of line segment and arc.

【Step】

User can implement command Pline in the forms of : Click command Pline in the main menu of Drawing, click button in the tool bar of Drawing, Click button in the basic drawing panel from the common option card, execute Pline command.

Click command Pline, the following immediate menu will pop up.



Pic3-40 immediate menu for polyline

- (1) Specify first point and second point as per prompt, then one line segment can be produced, then specify next points continually to draw constant combined line.
 - (2) Click 【2.】 in the immediate menu to set close or unclose,
 - (3) Click 【3.】 and 【4.】 in the immediate menu to specify polyline start width and terminate width.
- Click 【1.】 in the immediate to switch to arc, the following figure will pop up.




Pic3-44 immediate menu for polyline two

Specify first point and second point as per prompt, then one arc can be produced, then specify next points continually to draw constant combined arc.

Line and arc can be combined continually, user can switch via immediate menu, When drawing line and arc, it will enhance efficiency if user use dynamic input and intellectual point.



3.2.8 Draw hatching

【Command】 hatch

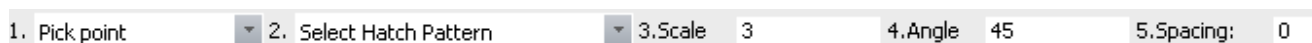
【Icon】 

【Definition】 Use filling pattern to fill closed area or selected object, then hatching part is produced.

【Step】

User can implement command hatch in the forms of : Click command hatch in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the basic drawing panel from the common option card, execute hatch command.

Click command hatch, the following immediate menu will pop up.



Pic3-45 immediate menu for hatching

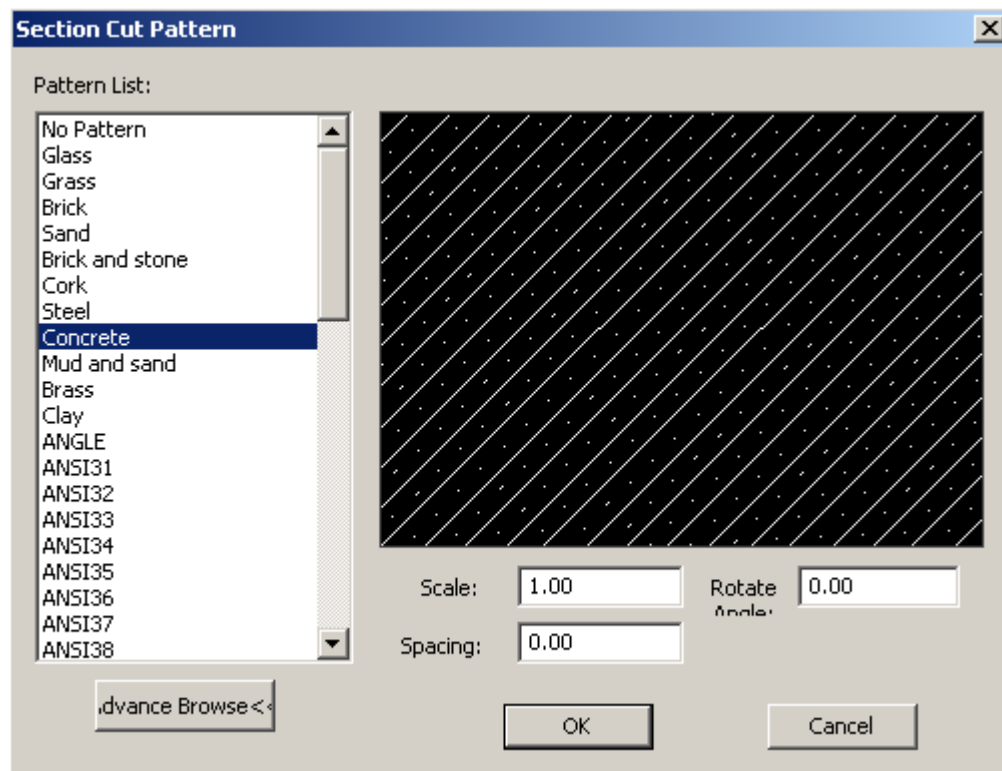
There are two modes to create hatching, one is select point, the other is select edge.

3.2.8.1 Select point to draw hatching

【Definition】 Search minimum loop from right to left according to the position of selected point, and hatching is produced for the loop. If the selected point isn't in the ring, the operation is invalid.

【Step】

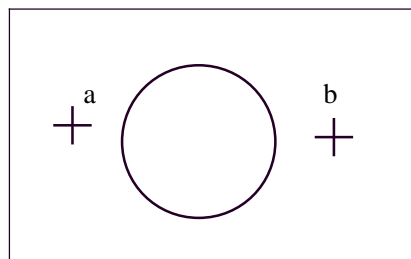
- (1) Click command hatch, choose **Select point** in 1: of immediate menu.
- (2) Click **【2: 】** in the immediate menu, user can choose option **【not select hatching】** or **【select hatching】**, once **【not select hatching】**, the pattern will be produced by default. If **【select hatching】** is chosen, the following dialog box will pop up, in which user can set hatching line scale, rotate angle, space between or other parameters.



(3) Then click OK button, select a point in closed loop by press left key, each curve in closed ring will be red, press right key to confirm. A hatching is drawn in the ring as per the definition in immediate menu. The operation is simple and convenient. It is applicable for various closed area.

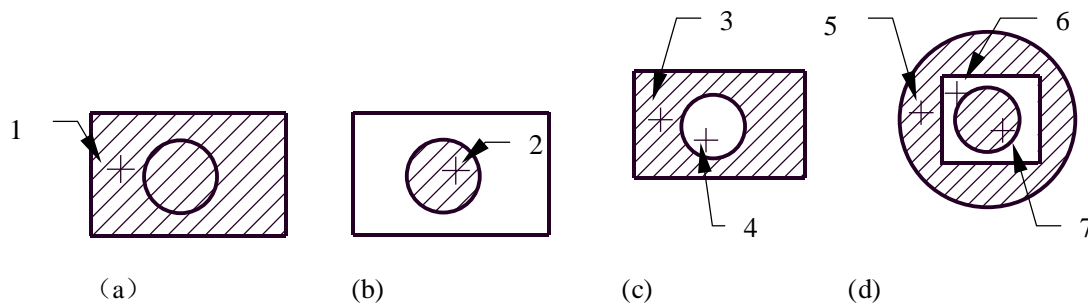
Note: the position of selecting point is in the loop, when the point is selected, system will start from the selected point to search the minimum closed loop from right to left

As shown in following figure , rectangle is a closed loop, and the inner circle is also a closed loop. If the selected point is at " a " position, than system will start to search to the left, the minimum closed loop is rectangle. If the selected point is at " b " position, then system will start to search to the left, the minimum closed loop is circle, this point isn't in the circle, no hatching can be produced.



Pic3-46 position of select point

Different position of selected point will draw different hatching, as shown in figure (a) and figure (b), in figure (c), select point 3 and select point 4 and a section with hole is drawn. In figure (d), select point 5 first, then select point 6, at last select point 7 to draw a complicated section.



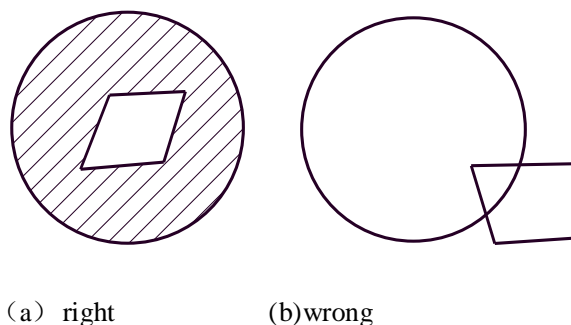
Pic3-47 example of hatching

3.2.8.2 Select border to draw hatching

【Definition】 Search curves that can be intersect and form closed loop as per selected curve. If the selected curves can not form a closed loop , the operation is invalid.

【Step】

- (1) Click command hatch, choose **select border** in 1: of immediate menu.
- (2) Confirm hatching pattern and parameter
- (3) Move the cursor to select several intersected curve that can form a closed loop, press right key to confirm, and the hatching is displayed , otherwise the operation is invalid, as shown in figure (a) and (b)
- (4) When the closed loop can't be produced by selecting border, use select point. In the specified area produce hatching. Eg. A little block is overlapped between a circle and a quadrangle in figure (b), hatching can't be produced by select border, but use select point can get hatching easily.



(a) right

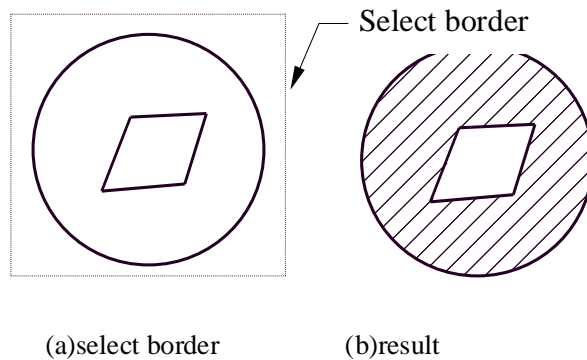
(b) wrong

Pic3-48 select border

The number for border to be selected is not limited , and the selected border will be red, press right key to finish selecting. The indefinite selection can't draw hatching, and the selected curve will renew after confirmation. The hatching occur in the closed loop when confirmed.

Example:

Select border (by window selection or single selection) to draw hatching, as shown in figure.



Pic3-49 draw hatching with selecting border

3.2.9 Filling

【Command】 solid

【Icon】

【Definition】 Fill solid within a closed area.

It is a figure type actually , can fill in closed area . Use **fill** to black some parts' section when necessary.

【Step】

User can implement command solid in the forms of : Click command solid in the main menu of Drawing, click button in the tool bar of Drawing, Click button in the basic drawing panel from the common option card, execute solid command.

Click solid command in drawing tool bar, select closed area to be filled via pressing left key to finish such operation.



3.2.10 Draw centre line

【Command】 centerl

【Icon】

If select a circle, arc or ellipse, two orthogonal centre line will be produce. If select two parallel line or non-parallel line (such as a cone) , the centre line for two lines will be produced.

【Step】

User can implement command centerl in the forms of : Click command centerl in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the basic drawing panel from the common option card, execute centerl command.

select closed area to be filled via pressing left key to finish such operation.

(1) Click centerl command in drawing tool bar, the following immediate menu will pop up.

1.&Extend beyond extension Line: 3

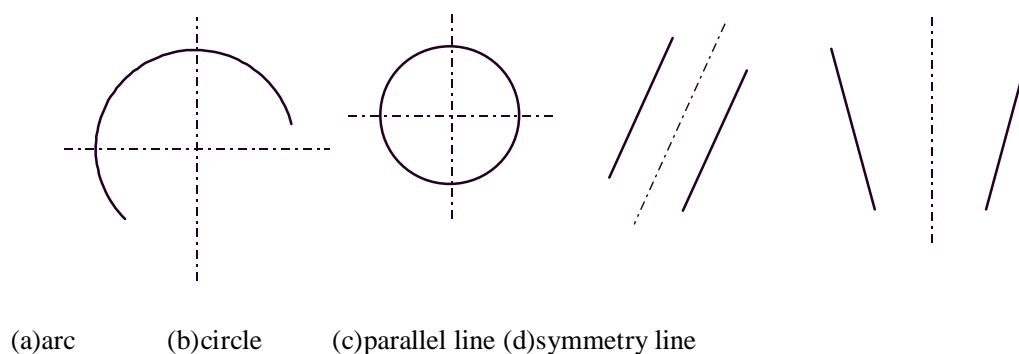
(2) Click **1: stretch beyond extension** (beyond the contour line), the operation hint will be "input real number", it is the default in the current box, input new extension length via keyboard, as shown in figure.

(3) Select the first curve as per requirement, if a circle or arc is selected, two mutual orthogonal centre line that extend beyond contour occur. If a line is selected, it will hint "select its parallel line", when two lines are selected, a centre line will be drawn between them.

(4) The above command can be proceeded repeatedly, press right key to terminate.

Example:

Draw centre line



Pic3-50 draw center line

3.2.11 Draw offset line

【Command】 Offset

【Icon】 



【Definition】 Draw offset line

User can create offset line from the object of line, arc, circle, ellipse, polyline, spline.

It has the function of chain select, the end to end graph element can be regarded as a whole part to offset.

【Step】

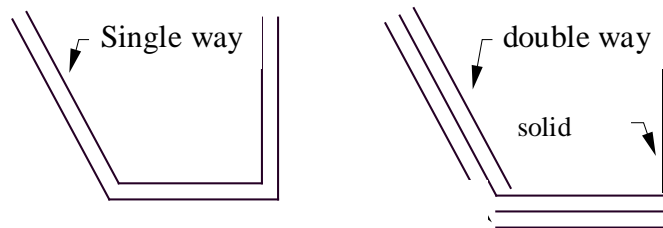
User can implement command offset in the forms of : Click command offset in the main menu of

Drawing, click  button in the tool bar of Drawing, Click  button in the basic drawing panel from the common option card, execute offset command.

Click command offset, the following immediate menu will pop up.

1. Single select	2. Specified the distance	3. Single direction	4. Hollow	5.Distance	5	6.Number	1
------------------	---------------------------	---------------------	-----------	------------	---	----------	---

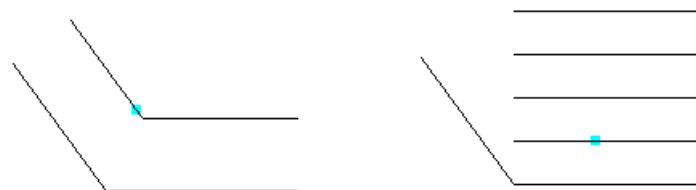
- (1) If choose **single select**, single element will be selected. If choose **select by chain**, it will select end to end element with the current element together, as shown in figure.
- (2) Choose **specified distance** in **【2: 】** of immediate menu, that means selecting arrow direction to define offset direction, the specified distance for producing offset line for given curve . if choose **landscape point**, it means producing offset line for given curve via specified point.
- (3) Choose **single direction** in **【3: 】** of immediate menu, means drawing offset line at one side. **Bi-direction** means drawing offset on the line's both sides.
- (4) Choose **solid** in **【4: 】** of immediate menu, means the space between original curve and offset line is filled. If choose **hollow**, that means just drawing offset line, there is no filling.
- (5) If it is **specified distance**, click **【5: distance 】** and input the distance value , there is default in the editing box.
- (6) Click **【6: number】** in the immediate, input needed offset line number.



(a)select by chain

(b)single select

Pic3-51 Draw offset line by specified distance



(a)select by chain

(b) single select

Pic3-52 Draw offset line by across point

3.3 Draw superior curve

Superior graph includes spline, point, ellipse, formula curve, polygon, wave line etc.

Superior curve is a specified curve drawing combined by basic element. They can meet some special requirement when designing.

3.3.1 Spline



【Command】 spline

【Icon】 

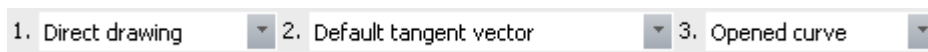
【Definition】 A smooth curve passing a series of given point.

Draw spline via given apex, the apex can be input via mouse or keyboard, or read from outer file.

【Step】

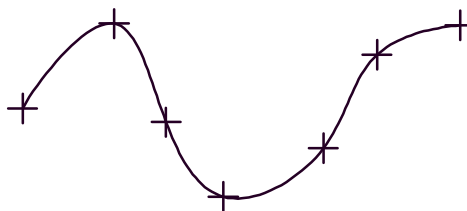
User can implement command Spline in the forms of : Click command spline in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the Superior drawing panel from the common option card, execute spline command.

(1) Click button , following immediate menu will pop up:



- (2) If choose **straight drawing**, input a series controlling point via mouse or keyboard as per the system hinting. A smooth spline is drawn.
- (3) If select **read from file**, the dialogue box **open spline data file** pop up, click button **OK** , then a spline can be drawn as per the data in file.
- (4) When drawing spline, user can select open or closed as per requirement for batch input in the immediate menu of **【3: open】** .


【Example】 Draw a spline through a series of interpolation point.



Pic3-53 Draw Spline

3.3.2 Point



【Command】 Point


【Icon】 

【Definition】 Draw point on the screen

Draw an isolated point at specified position of screen or draw equinoctial point on curve.

【Step】

User can implement command point in the forms of : Click command point in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the Superior drawing panel from the common option card, execute point command.

1. Isolated point 

(1)Click **1**: in **immediate menu**, there are three option: **isolated point**, **equinoctial point** or **equal-arc-length point**.

(2) If select **isolated point**, input point via mouse or keyboard directly, draw end point, mid point, centre or other feature point by tool point menu.

(3) If select **equinoctial point**, click **2: equinoctial number**, input value and select line to be parted, then it is parted instead of broken. If some curve needs breaking, refer to [Break of curve edit](#) in next chapter.

(4) If select **equal-arc-length point**, the arc will be parted as per specified length. Click **2**: in immediate menu, specified arc length and specified arc length by two points can be switched. If user chooses specified arc length, input equinoctial number in **3**: of immediate menu, and specify arc length in **4**: of immediate menu, then select the curve to be parted ,select **start point** and **equal-arc-length point**, an arc with equal-arc-length point is drawn .

(5) Such command can be repeated, click right key to terminate.

【Example】

Part a line into three partition

Trisect a line by point 1 and 2, click **editing tool** and choose **break**, select line as per instruction, then select point 1,if select this line again, it is easy to find the original line is broken at point 1. Use the same method to break the remaining part at point 2, the line is parted into three mutual independent line section , as shown in figure. This method is suitable to part other curve , such as circle and arc.



Pic3-54 Part a line into three partition

3.3.3 Formula curve



【Command】 fomul

【Icon】 

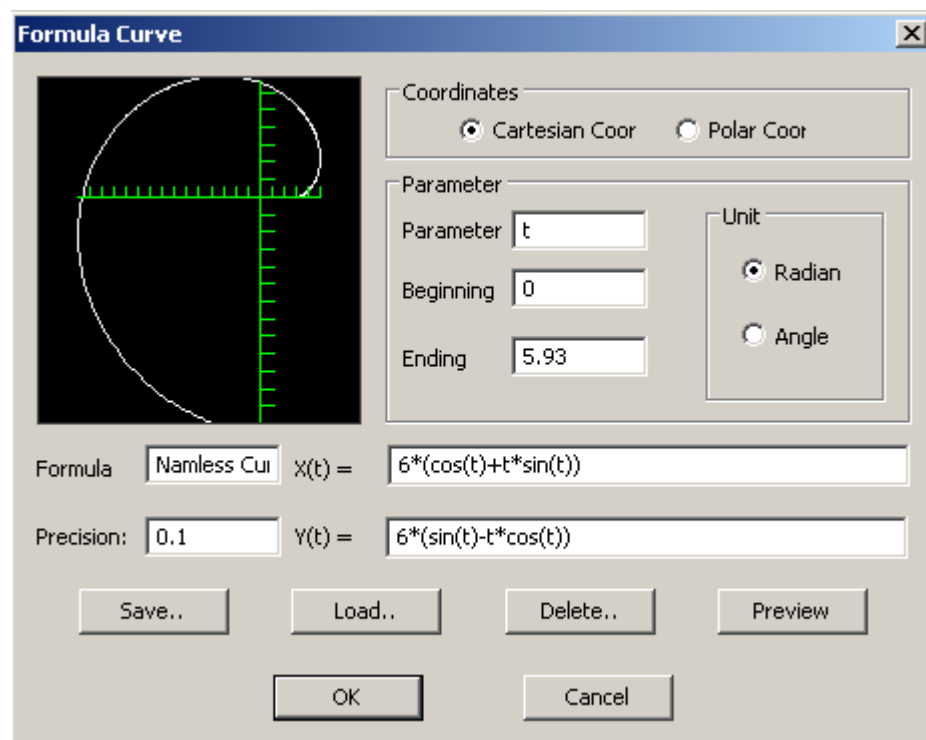
【Definition】 Formula curve is curvilinear figure by mathematical expression, draw corresponding mathematical curve according to mathematical expression or parameter expression.

The expression can be right angle coordinates or polar coordinates. User can design accurate die space and locus linear is by formula curve. Interactive enter mathematical expression, specify parameter, corresponding curve can be drawn automatically.

【Step】

User can implement command fomul in the forms of : Click command fomul in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the Superior drawing panel from the common option card, execute fomul command.

(1) Execute command fomul, following dialog box will pop up, in which user can select right angle coordinates or polar coordinates.



Pic3-55 formula curve

(2) Fill desired parameter in due block: variable name, beginning variable and ending variable value , and variable unit.

- (3) Input formula name, formula and precision. Click **preview** , the defined curve is shown in the left preview block.
- (4) There are three button **save**, **load** and **delete** in dialogue box, if click **save**, the current curve will be saved. **Load** and **delete** are for existing curve . When clicking any of them, the system will list existing curve for selection.
- (5) When the curve is set, click button **OK** , input locating point as per instruction, a formula curve can be drawn.
- (6) The above command can be proceeded repeatedly, press right key to terminate.

3.3.4 Ellipse



【Command】 ellipse

【Icon】 

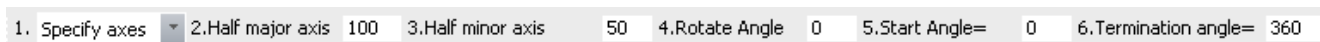
【Definition】 Draw ellipse or ellipse arc

User can create ellipse or ellipse arc in the forms of: Given semi minor axis, two points on axis, centre-start point.

【Step】

User can implement command ellipse in the forms of : Click command ellipse in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the Superior drawing panel from the common option card, execute ellipse command.

Click command ellipse, the following immediate menu will pop up.



Pic3-56 immediate menu for ellipse

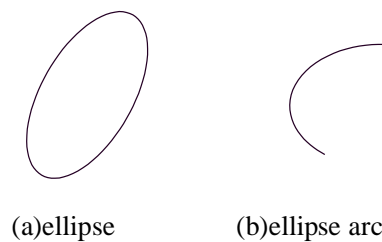
In the above immediate menu, user draw an ellipse with 0 degree rotate angle, semi major axis is 100, semi minor axis is 50, and the locating point is center . Use mouse or keyboard to input a locating point, When moving the cursor , the ellipse with semi major axis is 100, semi minor axis is 50 will also moves. once the position is determined, an ellipse is drawn.

- (1) If click 2:semi major axis or 3:semi minor axis , the system will remind re-defining its value.
- (2) If click 4:rotate angle, input the angle degree to determine the ellipse direction.
- (3) If click 5:start angle and 6:terminate angle , input the degree, when the start angle is 0 degree, and terminate angle is 360 degree, the ellipse is complete, if change the degree, will get an ellipse arc from start angle to terminate angle.
- (4) If choose two points on axis in 1: of immediate menu, the system will hint input two end point on axis, and then input the length of another axis, dragging mouse can determine ellipse shape.

- (5) If choose centre-start point in 1: , input centre point of ellipse and a start point of a axis, and then input another axis length, dragging mouse can determine ellipse shape.

Example:

Draw an ellipse and an ellipse arc according to above operation step , as shown in figure. Figure (a) is a complete ellipse with 60 degree rotate angle. Figure (b) is an ellipse arc, with a 60 degree start angle and 220 degree terminate angle.



Pic3-57 Draw ellipse

3.3.5 Draw regular polygon

【Command】 polygon



【Icon】 

【Definition】 Draw closed regular polygon.

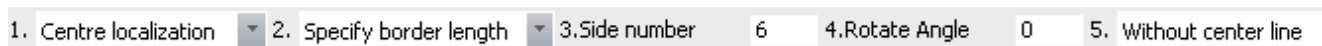
Draw a regular polygon with given point , given radius and given border number. The property of the created polygon is polyline.

User can draw polygon quickly by each kind of parameter, including radius, edge, etc.

【Step】

User can implement command polygon in the forms of : Click command polygon in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the Superior drawing panel from the common option card, execute polygon command.

Click command polygon , following immediate menu will pop up:

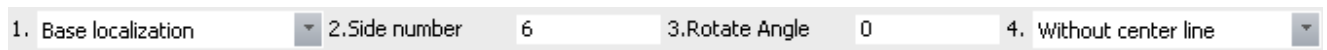


Pic3-58 immediate menu for regular polygon 1

Select 【1: centre localization】 in the immediate menu.

- (1) Click 2: can select **specify radius** or **specify border length**. If select **specify radius**, input radius of inscribing (or circumscription) circle to polygon. If select " specify border length, input the length of each border.
- (2) Click 3: can choose **inscribing** or **circumscription**, it means the polygon to be drawn is inscribing or circumscription polygon to a circle.

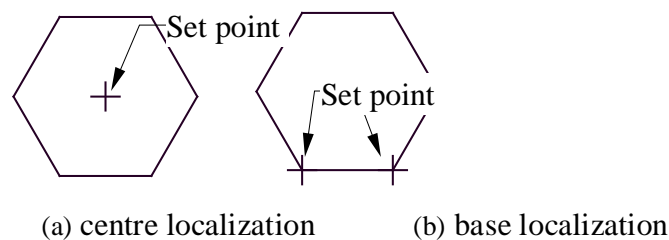
- (3) Click **4: side number**, determine the side number for polygon by inputting a new number. The side number is integer between 3 and 36.
- (4) Click **5: rotate angle**, input a new angle degree to determine the rotate angle of polygon.
- (5) When all content are set in the immediate menu, input a centre point , it will hint "the point on circle or radius of inscribing (or circumscription) circle." If input a radius value or a point on circle via mouse or keyboard, the corresponding regular hexagon is drawn. If choose centre localization in 1: of immediate menu, the menu and operation instruction will be changed as follows



Pic3-59 immediate menu for regular polygon 1

The above menu means drawing a regular polygon regarding bottom margin as localization base. Border length and rotate angle can be input as per above mentioned operation. Input the first point as per instruction, then it will hint "next point or border length", the second point or border length determine the size of polygon. When they are input, a regular hexagon can be drawn , the border length is the distance between point 1 and point 2. the rotate angle is set as per user's request.

Example:



Pic3-60 draw polygon

3.3.6 Draw Arc fit spline

【Command】 nhs

【Icon】

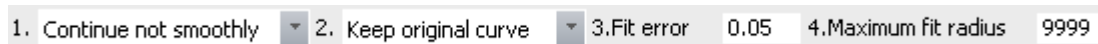
【Definition】 Use several arc to fit existing spline.

User can specify fitting precision, with inquiry function for easier code programming.

【Step】

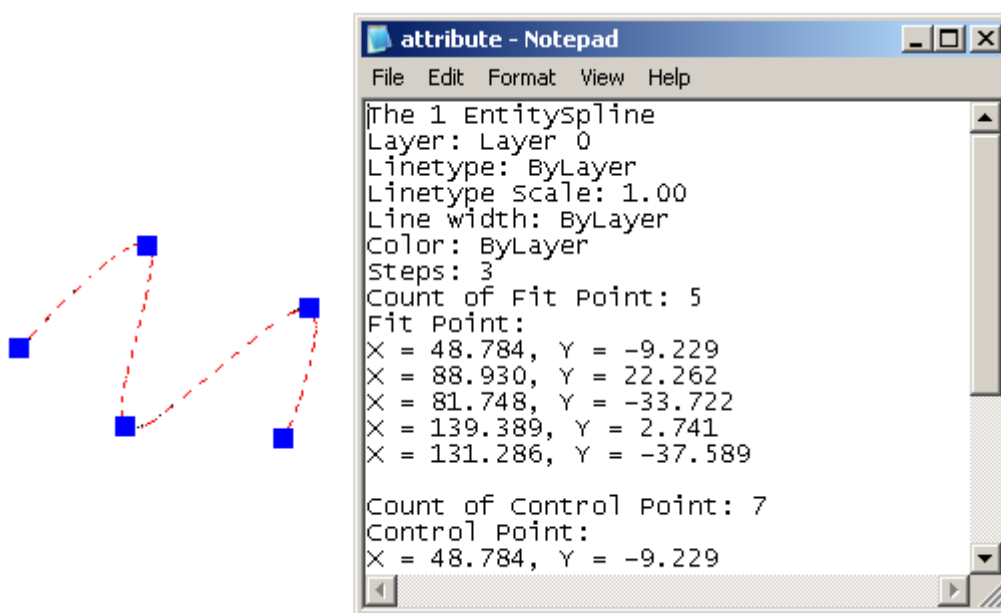
User can implement command nhs in the forms of : Click command nhs in the main menu of Drawing, click button in the tool bar of Drawing, Click button in the Superior drawing panel from the common option card, execute nhs command.

Click the command nhs, following immediate menu will pop up.



Pic3-61 immediate menu for Arc fit spline

- (1) Click **1**: in immediate menu, can choose **continuous not smooth** or **continuous smooth**.
- (2) Click **2**: in immediate menu, can choose **keep original curve** or **delete original curve**.
- (3) Click **element attribute** in **Inquiry** pull down menu of tool menu, press right key to confirm.
- (4) **Inquiry result** dialogue pop up, pull the scroll bar to see the fitting arc attribute.



Pic3-62 Inquiry result of spline

3.3.7 Part enlarge

【Command】enlarge

【Icon】

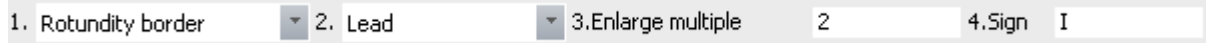
【Definition】Create enlarged part graph view as per given parameter.

At the same time, user can set edge shape, such as round edge, rectangle edge, then input text for the enlarged view, its dimension value will keep the same as that of original graph.

【Step】

User can implement command enlarge in the forms of : Click command enlarge in the main menu of Drawing, click button in the tool bar of Drawing, Click button in the Superior drawing panel from the common option card, execute enlarge command.

Click command enlarge, the following immediate menu will pop up.



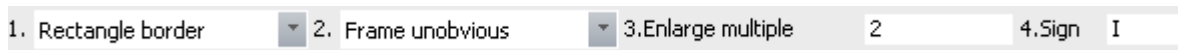
Pic3-63 immediate menu for Part enlarge

- Partly enlarged by round window

- (1) select 1: round border.
- (2) Then select 2: scale , 3: sign, user can input scale value and part view name respectively.
- (3) Input center point of partly enlarged graph, Input a point on round border, or input radius of round border.
- (4) select lead line or without lead line.
- (5) It will hint sign insert point then, if there is no need to input text, user can click right key. Otherwise, the moving cursor will determine position of text to be inputted, click left key to insert text.
- (6) Then it hints entity insert position. The virtual image of partly enlarged graph will be displayed dynamically along with moving cursor. Specify a proper position and click left key, the partly enlarged graph is shown on screen.
- (7) When a sign insert point is inputted in step (7), it hints sign insert point, move cursor to proper position and input insert point, text will be produced.

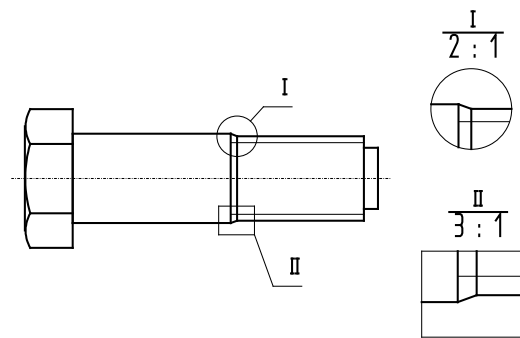
- Partly enlarged by rectangle window

- (1) select 1: rectangle border in the immediate menu.



- (2) User can switch 2:frame visible to 2:frame invisible. Then select 2: scale , 3: sign, user can input scale value and part view name respectively.
- (3) Input two rectangle angle points as per hint, if frame visible is select in step 1, rectangle frame will be shown, otherwise the frame is invisible.
- (4) A new immediate menu will pop up, in which user can select lead line or without lead line.
- (5) It will hint sign insert point then, if there is no need to input text, user can click right key. Otherwise, the moving cursor will determine position of text to be inputted, click left key to insert text.
- (6) Then it hints entity insert position. The virtual image of partly enlarged graph will be displayed dynamically along with moving cursor. Specify a proper position and click left key, the partly enlarged graph is shown on screen.
- (7) When a sign insert point is inputted in step (7), it hints sign insert point, move cursor to proper position and input insert point, text will be produced.

Example: A part enlarge example is shown in the following figure.



Pic3-64 Partly enlarged

3.3.8 Draw wave line

【Command】 wavel

【Icon】

【Definition】 Create wave line as per given mode, adjust each curvature and direction of wave curve section can change wave crest height.

【Step】

User can implement command wavel in the forms of : Click command wavel in the main menu of Drawing, click button in the tool bar of Drawing, Click button in the Superior drawing panel from the common option card, execute wavel command.

Implement command Wavel, an immediate menu will pop up.

Click **1: wave crest**, input wave crest value in the scope of -100,100 to define wave crest height.

Specify several continuous points via mouse as per menu requirement, then a wave line is shown, there is a wave crest and a wave trough between every two points, press right key to terminate.

Example:


The figure is drawn as per above operation step.



Pic3-65 Draw wave line

3.3.9 Dual fold line



【Command】 condup

【Icon】 

【Definition】 Draw dual fold line

Due to limited paper setting, use dual folding line to show some drawing can't be designed in scale, control break point distance when drawing.

【Step】

User can implement command condup in the forms of : Click command condup in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the Superior drawing panel from the common option card, execute condup command.

Click command condup, an immediate menu will pop up.

- (1) If choose **break point distance** in **1**: of **immediate menu**, input distance value in **2**: **distance**, select line or break point, a dual folding line is produced as per given break point distance.
- (2) If choose **break point number** in **1**: input desired number in **2**: **number**, select line or point to draw dual folding line as per given break point number.

3.3.10 Draw arrow



【Command】 arrow

【Icon】 

【Definition】 Draw solid arrow at specified position.

Draw solid arrow as per specified positive direction or negative direction on line, arc, spline or a certain point.

【Step】

User can implement command arrow in the forms of : Click command arrow in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the Superior drawing panel from the common option card, execute arrow command.

Implement the command arrow, corresponding immediate will pop up.

(1) Click **1**: in **immediate menu**, **positive direction** and **negative direction** can be switched. User can draw a negative or positive direction arrow on line ,circle or point.

(2) The arrow direction point is defined as follows:

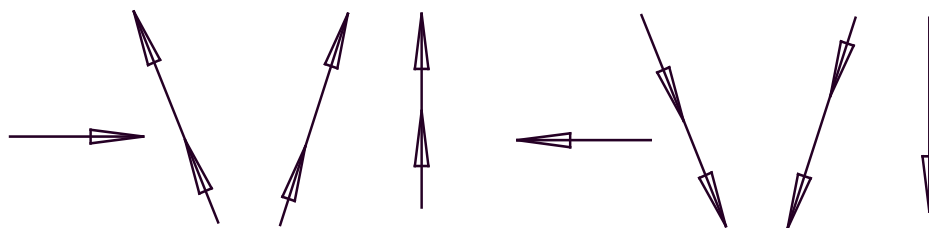
Line: the contained angle between the arrow direction and X straight semi-axis is equal to or greater than 0 degree , and less than 180 degree, it is called positive direction arrow when the contained angle is equal to or greater than 180 degree and less than 360 degree , it is called negative direction arrow.

Arc: Anti clockwise direction is arrow positive direction, clockwise direction is negative direction.

Spline: Anti clockwise direction is arrow positive direction, clockwise direction is negative direction.

Specified point: no negative or positive direction, the arrow points to specified point. refer to figure.

As shown in following figure.



(a)forward arrow

(b)reverse arrow

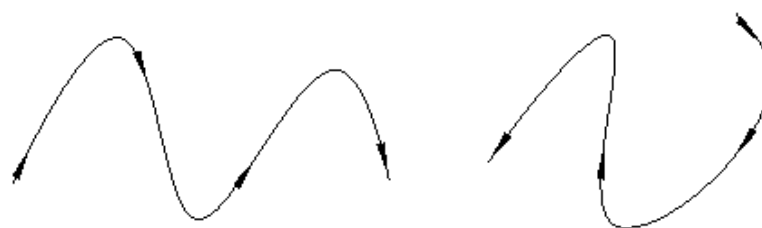
Pic3-66 arrow on line



(a)forward arrow (b)reverse arrow

Pic3-67 arrow on arc

3-68 arrow on point



(a)forward arrow

(b)reverse arrow

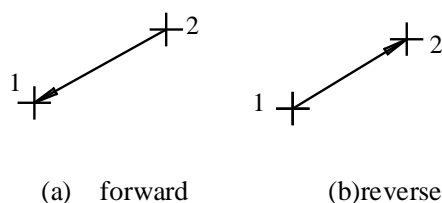
Pic3-69 arrow on spline

(3) Select line, arc or a point as per instruction, then it hints " arrow position", select definite position, and move mouse, a green arrow is shown and moving on the selected curve with the moving cursor , press left key when the desired position is chosen, the arrow is drawn.

(4) If draw an arrow on a point, user can draw a lead line from the arrow and determine its length, the arrow direction can be chosen within 360 degree, the length and direction of lead line are changed

with the moving cursor. Press left key to get a lead line with arrow. If it doesn't need leading line, press left key when arrow position is determined, don't drag the mouse.

(5) User can draw line with arrow like drawing two-point line. If selects **positive direction**, the arrow point points to the first point from the second. If select **negative direction**, the arrow point points to the second from the point, as shown in figure.



Pic3-70 line with arrow

Drawing method: Specify a point on drawing area via left key when it hints "select line, arc or one point", drag mouse, a dynamic line with arrow is changing with the moving cursor. When it is moved to appropriate position, press left key to input point two, a line with arrow is drawn.

3.4 Image

3.4.1 Summary

When drawing CAD image, some raster image, generally speaking, is needed to insert, and this raster will be combined with image. Eg. It will be regarded as base image, practicality reference or used for logo design. In CAXA Draft, image can be added to graph based on vector as reference, what's more, user can check, edit and print.



3.4.2 Insert image

【Command】 insertimage

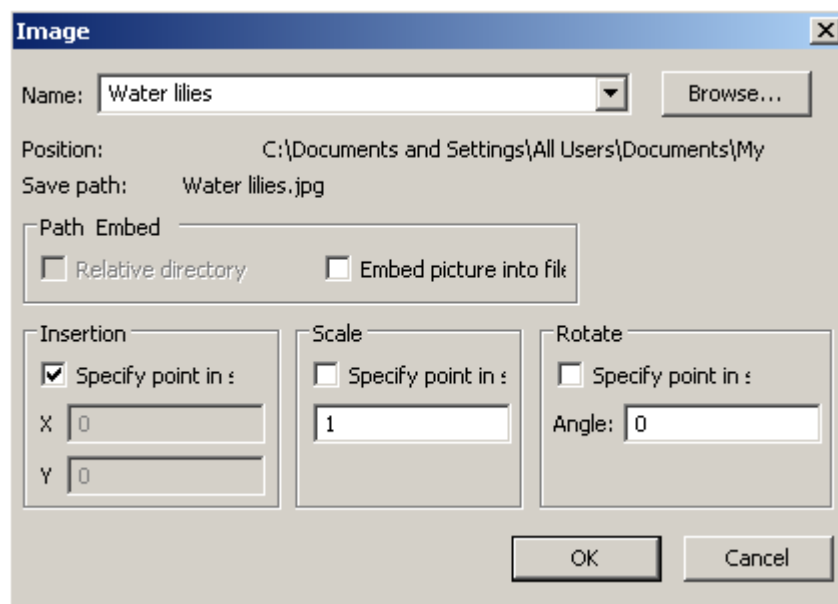
【Icon】 

【Definition】 Select image and insert it to current image as reference.

【Step】

User can implement command insertimage in the forms of : Click command insertimage in the main menu of Drawing, click  button in the tool bar of Drawing, Click  button in the Superior drawing panel from the common option card, execute insertimage command

Implement command insertimage, select file to be inserted, the following immediate menu will pop up.



Pic3-71 insert image

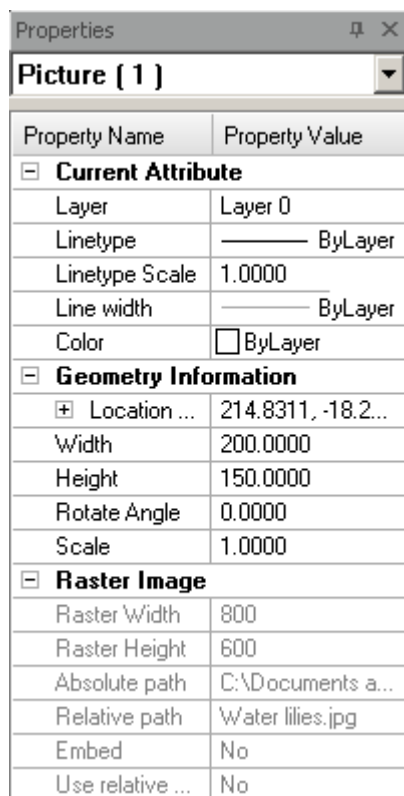
- (1) **【Name】** The selected file name will be displayed, click Browse to re-select file.
- (2) **【Position】** Display path of selected file.
- (3) **【Save path】** Display specified path when the file is attached to current graph. For path of file, in addition to absolute path, user can set and use relative path or insert into current file. But when using relative path, the current draft must be saved first.
- (4) **【Insertion point】** Specify insertion point in the selected image, the default value is “Specify on the screen”, the default insert point coordinate is (0,0).
- (5) **【Scale】** Specify scale modulus, if select “Specify on screen”, then instruct by command or input directly. If “Specify on screen” is not selected, user can input scale modulus. The default scale modulus is 1.
- (6) **【Rotate】** Specify rotate angle of selected image, in “specify on screen” is selected, then rotate by positioning equipment until such dialog box exits, user can also input angle value when it prompts. If the option “Specify on screen” is not selected, input rotate angle directly in the dialog box, the default rotate angle is 0 degree.

3.4.3 Edit image

In CAXA Draft, user can edit the inserted file in several forms, such as feature edit, solid edit, picture management.

3.4.3.1 Character edit

Select image, feature option panel is shown as follows:



Pic3-72 properties of image

User can check property, geometry information of the edited file.

3.4.3.2 Solid edit

Solid edit includes Pinch point edit (zoom and move) , move, zoom, delete, array, mirror, rotate etc. But it doesn't support curve edit, such as trim, chamfer, edge, break ,stretch etc.

3.4.3.3 Image manage

【Command】 image

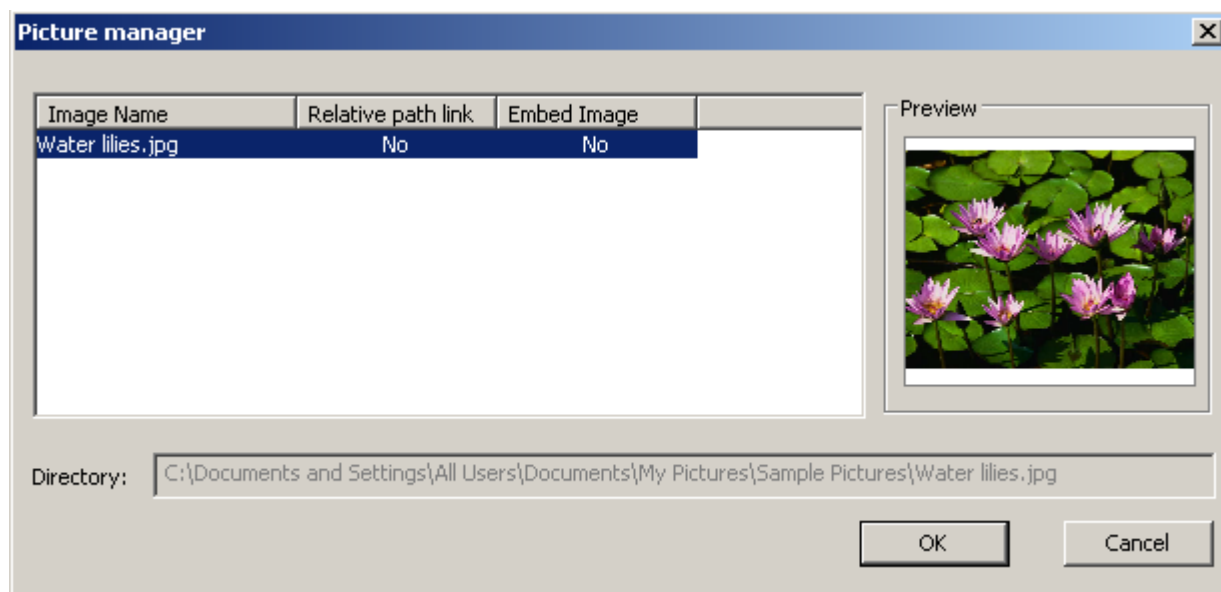
【Icon】

【Definition】 Set parameter via unified graph management such as file save path etc.

【Step】

User can implement command image in the forms of : Click command image in the sub menu of image from the main menu of Drawing, click button in the tool bar of Drawing, Click button in the Insert panel from the common option card, execute ellipse command.

Click command image, the following dialog box will pop up.



Pic3-73 picture manager

Click OK or No under the option of Relative path link and insert image in the above dialog box to modify, but before using relative path, please save current file first.

3.5 Block

3.5.1 Summary

CAXA Draft provides function of combining different kind of graphic element to block. Block is graphic entity with complex form, which is widely used. The feature of block is introduced as follows:

(1) Block is a complex graphic entity, which can be user-defined, once it is produced, original several individual entity can be unified as a entirety. Operations of moving, copy deletion etc. can be done to it.

(2) It can be broken, then element that forms block will be individual again.

(3) A group of images display order can be identified by block.



(4) A group of images association can be cited by block.

(5) Relative non-graphic information can be saved by block, such block name, material etc. All these are name block attribute.

(6) Graph in the block will have different color, linear, or line weight in different layer, the newly created block is always in the current layer, but the information about original layer, color and linear feature of the object is saved in the block reference. User can control keeping original feature or inheriting current layer, color, linear, or line weight setting. The object in the block

(7) Block element includes icon, dimension, text, frame, title bar, specification etc in CAXA Draft, all these elements can be operated by block operation except block production.

Operation of Block mainly includes: create block, property define, insert block, edit block etc.

User can implement command block in the forms of : Click command block in the main menu of Drawing, click  button in the tool bar of Object, Click  button in the base drawing panel from the common option card, execute block command via keyboard.

3.5.2 Creat block

【Command】 block

【Icon】 

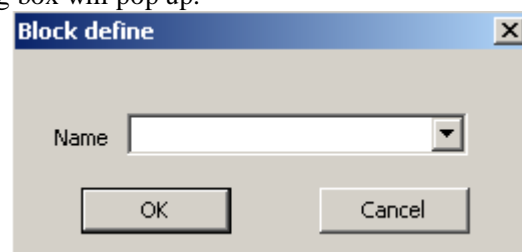
【Definition】 Select a group of image and define them as one object.

Each block includes block name, one or more object, base point coordinate value for block insertion, and relative property data.

【Step】

Click sub menu block or block combination button, or click block command via keyboard to implement.

Then select objected to be combined into block, and click yes to confirm, then specify base point for the block, click right key, the following dialog box will pop up.



Pic3-74 create block

Input name for the block in the name frame, the name can have at most 255 characters, including letter, number, space or other special characters. Block name and block definition are saved in current graph.

【Explode allow】 this check box is used to specify whether it can be exploded or not. 【Unify scale】 , this check box is used to specify whether it will be zoomed by scale or not.

Click button 【Hide】 to set whether hide or not for the newly created block. As for hide, we will introduce it in next section.

Click button **【OK】** , then the block is created. If click **【cancel】** , such operation will be cancelled.

If select object first, then click command block to create block, user can directly specify base point first.

3.5.3 Block hide

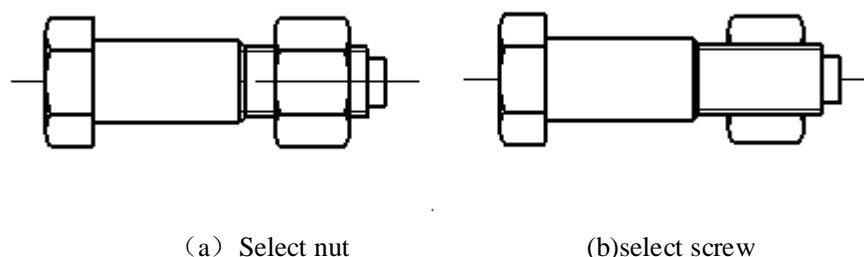
CAXA Draft provides function of 2D auto hide, which brings more convenience to users, the advantage is more outstanding when spare part positions are overlapped in the course of drawing assembly diagram.

The block graphic that has closed outer contour is regarded as front graphic area, other graphic in this area will be removed automatically to realize 2D hide. The hide of existing hide area can be cancelled, and the removed graphic will be resumed and displayed on screen.

When the block is created, user can select whether hide or not in the feature option panel.

Example:

E.g.1 The figure is example of hiding block.

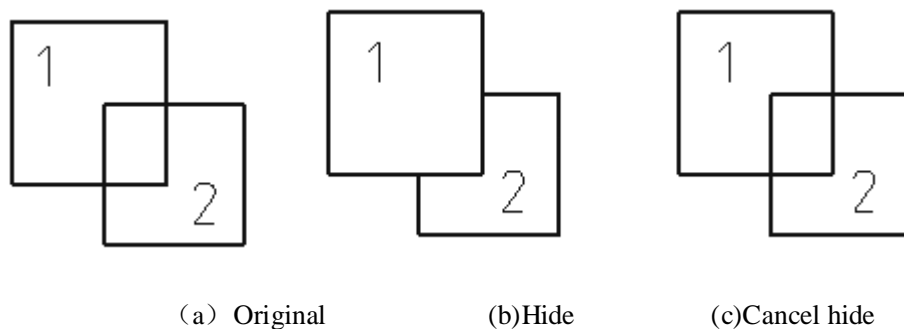


Pic3-75 block hide example

The nut and bolt are defined as two blocks respectively, when they are fitted together, the problem of block hide will occur, select nut in figure (a) as front entity, bolt and other overlapped part will be hidden. When selecting bolt, it will be front entity, and then corresponding part of nut will be hidden, as shown in figure (b).

E.g.2 Figure 4.58 is an example of block hiding and cancel hiding.

In figure (a), there are two rectangles that are defined as two blocks, which are overlapped mutually. When selecting top left one, corresponding part of lower right one will be hidden, as shown in figure (b). Select mode of cancel hide, then select block 1, the hidden part in block 2 will be resumed, as shown in figure (c).



Pic3-76 hide and cancel hide

3.5.4 Attribute definition

【Command】 attrib

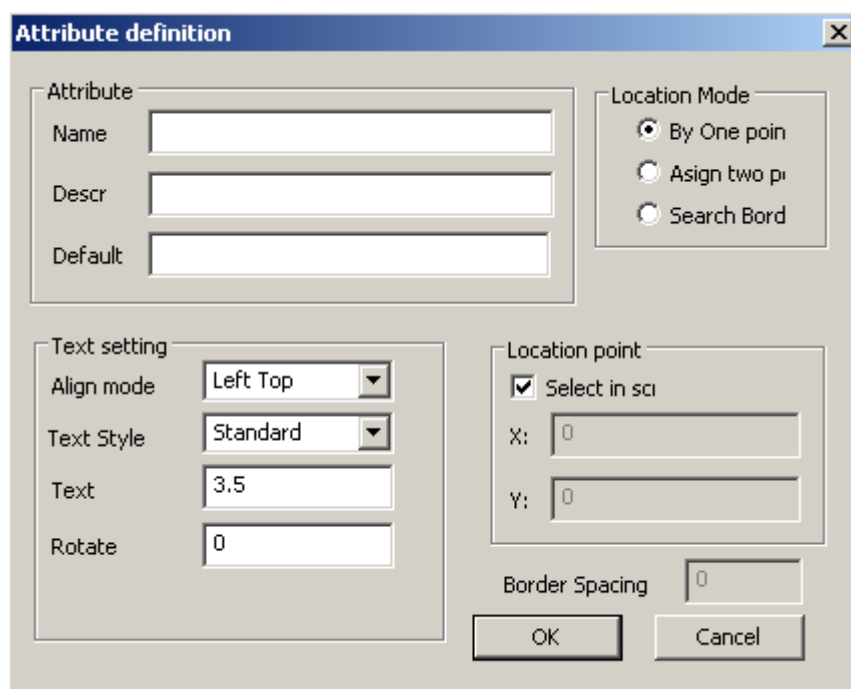
【Icon】

【Definition】 Create a group of attribute definition that is used to save non-image data in the block.

Attribute may include the data of part code, name, material or other information. If the attribute definition is created, user can select is as object when creating block definition. But if attribute definition has already been combined into the block, it will prompt by specified characters“input attribute” when inserting block. Then each later reference of that block can be used as specified different value of the attribute.

【Step】

Click sub menu Block or attribute button , implement command attrib via keyboard, the following dialog box will pop up.



Pic3-77 Attribute definition

Input attribute name in the name frame by any character, then the content will be displayed in the graph by default.

In the frame of **【describe】**, input data, actually it is used to prompt when inserting block including such attribute definition. User can choose to fill or not fill, if it is blank, the attribute name will be as prompt.

In the frame of **【Default value】**, input data to specify default attribute value.

【Anchor point】 is used to specify attribute position, user can input coordinates of X axis and Y axis, or select the check box **【screen selction】**

【Text setting】 is used to specify attribute text alignment, text style, text height, rotate angle.

Click OK, the attribute definition is finished, click cancel to cancel such operation.

3.5.5 Insert block

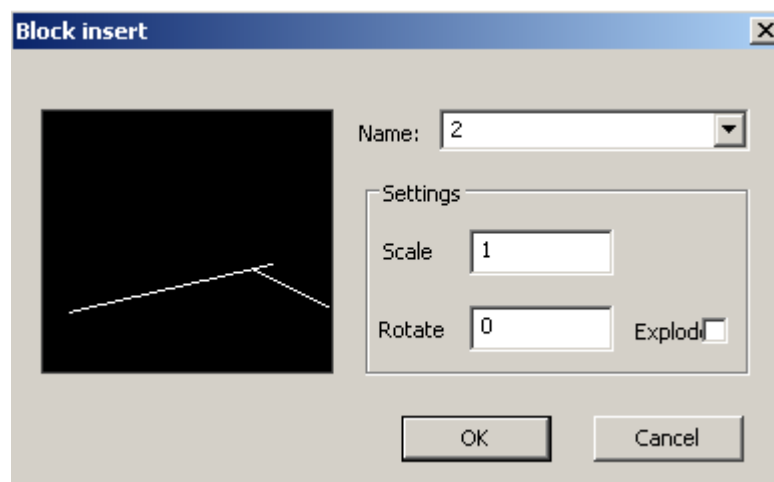
【Command】 insertblock

【Icon】 

【Definition】 select one block and insert it into current image.

【Step】

Click sub menu insertblock, implement insertblock command via keyboard. The following dialog box will pop up.:



Pic3-78 insert block

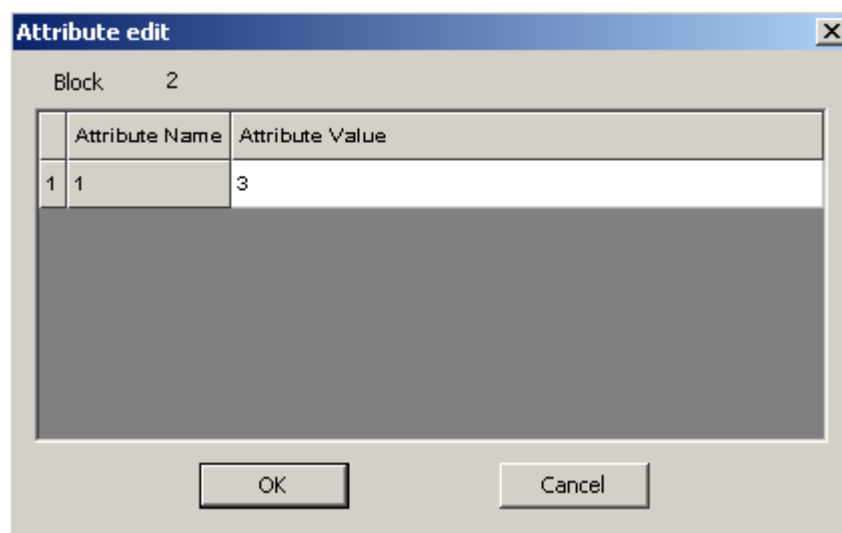
In the frame of **【Name】**, input data or click the block to be inserted.

Specify zoom scale along X axis direction and Y axis direction for the block to be inserted, select Unify scale to specify unified scale value for X axis and Y axis.

Preview of the block to be inserted is displayed on the right side. **【Rotate angle】** is used to specify the rotate angle for the block to be inserted in current image.

Click OK, the block will be inserted, click cancel to cancel such operation.

If the inserted block included attribute, following dialog box will pop up when inserting.



Pic3-79 attribute edit

Double click **【Attribute value】**, the under cell can be edited, when the block is inserted, double-click block, the above dialog box will pop up for attribute edition.

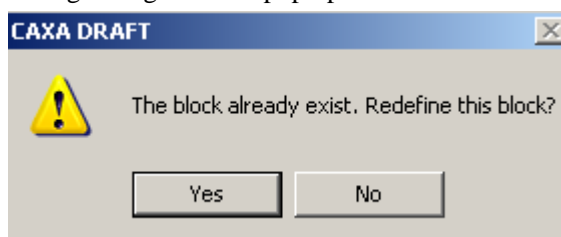
3.5.6 Edit block

User can edit the inserted block in the current image, such as object in the block, color, linear, block attribute data and definition etc.

When several blocks with the same file name are inserted into current image, all modification of the block ,except attribute definition, will affect the cited block with the same name in the current image.

3.5.6.1 Modify block definition

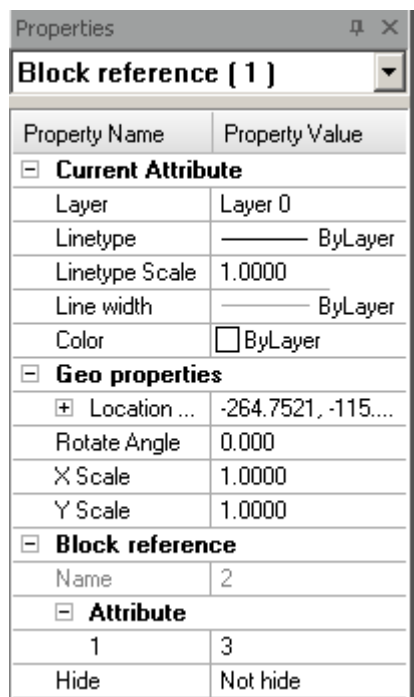
In the current image, if the block is already defined, when creating new block and the newly inputted name is the same as the existing one, the following dialog box will pop up.



Pic3-80 dialog for exist block

Click yes to overlay the existing block definition, the cited block in current image will all be updated, if click no, user should define the block again.

Check and modify block definition in the Feature option panel, select a block and open feature option panel, the following dialog box will pop up.



Pic3-81 properties of block

User can modify block layer, linear, line weight, color, anchor point, rotate angle, zoom scale, attribute definition, selection of hide etc.

3.5.6.2 Modify block color and linear.

【Definition】

Base feature of image includes layer, linear, line weight. When modifying block base feature, the displayed feature of the object in the block can change along with block, or keep its original feature.

(1) Object feature in the block will keep the same if the object color, linear, line weight are not inherited from current setting.

So user should set color, linear, line weight for each object in the block definition, instead of using BYBLOCK or BYLAYER as setting of color, linear, line weight when creating such object.

(2) The feature displayed in the block inherit feature specified to block. When the block layer, color ,linear, line weight is modified, the object feature in the block will change together.

For this selection, set current color or linear as “BYBLOCK” when creating object included in block definition.

【Step】 User can modify object feature in the block via feature option panel, as shown in figure 3-85.

3.5.6.3 Modify block attribute

Block attribute modification mainly includes: attribute data edit, attribute definition edit.

The method for attribute data edit is: double click block to be modified, a dialog box will pop up, as shown in figure 3-83, or use feature option panel to modify.

The method for attribute definition edit is: Use block editor or reigning edit block to enter into edit status, then

double-click attribute definition. Or modify it via feature option panel, click save to save the modification. The modification to block attribute definition is of no effect to the already inserted block. But when re-inserting the block with the same file name, it block attribute definition will use the modified one.

3.5.6.4 Block edit

【Command】 bedit

【Icon】 

【Definition】 Edit block.

【Step】

Click sub menu Block or implement bedit command via keyboard. In addition to other edition, there are some special function for block edition, such as attribute definition etc. When the function area is open, such function is in the added “block edit” function panel. When it is close, those function will be on the tool bar of block edit.

Once the modification is finished, click exit, it will prompt “whether modify”, click yes to save the modification, click no to cancel the modification.

3.5.6.5 Block reigning edit

【Command】 refedit

【Icon】 

【Definition】 Reigning edit block.

The difference between Reigning edit and block edit is: when marking, measuring for reigning edit, user can refer to other object in current image. But for block edit, only the object in the block will be displayed.

【Step】

Click sub menu Block or implement refedit command via keyboard. Select block then click right key to select Refedit.

Implement Refedit, select block to be edited, in addition to other edit operation, there are some special function for refedit, such as add into block, move out of block, save and exit, exit with save. When the function area is open, such function is in the added “refedit” function panel. When it is close, those function will be on the tool bar of refedit.

Add to block: Select other object from current image, and add it to the block definition that is being edited.

Move out of block: Move the object being edited out of block to current image.

Save and exit: Save the edition of block definition and exit refedit.

Exit without save: Cancel the block definition edition.

Chapter 4 Library

4.1 Summary

In this chapter, Graph library will be concretely introduced , which is a significant chapter in CAXADraft.

Basic unit of Graphics Library is called symbol. Symbol is object composed of some basic image, as well as parameter, property, dimension or other special attribute. A group of image object can be quickly created as per desired parameter by extracting symbol, so it will reduce later operation.

symbol is divided into parameter symbol and fixed symbol according to the parameter , which is combined by one view or several views (less than 6 views). And each extracted view can be defined as block, the block can be hidden when being used. It is much more convenient to draw spare parts, installation diagram and other engineer chart by Graphics Library and block operation.

Feature of CAXA Draft library:

(1) Rich symbol

There are altogether thirty thousand symbols in Graph library, the symbol are classified into hundreds of sorts, including standard part, electric element, engineer symbol etc. By using such symbol, user can speed up drawing for all kinds of industry.

(2) Standard symbol

Base symbols in the library of CAXA Draft are all made as per national standard, in order to ensure the created symbol meet standard requirement.

(3) Open

CAXA Draft library is completely open, except attached symbol when the software is installed, user can define symbol according to his request.


(4) Parametric

The symbol is completely parametric, user can define dimension, property or other parameter, it is convenient create and manage symbol.

(5) Directory structure

By using directory structure save, user can move, copy ,share symbol conveniently.

We will introduce ,in this chapter ,symbol extract, symbol drive, symbol management, symbol conversion in forms of :

User can implement command sym in forms of : click corresponding button in the sub-menu of library from main menu Drawing, click button in the tool bar of library, click the button of  in the commonly-used Basic drawing panel from the option card , implement related command or short cut key. User can dragging via library option panel to extract symbol., execute via library option panel.




4.2 Pick Symbol

【Command】 sym

【Icon】 

【Definition】 Configure parameter for suitable symbol and it, then pick it out of graph library and add it to current image.

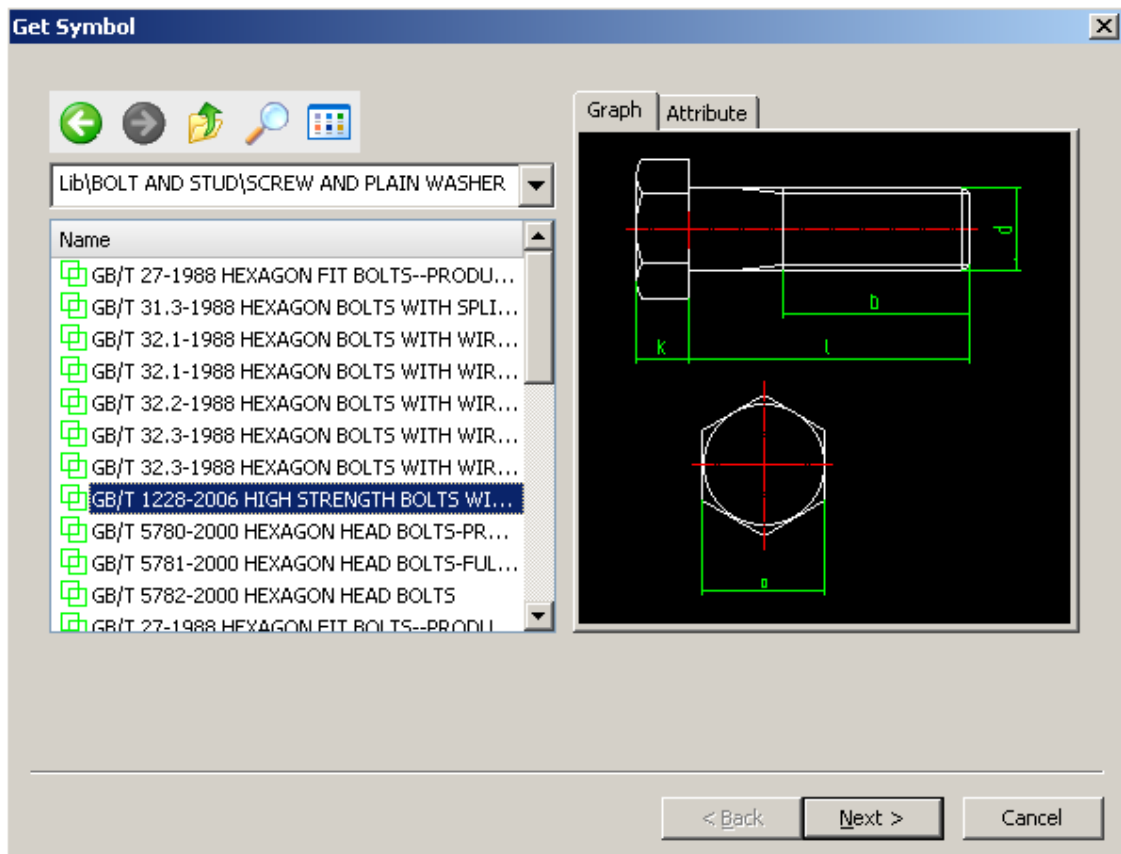
【Step】

User can implement command sym in forms of : click  button in the sub-menu of library from main menu Drawing, click  in the tool bar of library, click the button of  in the commonly-used Basic drawing panel from the option card , implement command of sym, execute via library option panel.

The cause for picking parametric symbol and non-parametric symbol is different, it will be introduced as follows.

4.2.1 Pick Parametrization Symbol

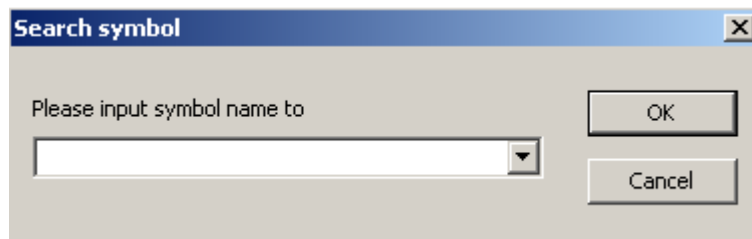
Implement command Symbol, following dialog box will pop up.



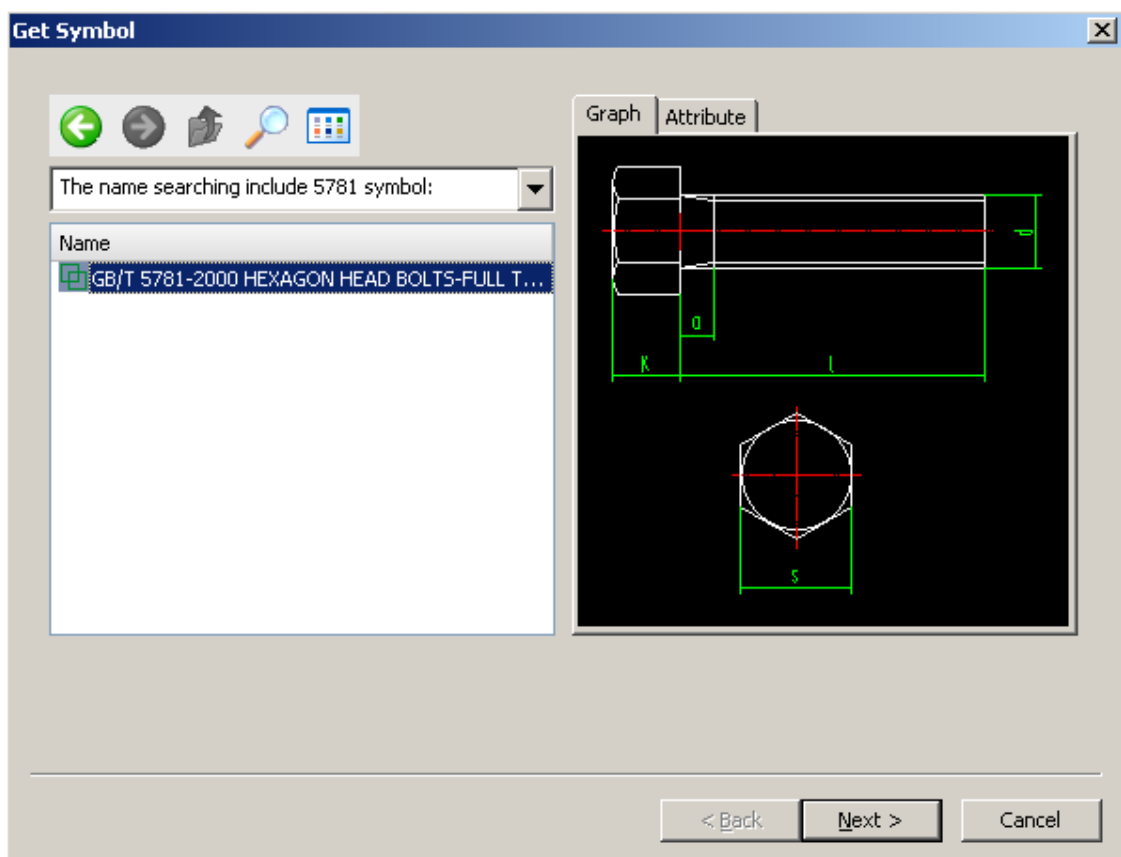
Pic8-2 Pick Symbol

Thousands of symbol are classified into several sorts , so that user can find desired one immediately, as shown in above figure, select symbol at the left side, the selected one will be display in the preview frame at the right side. Press button and control in the above dialog box to proceed quick searching. Step of searching is shown as follows:

- (1) Operation of symbol search is the same as that of Window explorer, the lower side is folder, symbol structure tree is at the upper side. User can convert in different directory structure by 2 controls.
- (2) 、 and button means respectively back, forward, up, user can convert between different directory by these buttons.
- (3) this button is used to convert browse mode, click it to convert from list mode to breviary mode.
- (4) Click , following dialog box will pop up.

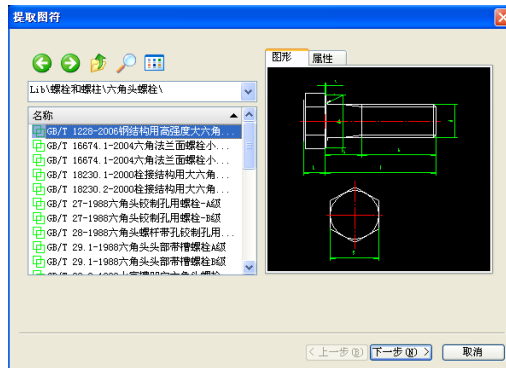


Input a part of symbol name in this frame, the system will search corresponding symbol automatically, for example, GB5781-86 HEX SCREW WITH COMPLETE THREAD GRADE C , just input GB5781-86 or HEX SCREW WITH COMPLETE THREAD to search its symbol. In addition, Dim search function is added, input the object's name or type in the search frame, all symbols related to the input content are listed.

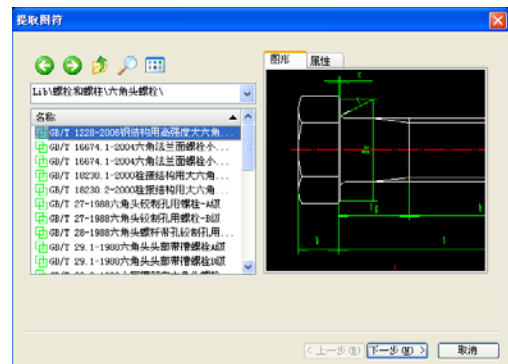


(5) The right part of above box is preview frame, including label attribute and figure, by which user can preview figure and attribute of the selected current symbol. Click label Attribute and switch to attribute preview mode. The basic point of each view is marked with highlighted +. Press right key to zoom in the symbol, as shown in following figure. Press right key and left key

simultaneously to renew it.

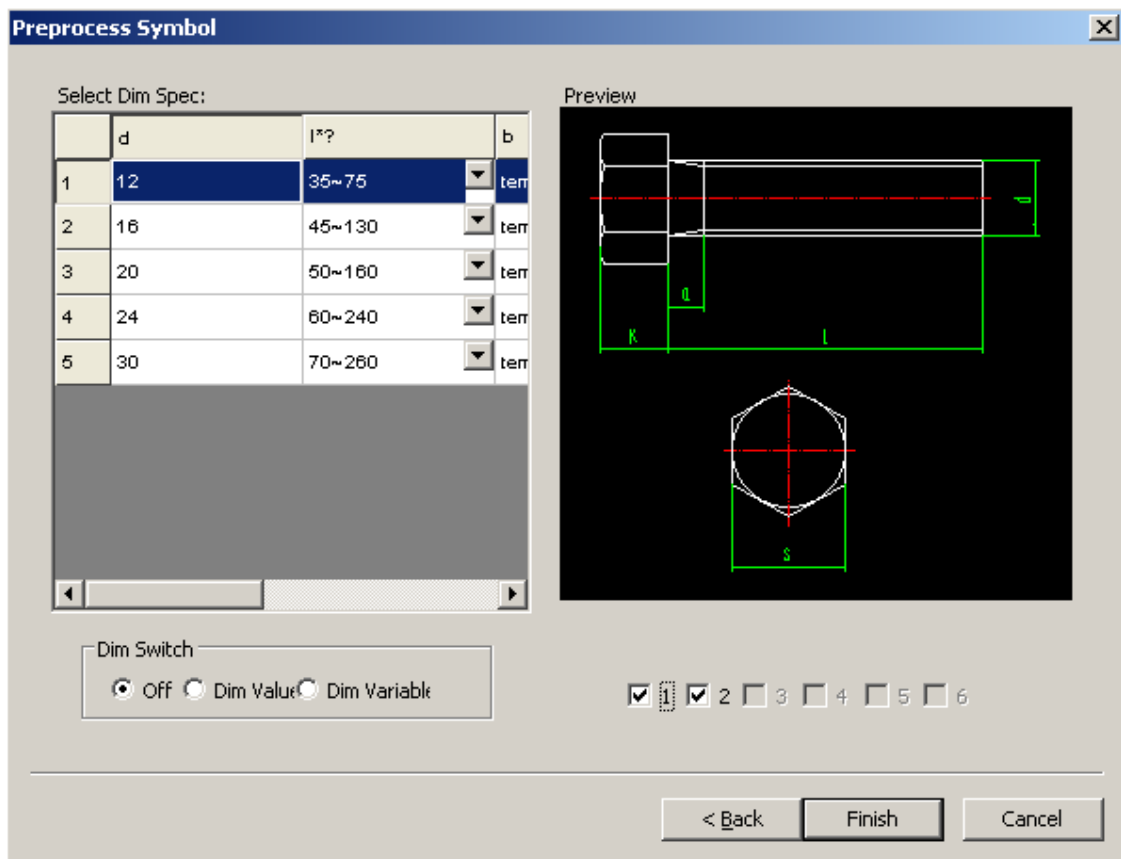


Before zoom out



after zoom out

(6) When the symbol is selected, press button next to get symbol pretreatment dialog box, as shown in following figure:



Pic8-4 Preprocess Symbol

The right part of upper figure is symbol preview area, there are 6 view controlling switches listed in the lower side. Press left key to open or close any view, the closed view can't be extracted. Not all symbols have 6 views. Generally, two or three views are enough.

The left part of upper figure is symbol treatment area, item one is size selection by electronic table form. Size variable name is in table head, each size variable name and meanings can be seen ocularly at the right side preview area. If the figure display is too small, click right key in preview area, this point will be the center for zoom in. Press right key and left key simultaneously to renew it. Edit any cell in table by mouse or keyboard. Press F2 for direct editing.

Note: The size variable name has mark "*", that means it's serial variable, each cell in the corresponding rank has a range, such as "10~40", select one concrete value. The operation method is: click left key at corresponding cell, a pulldown button at the cell right side will pop up, press it and all serial value of current range will be listed, click the desired value by left key, the selected value will be displayed in the original cell. If there is no needed value, input a new value in the cell directly.

If the variable name has mark "?", that means it can be set as dynamic variable, which indicates the size value is unlimited. When one variable is set as dynamic variable, it will not be restricted by give data. when extract The variable size can be changed arbitrarily by inputting new value or dragging mouse. Then select corresponding cell by clicking right key, there will be a mark "?" after that value.

When data is input, confirm other parameter, detail is shown as follows:

Dimension switch is used for controlling measure dimension after graphics is extracted. User can click left key, Off means the extracted symbol has no mark dimension. Dimension means the extracted symbol has actual dimension. Dimension variable means measuring dimension variable name instead of actual dimension.

Symbol treatment controls symbol output form, under default condition, each view of symbol is a block insert. Explodemeans exploding block, make each view be mutual

independent element. Hide means allowing the symbol to be hidden after it is extracted, refer to relative section of chapter "[Drawing](#)" for concrete information. Original means the symbol keeps the same form after it is extracted.

If the selected symbol is not the desired one, click back to extract other symbol. If all set is finished, click button ok and the system will return to drawing state again, and the symbol is hanging on the + cursor.

(7) Select symbol locating point via mouse or keyboard as per prompt, once specified, the symbol will rotate instead of moving. Input rotate angle as per instruction, or accept the default angle 0 degree (no rotate) and press right key to confirm. When the symbol is rotated at appropriate place along with the moving mouse, press left key to confirm.

If dynamic definition size is set, and this size is included in current view, “drag and define x value” will be displayed on the status bar when the view rotate angle is specified, among which X indicates size name, the size will be changed along with the moving mouse. Drag the mouse to appropriate position to get the final size. symbol can have several dynamic sizes.

Then an symbol view is extracted, if the symbol has several views, the open views will be hung on + cursor automatically. When all open views of an symbol are extracted, the system will begin to extract again. If no view is needed, press right key to terminate such operation.

4.2.2 Pick fixed symbol

In Graphics Library of CAXA Draft, most symbols are parameter symbol, only some of them are fixed symbol, such as electrical elements and hydraulic pneumatic symbols. Extracting fixed symbol is more simpler.

Implement command of picking fixed symbol, select desired symbol, click button of next, the following immediate menu will pop up.



Pic8-5 immediate menu for picking fixed symbol

Click **【1.】** in the immediate menu, select explode or non explode for the symbol to be created.

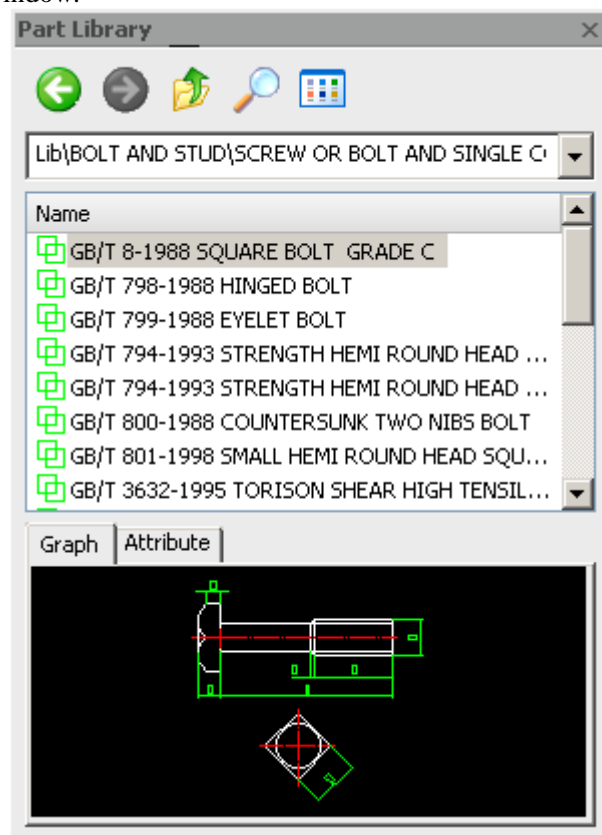
Click **【2.】** , select hide or non-hide for the symbol.

Then Click **【3.】** and **【4.】** in the above immediate menu, set zoom multiple for horizontal zoom and vertical zoom , the default enlarge multiple is 1. Or user can click corresponding immediate menu and input proper zoom multiple in the editing box.

When the above parameter is confirmed, select locating point and input rotate angle, then the symbol is extracted.

4.2.3 Pick symbol from library option panel

In CAXA Draft, user can use library option panel to extract symbol. Open library option panel, select desired symbol to extract, the method is introduced in 4.2.1, press left key while dragging it to the right window.






4.3 Drive symbol

【Command】 symdrv

【Icon】 

【Definition】 Drive the extracted unbroken symbol , i.e. change its specification and measure dimension or output form or other parameter.

【Step】

User can implement command symdrv in forms of : click  button in the sub-menu of library from main menu Drawing, click  in the tool bar of library, click the button of  in the commonly-used basic drawing panel from the option card, implement command of symdrv.

Click command symdrv, all non-exploded symbol in current drawing will be highlighted. Select symbol to be changed by left key.

symbol pretreatment dialog will pop up, in which user can modify symbol specification, switch and symbol treatment etc if necessary .

Click button OK after modification. The modified symbol is shown in the drawing area instead of the original one, but with the same locating point and rotate angle.

4.4 Symbol definition

【Command】 symdef




【Icon】 

【Definition】 It is the process for setting up user's own Graphics Library as per actual requirement.

At different occasions or different technological backgrounds, user may need some special figure or symbol. Commonly used symbol can be defined by the command of symbol definition, and enlarge existing library.

Symbol is divided into fixed symbol and parametric symbol, the definition method is different for both of them.

【Step】

User can implement command symdef in the forms of : Click  in the sub menu of Library from main menu of Drawing, click  button in the tool bar of Library, Click  button in the basic drawing panel from the common option card, execute symdef. command.

4.4.1 Fixed symbol definition

【Definition】 Create fixed symbol with parametric.

Some commonly used graph ,which doesn't need parametric drive, can be regarded as fixed symbol in the library for future use.

First, draw a graph needs to define as per actual dimension scale in the drawing area, select dimension or non-dimension as per request.

【Step】

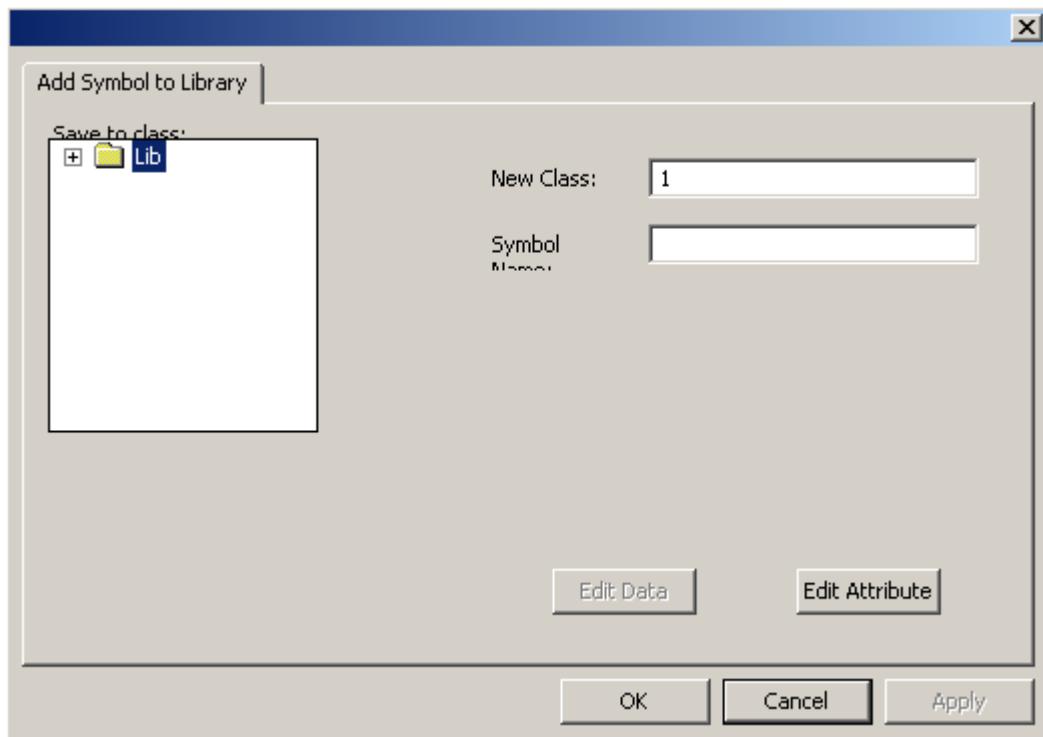
When the image is finished, click command symdef, Select all element of the first view and click right key, click right key to confirm after selection.

The system will hint "specify view basic point", click left key to specify or input via keyboard , it will be better to select key point or special position point, center , end point etc as basic point.

If the select object includes dimension, it will prompt “specify one variable for each dimension, since it is defining fixed symbol, click right key directly, it will prompt “ there is still innominated dimension, click next to continue?” , dimension nominating will be cancelled if click yes .

When all element and basic point of the first view is specified, defined next ones as per step 4 and 5.

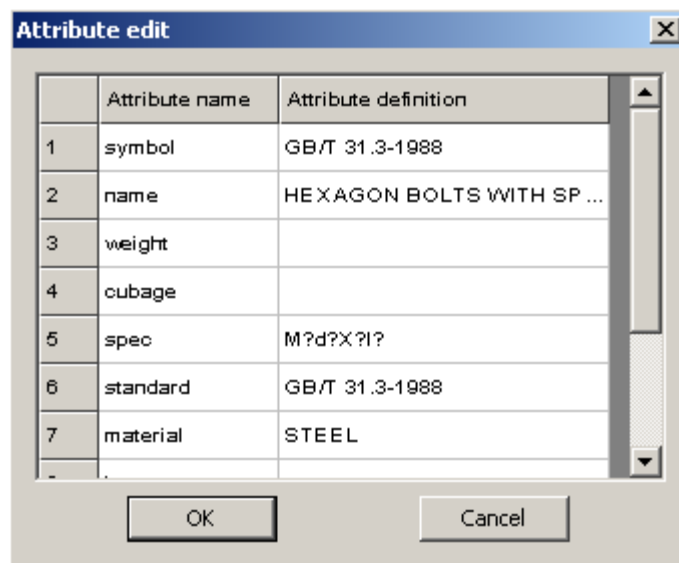
After the element and basic point of the last view is input, put in Graphics Library dialog box pop up as shown in the following figure, since it's fixed symbol, back and data input button are invalid.



Pic8-7 Add symbol to library

Select position for the to be created sort on the left side, input a new soft name in the frame of New sort, then input symbol name .

Click button **Define Attributea** , attribute input and edit dialog box will pop up, electronic board has provided 10 default for attribute, new attribute can be added, or deleted the existing attribute if necessary. When input focus in the table, and press key F2, the current cell is in edit state, and the insert symbol is behind the text in the cell. If new attribute needs to be added, input in the row with mark * , which stays at select area of the left most table. Position any row via cursor, press button insert , a blank row will be inserted above that row, add new attribute here. Click the selected area in the leftmost row, and choose this row , press button delete to delete the row.



Pic8-8 Attribute edit

When all option are filled in, click button ok to confirm. Then the new symbol is added to Graphics Library.

The operation for fixed symbol definition is finished now, when get symbol, the new symbol can be found in corresponding type.

4.4.2 Define parametrization symbol

【Definition】 Create parametric symbol, of which the dimension can be driven.

If the symbol is defined as parameter symbol, the symbol dimension can be controlled when being extracted. It's more flexible to use, but the operation is more complicated than that of fixed symbol.

Draw a figure to be defined as per actual dimension scale in the drawing area, and make necessary measure dimension .

When defining parametric symbol, pay attention to following items to prepare image:

(1) Hatching, block , character or fill of symbol defined by locating point.

Since hatching treatment is proceeded by a locating point to search closed loop it exists. But using command hatching in electronic board can draw several section via several locating points in one time. Owing to this, drawing hatching

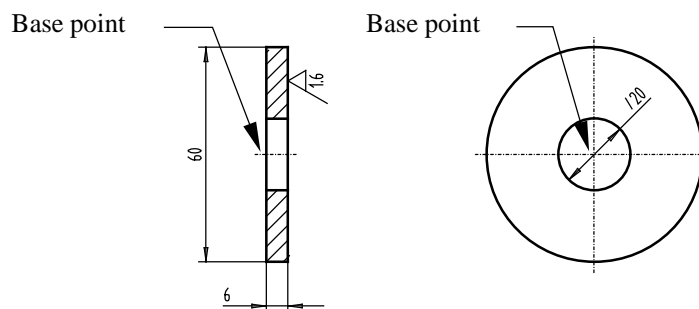
in the course of symbol drawing must use "hatching" command independently to each closed section.

(2) If it doesn't affect define or extraction, try to make less measure dimension when drawing, in order to reduce data input. E.g. fixed dimension is unnecessary to mark, dimensions of two related parts needs to mark one, small diameter of thread is commonly 0.85 times of big diameter thread, measure big diameter is enough. To unimportant chamfer or root radius, if the changing range in whole standard data group is not large, draw the same dimension and define it as fixed value, on the contrary, include appropriate proportion with a measured dimension, and defined it as the form $0.2 * L$, so it doesn't need to mark.

(3) Dimension line should be led out from feature point when measuring, draw a point as a dimension outgoing point, or break corresponding element for dimension place if necessary. It will help to locate and sorption measure.

(4) Try to draw symbol accurately for strong association when defining element, and avoid dimension sorption mistake. Extract a standard data group for drawing dimension, the graphics proportion will be more clearer, and auto-sorption can be proceeded correctly.

For example, define a washer as shown in the next figure, when drawing is finished , click library operation in submenu of Draw, press button define symbol in the sub-menu.



Pic8-9 Draw

(1) Specify view

Select all element of the first view as per instruction with some relating dimension, and press right key to confirm.

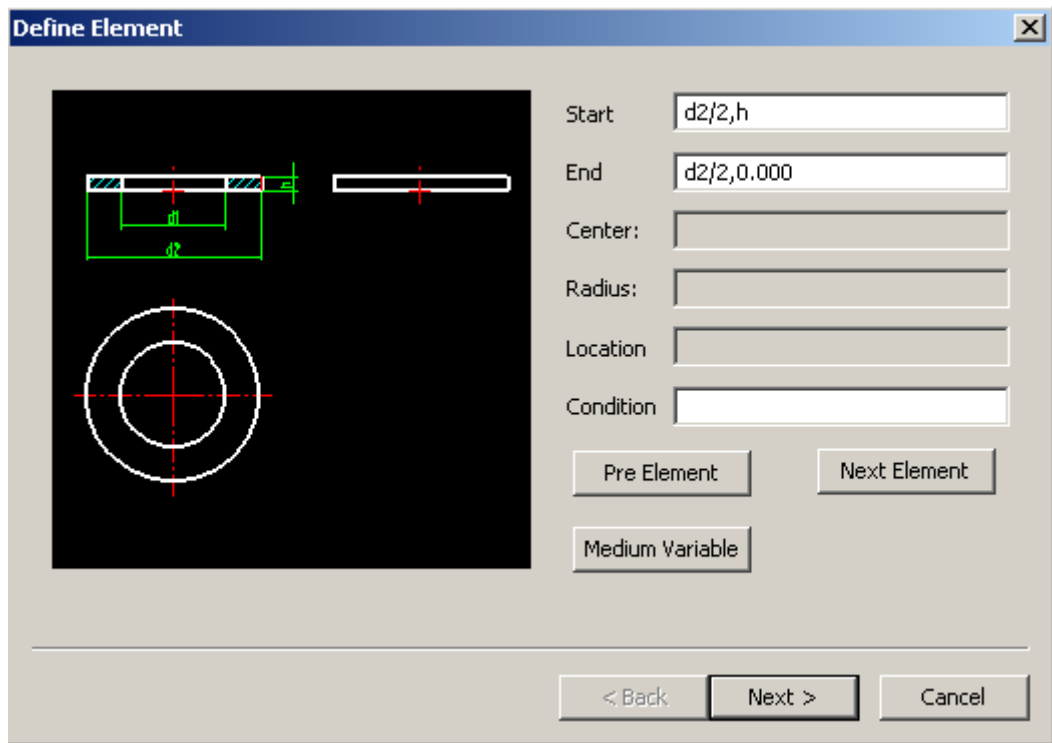
Then it hints "specify view basic point", press left key to specify or input via keyboard. Basic point is reference point for symbol extraction positioning, the latter element definition regards basic point as reference, try to select reference point at key position or special position, such as center, end point etc. Make full use of tool point, intellectual point, navigating point and grid point to specify the basic point. If the basic point selection is not correct, element definition express will be more complicated, and insert positioning of symbol extraction will be more inconvenient.

Set a variable for each dimension as per instruction, select every dimension by clicking left key, the dimension will be highlighted when it is selected. Name each dimension in editing box, which should be consistent with common and standard ones, press enter key to confirm variable name, then the dimension color will be renewed. User can continue to select other dimension, or re-select dimension with variable name. Press right key to confirm when all dimension names of the view are inputted.

Then specify element, base point, dimension variable of view two, view three ,view four etc. The method is the same as that of view one.

(2) Element define

When all views are defined, following dialog box of define element will pop up.



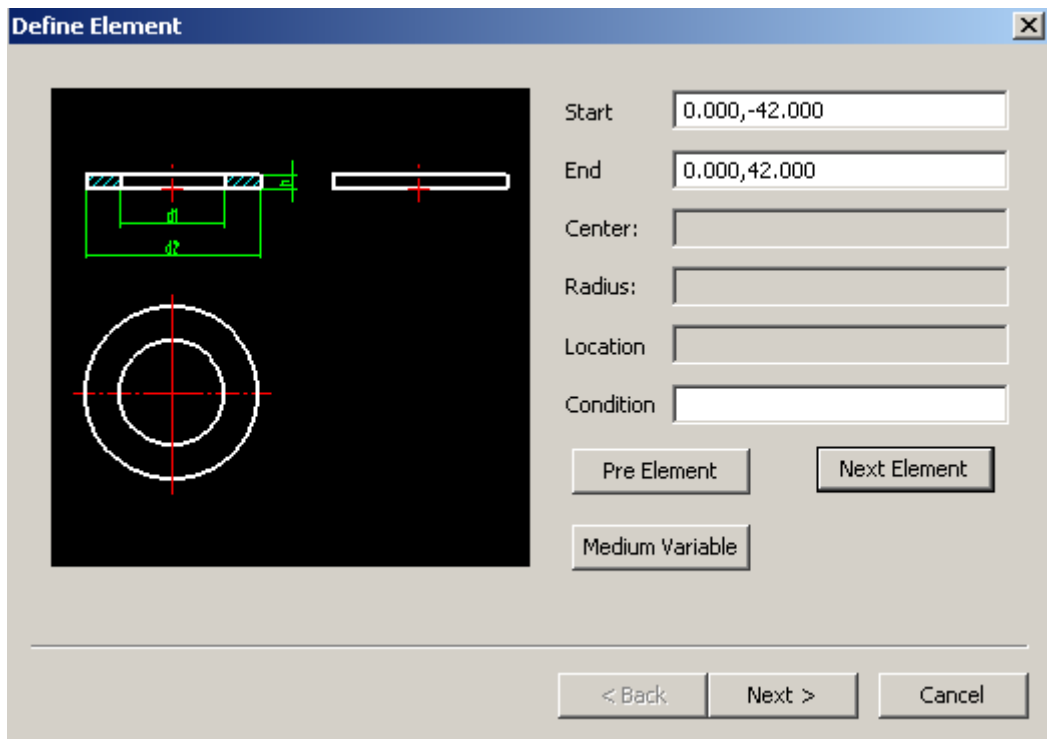
Pic8-10 define symbol

Element definition means symbol parameterization, use dimension variable one by one to show every element expression, eg. start point, end point of line, circle centre and radius expression, etc. Make coordinate expression relating to basic point for each element definition point. The expression will affect symbol extraction. Press button last element and next element can inquire and modify definition expression of each element., select in the preview area by clicking left key. If the figure in preview area is complicated, click symbol preview area by right key, the graphics will be enlarged by scale for watching and selection. Press right key and left key simultaneously, the graphics size in preview area will be renewed. If the graphics needs modifying, click back to return to the previous operation.

CAXA Draft can produce some simple element definition expression automatically. When element definition is being done, it can modify and improve undefined element expression as per the defined element expression. It should be noted f:

- Define center line:

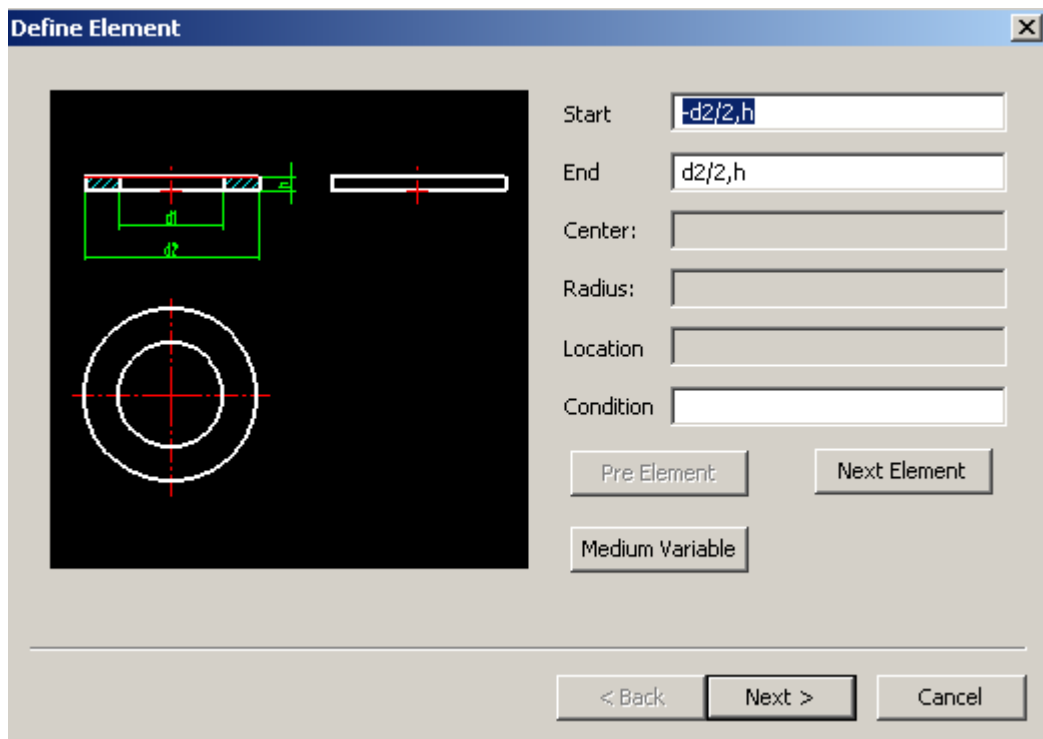
When define centre line, not all the definition expressions are tallied with actual coordinates in drawing, define it as per 2-5 drawing units of the exceeded contour, as shown in the upper figure. Define the start point and end point of centre line in front view, refer to the upper figure to select the basic point, refer to following figure.



Pic8-11 define center line

● Defining hatching and filling locating point

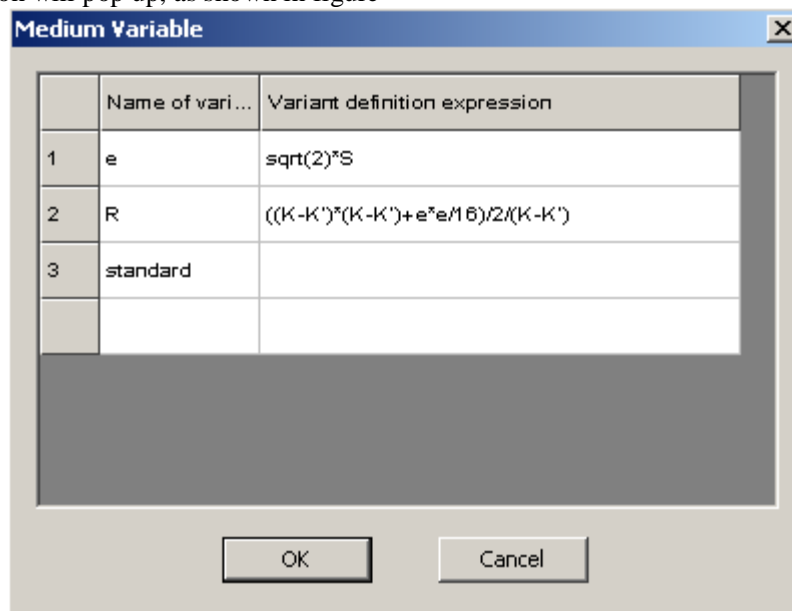
select a point that always keeps in closed loop in spite of different size, in order to assure correct hatching and filling for different dimension. This point is very important. As shown in the following figure, the locating point of front view upper part is defined, this can assure the locating point will be kept in closed loop under any condition.



Pic8-12 Defining hatching and filling locating point

● Intermediate variable:

There is a button intermediate variable in above box, click it and a dialog box of intermediate variable definition will pop up, as shown in figure



Pic8-13 define medium variable

A long expression or regularly used expression can be expressed by a variable, which can simplify the expression and improve computing efficiency when extracting symbol.

Intermediate variable is function of dimension variable and the defined intermediate variable, the former defined intermediate variable can appear in the later defined intermediate variable expression. Once intermediate variable is defined, it can be used in element definition expression like other dimension variable. In the dialog box of Intermediate variable definition, input intermediate variable name in the left part of the box, and input expression in its right part. Click button ok, this variable can be used when setting up library. For example, in the upper part of washer, the hatching locating point Y coordinate is y, coordinate for the hatching locating point Y in lower part can be written as -y.

Intermediate variable can also define individual intermediate variable. E.g. when installing some mechanical parts like washer with other parts, they are chosen as per nominal value (such as nominal diameter), that nominal value is not measured on parts drawing. Another example, many flanges have bolt hole, the flange diameter decides bolt hole number. If record the bolt hole number, it will help user know the desired bolt number when extracting flange. But the bolt hole number is not the dimension in drawing. Use individual intermediate variable under such condition. It's very simple to define independent intermediate variable. E.g. define nominal diameter D0 of washer, input D0 in variable cell of intermediate variable definition dialog box, and it doesn't need to input anything into corresponding variable definition expression cell. D0 is shown in variable list when defining next variable attribute. Input corresponding data when standard data is input.

- Condition:

The condition decides whether corresponding figure element appear in the extracted symbol. E.g. GB31.1 hex screw with hole bolt grade A and grade B, when the thread diameter d is M6 or more, there is a hole in the screw. But when the thread diameter is M3,M4,M5, there is no hole. When defining corresponding circle of the hole, input $d > 5$ in the editing box, circle can appear under this condition. Electronic board will decide whether or not including such element as per specified dimension for extracting symbol. As for other graphics element, just fill nothing in the editing box of condition. In addition to logic expression, Electronic board regards expression more than 0 as true, equal to or less than 0 as false. Condition for figure element that never occurs can be

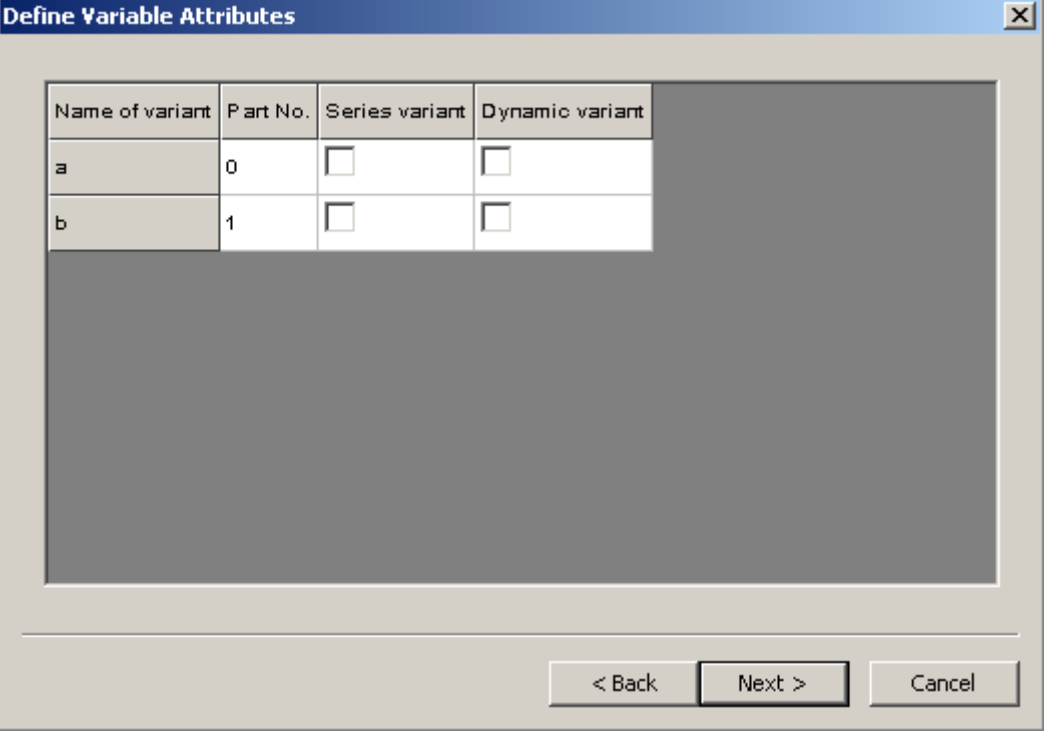
defined as -1, the element that always occurs can be defined as 1 or no condition. Condition can be combined by two expressions, e.g. meet the requirement $d > 5$ and $d < 36$, input $d > 5 \& d < 36$ in editing box indicating operational symbol and, meet requirement $d < 5$ or $d > 36$, input $d < 5 | d > 36$ indicating operational symbol or, symbol | is the same as C language, means operation or, press Shift key and \ key simultaneously can get |.

● Mathematical function:

Mathematical function is often used in defining figure element and intermediated variable. The function format is the same as that of C language. All function parameter should be placed in brackets, the parameter itself is expression. They are altogether 17 functions, sin, cos, tan, asin, acos, atan, sinh, cosh, tanh, sqrt, fabs, ceil, floor, exp, log, log10, sign. The parameter unit for trigonometric function sin, cos, tan is angle, e.g, $\sin(30) = 0.5$, $\cos(45) = 0.707$, $\tan(45) = 1$. The result unit for anti trigonometric function asin are hyperbolic function. sqrt(x) means square root of x. e.g, $\sqrt{25} = 5$. fabs(x) means modulus of x, $\text{fabs}(-36) = 36$. ceil(x) means minimum integer is equal to or more than x, e.g, $\text{ceil}(5.4) = 6$. floor(x) means maximum integer is equal to or less than x, e.g, $\text{floor}(3.7) = 3$. exp(x) means e to xth power. log(x) means ln x (natural logarithm), log10(x) means denary logarithm. sign(x) means when x is more than 0, return to x, when x is equal to or less than 0, return to 0., "%" means remainder operation symbol, "*" means multiply, "/" means divide. Use arc brackets to indicate expression, instead of square brackets or brace. Operation priority is embodied by nesting arc brackets. Following expression is legal expression: $1.5 * h * \sin(30) - 2 * d^2 / \sqrt{\text{fabs}(3 * t^2 - x * u * \cos(2 * \alpha))}$.

(3) Variable attribute definition

Click next when the element definition is completed, variable attribute definition dialog box will pop up, as shown in the figure below.



The dialog box titled "Define Variable Attributes" contains a table with the following data:

Name of variant	Part No.	Series variant	Dynamic variant
a	0	<input type="checkbox"/>	<input type="checkbox"/>
b	1	<input type="checkbox"/>	<input type="checkbox"/>

At the bottom of the dialog box are three buttons: "< Back", "Next >", and "Cancel".

Pic8-14 define variable attribute

Define variable attribute through this box: serial variable, dynamic variable. The default variable attribute is no, i.e., the variable is not serial variable or dynamic variable. Click corresponding cell via left key, the character color in this cell will be blue, press space bar to switch yes and no, or input y or n via keyboard to switch directly. The variable serial number begins with 0, which decides every variable's marshalling sequence when inputting standard data and selecting dimension specification. Generally, 0 is specified as dimension variable serial number, and the variable is the main basis for selecting dimension specification, the default serial number is specified in the list, which can be modified.

(4) Put symbol into library

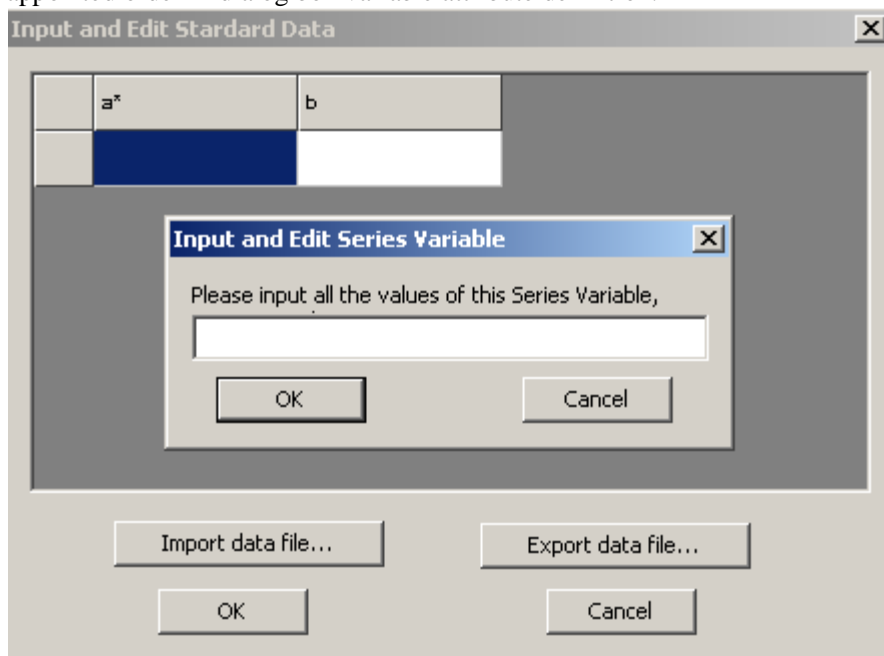
Execute command Variable attribute definition, then click Next, corresponding dialog box will pop up.

Select one corresponding type for new symbol in the symbol box of big type and small type, or input a new type, then input the newly created symbol name in symbol name box.

Click button attribute edit, a dialog box will pop up, input symbol attribute in the box, which can be previewed when extracting symbol, what's more, the unbroken symbol after extraction have records of attribute information for enquiry.

Click button data edit, following dialog box will pop up, The dimension variable is arranged

as per the appointed order in dialog box variable attribute definition.



Pic8-15 input data

1. When focus is input in the table, press function key F2, the current cell is in edit state, and the insert symbol is behind the text in the cell.

If add a group of new data, input in the row with mark "*" directly , which stays at select area of the left most table.

Input serial measure dimension of data in any row, dimension upper limit and lower limit is separated by character except number, decimal point and letter E, for example, 8~40, 16/80, (25,100) etc, try to keep unified .

There is a mark * behind serial variable name in title line, click the cell with serial variable, dialog box serial variable input and edit will pop up, input all serial variable value from small to great in this box, separate by comma, and bracket those seldom used data.

Change corresponding rank width by dragging the pressed left key flatly, when the cursor is in the right margin of the rank for faulty width. Sameness, if the row height is not appropriate, dragging the pressed left key vertically when the cursor is at juncture of two contiguous lines in left selection area of the table, so the row height is changed.

This dialog box provides various edit function for inputted data with the unit of row.

Locate the cursor at any row, press key insert and a blank row will be inserted above that row,

click the left selection area of row, and press key delete to delete the selected row.

Cut or copy data by dragging mouse when one row or several rows are selected, press left key and Ctrl key simultaneously to select several lines. Press left key and drag, the cursor shape will be changed, it will hint the current status is for cut or copy, if it needs copy, press Ctrl simultaneously. Release the mouse when it is dragged to proper location, the selected data will be cut or copied.

Click or double click any cell, the data will be highlighted, press ctrl and X simultaneously to cut it, press key Ctrl and C simultaneously to copy, locate the cursor at proper cell, then press Ctrl and V, the cut or copied data is pasted in the cell.

Save the inputted data as data file for future use, or read data from outer data file.

When all dimension data in each group is recorded, input serial dimension value from small to great in new row if there is serial dimension, bracket the seldom used value, use comma to separate each value. All value of one serial dimension must be inputted in the same row.

If there are more than one serial dimensions of symbol, the priority order of serial dimension value in each row should accord with the appointed order at the time of variable attribute definition.

When all options are filled, click button ok to add created symbol to the Graphics Library.

When re-extracting symbol, the created symbol will appear in corresponding type

4.5 Graphic Library management

【Command】 symman




【Icon】 

【Definition】

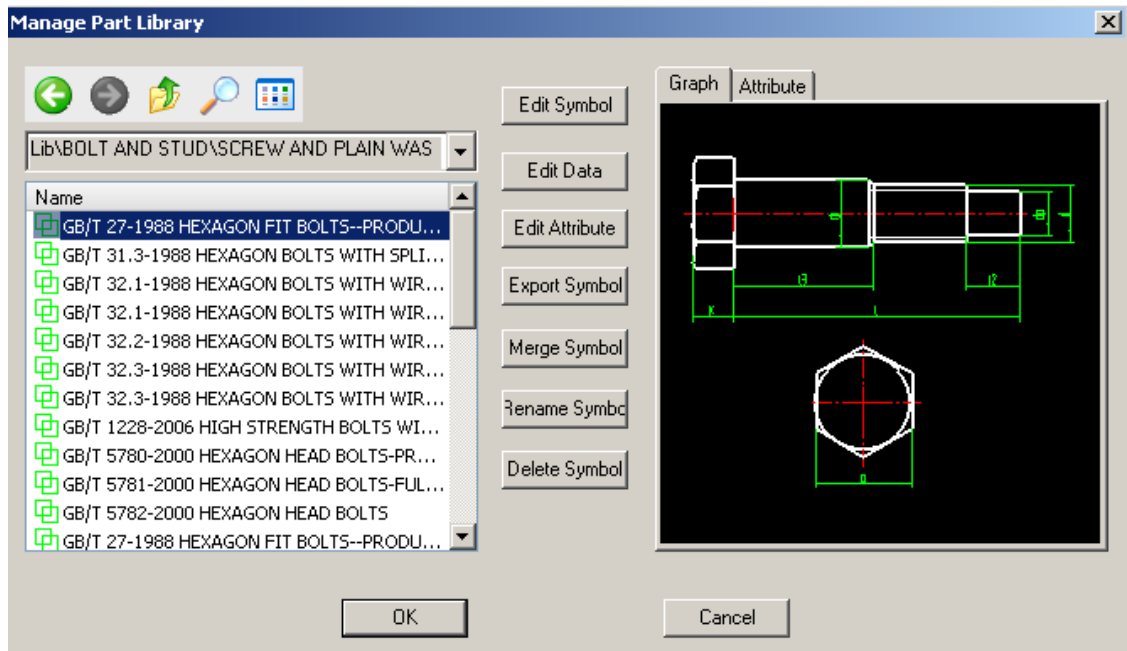
It is an open Graphics Library to users, user can extract symbol, define symbol and manage Graphics Library by Graphics Library management tool of CAXA Draft.

The operation of library management includes: symbol edit, data edit, attribute edit, export symbol, merge symbol, re-name symbol, delete symbol.

【Step】

User can implement command symman in forms of : click  button in the sub-menu of library from main menu Drawing, click  in the tool bar of library, click the button of  in the commonly-used basic drawing panel from the option card, implement command of symman.

Click command symman, following dialog box will pop up.



Pic8-16 manage part library

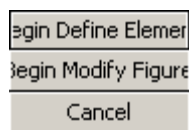
4.5.1 Edit Symbol

【Definition】

Symbol edit is actually re-definition of symbol, user can modify the original symbol generally, modify partly, add or reorganize existing symbol, and define it as a new symbol in similar type.

【Step】

Select symbol name to be edited in "**Manage Part Library**", as shown in figure, and preview the symbol in right preview box.



Click button "**Edit Symbol**", a dialog box will pop up, select first item element definition, user can modify element definition or dimension variable attribute, and the Graphics Library management dialog box is closed.

If the graphics, basic point, dimension and dimension name need to edit, user can select second item edit graphics, sameness, Graphics Library management dialog box is closed. Since the symbol need to insert in drawing area for edit, the system will remind saving the currently opened file if it's not saved. If the file is saved, it will be closed. Then each view of symbol will be

displayed , user can edit and modify it now. Because the symbol still keeps previous defined information, user can only modify the part that needs modification.

The difference between symbol edit and symbol extract is, all view and dimension variable of symbol is shown on the screen, each view is broken into individual element, every element definition expression, every dimension variable attribute and every dimension value are kept, which will reduce repeated labor.

Edit the graphics in the drawing area, such add/delete curve or dimension etc.

When the modification is finished, redefine the modified symbol as per the method introduced in section symbol definition.

If input an symbol name different from the previous one when storing the symbol, a new symbol is defined. If use the original symbol name and type, it is just modification to original symbol.

4.5.2 Edit Data

【Definition】 Modify, add or delete the original data of parametrization symbol.

【Step】

Select symbol name, of which the data is to be edited in "**Manage Part Library**" dialog box, preview the symbol in right side box, the operation method is the same as that of symbol extraction.

Click button "Edit Data", "Input and Edit Standard Data " dialog box will pop up.

Modify data in the box, operation method is the same as that of symbol definition data input.

Click button ok when it is modified, and return to "Manage Part Library" dialog box, proceed other Graphics Library management operation. When all operations are finished, click button ok to confirm.

4.5.3 Edit Attribute

【Definition】 Modify, add or delete the original attribute of symbol.

【Step】

Select symbol name, of which the attribute is to be edited in "**Manage Part Library**" dialog box, preview the symbol in right side box.

Click button "Edit Attribute", "Input and Edit Standard Data" dialog box will pop up.

Modify the attribute in the box, the operation method is the same as that of symbol definition

attribute edit, refer to relative section for concrete information.

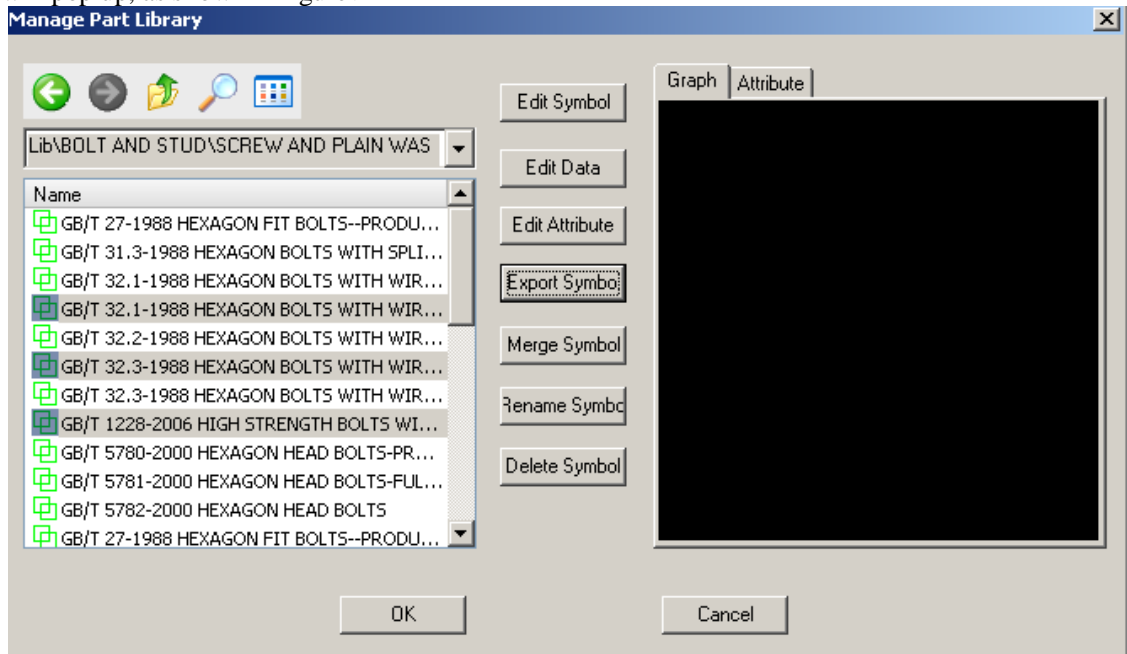
Click button ok when it is modified and return to "Manage Part Library" dialog box, proceed other Graphics Library management operation. When all operations are finished, click button ok to confirm.

4.5.4 Export Symbol

【Definition】 Save symbols need derivation by the format of *.idx.

【Step】

Click button Export symbol in "Manage Part Library" dialog box, corresponding dialog box will pop up, as shown in figure.



Pic8-17 export symbol

All corresponding symbols in this type are listed in the box, select symbol to be exported, if all symbols need to export, click button all.

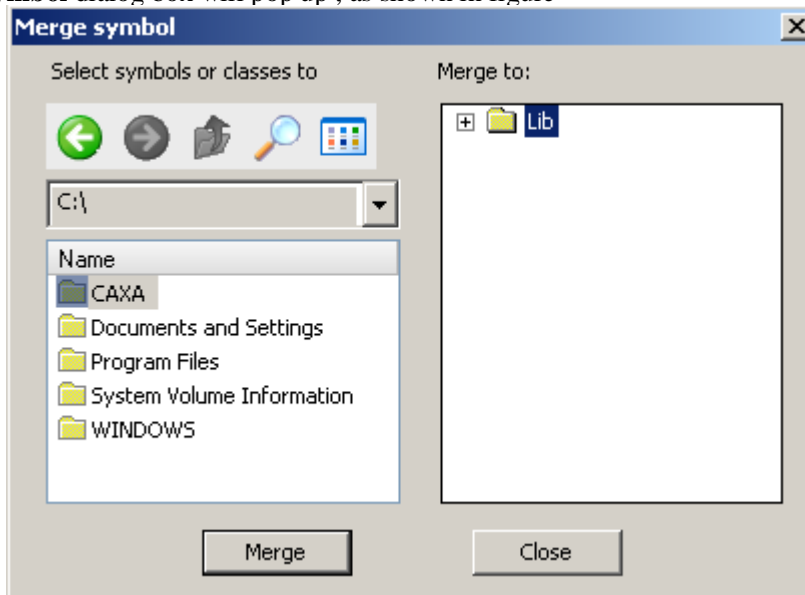
Click button export then, save as dialog box will pop up, input file name of Part Library index, and click button save to finish symbol export.

4.5.5 Merge symbol

【Definition】 Merge symbol into Graphics Library by the format of *.idx.

【Step】

Click button **merge symbol** in **Manage Part Library** dialog box, open Part Library index file dialog box will pop up, select index file , of which the Part Library is to be switched , click button open , **merge symbol** dialog box will pop up , as shown in figure



Pic8-18 merge symbol

All symbols in index file are listed in the symbol list , select the desired symbol to switch. If all symbols need to be switched, click button all, and choose type that symbols are to be put in, or input a new type name to create new type, then click button merge, The process bar at the bottom of dialog box will show switch progress.

When the switch is finished, return to Manage Part Library dialog box for other Part Library management operation . Click button ok to finish all operation.

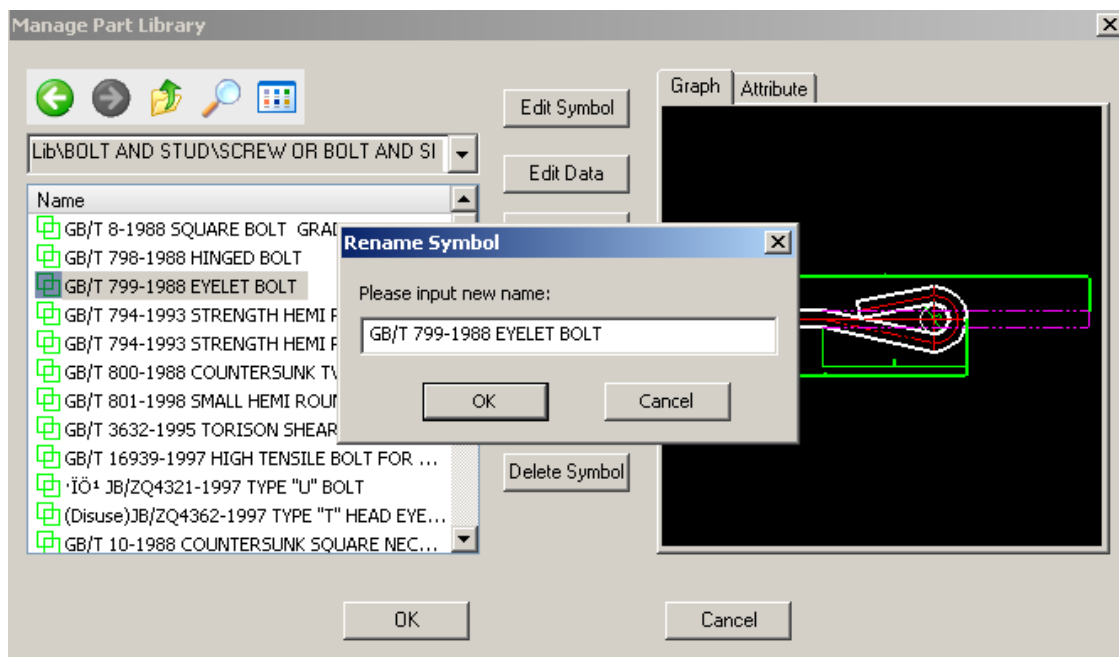
4.5.6 Rename Symbol

【Definition】 Modify symbol's original name, major class name and secondary class name.

【Step】

Select the desired symbol in Manage Part Library dialog box, preview it in right side box. The operation method is the same as that of extracting symbol.

Click button Rename Symbol, select items need modifying, if modify symbol name, click Rename Current Symbol, the corresponding dialog box will pop up, as shown in figure:



Pic8-19 rename symbol

Input new symbol name in the edit block.

Click button ok and return to Manage Part Library dialog box for other Part Library operation, click button ok when all operations are finished.

4.5.7 Delete useless symbol

【Definition】 Delete useless symbol, or delete useless big type or small type.

【Step】

Select the desired symbol in Manage Part Library dialog box, preview it in right side box, the operation method is the same as that of extracting symbol.

Click button Delete Symbol , select the symbol to be deleted, to avoid wrong operation , the system will inquire whether this symbol needs to delete really, click button ok or cancel as per actual condition.

When the delete operation is finished or canceled, return to Manage Part Library for other Part Library operation, click button ok to finish all operations.

4.6 Graphic Library exchange




【Command】 symexchange

【Icon】 

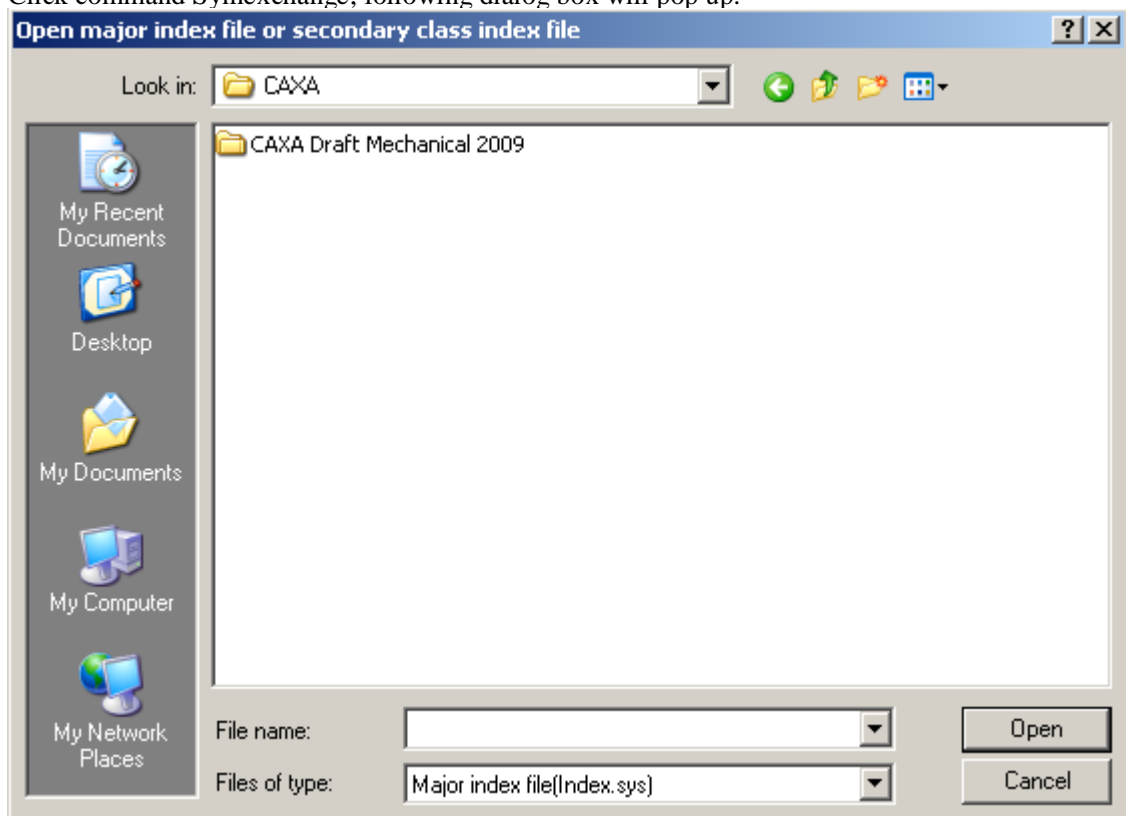
【Definition】

Graphics Library switch is to switch user-defined Graphics Library of old version into current Graphics Library format, or add Graphics Library in other computer to Graphics Library in current computer.

【Step】

User can implement command symexchange in forms of : click  button in the sub-menu of library from main menu Drawing, click  in the tool bar of library, click the button of  in the commonly-used basic drawing panel from the option card, implement command of symexchange.

Click command Symexchange, following dialog box will pop up.



Pic8-19 transform library

Prime index file(Index.sys): switch all types of Graphics Library simultaneously ;

Small type index file(*.idx): select individual type of Graphics Library to switch.

Chapter 5 Modify

5.1 Summary

Modify will be introduced to users in this chapter .

To edit and modify graph is one of the basic functions of the mutual software. It is important to improve the speed and quality of plot. CAXA draft has considered the requirement of users, and offered users function of edition and modification.

The edition and modification of CAXA draft consists of base edition, Modify, and attribute modification, among which, base edition includes usually-used edition function such as copy, cut and paste etc. Modify means moving, trimming, rotating or other operation. Attribute modification is mainly used to modify layer, linear, color etc.

As a drawing software based on the Windows platform, in order to meet user's drawing requirement, CAXA electronic board supports technology of OLE, user insert OLE of picture, chart, text, electronics form etc. in file generated in CAXA electronic board, and also, voice, animation, movie edition can be inserted. multi-media information, in addition, graph drawn in electronic board can be inserted to software that supports OLE, such as WORD. The content in this section will be introduced as follows .

5.2 Base edition

5.2.1 Summary

Base edition includes undo, redo, select all , copy and copybase, cut , paste , delete and delete all, OLE and edition.

Every function of base edition can be implemented in the following forms, corresponding keyboard command or shortcut key, corresponding button in the main menu of edit, corresponding button in usually-used option card, or button in the tool bar.


Each function will be explained concretely as follows.

5.2.2 Undo and Redo

Undo and redo is a pair of mutually related commands.


5.2.2.1 Undo

【Command】undo

【Icon】 

【Definition】 To cancel the latest edition.

【Step】

User can implement command Undo in the forms of : click Undo button in the main menu of edit, click the button of  in the toolbar of Standard, using short-cut keys of [Ctrl] + [Z]

For example, one figure is delete by mistake, user can use the command of Undo is cancel last operation, multilevel backing can be realized through such command as you like.

5.2.2.2 Redo


Redo is the opposite course of undo, only when Undo command is used, can user use redo command.

【Command】 redo

【Icon】 

【Definition】 To cancel the latest undo operation.

【Step】

User can implement command Redo in the forms of : click redo button in the main menu of edit, click the button of  in the toolbar of Standard, using short-cut keys of [Ctrl] + [Y].

Note: Undo and Redo can only be used in graphic drawing , they are not useful for OLE.

5.2.3 Select all

【Command】 selall

【Icon】 

【Definition】 Select all object in the open layer which meet the condition of pick filter

【Step】

User can implement command Selall in the forms of : click Selall button in the main menu of edit, implement Selall command, using short-cut keys of [Ctrl] + [A].

Once this command is implemented, all object in the open layer will be selected even if the pick filter are not set for the object.

5.2.4 Copy, cut and paste

Copy , cut and paste is a group of mutually related command, they will be introduced as follows.



5.2.4.1 Copyclip

【Command】 copyclip



【Icon】 

【Definition】 Save the selected figure in the clipboard to paste.

【Step】

User can implement command Copy in the forms of : click Copy button in the main menu of edit, click the button of  in the commonly-used panel from the option card , click the button of  in the toolbar of Standard, implement command of Copyclip, using short-cut keys of [Ctrl] + [C]

When the command of Copyclip is implemented, select object to be copied then confirm, the selected one will be saved in the clipboard for paste . Meanwhile, user can select a figure to be copied first , then click Copyclip.

Note: User can use the command of Copy as per base-point, the difference between copy and copy as per base-point is : user should specify base point of figure, when paste, it also needs to specify base placement. But for Copyclip, it doesn't need to specify base point. When paste, the default base point will be the selected down-left point. User can use copybase command in the forms of : Click copybase command in the main menu of edit, click button  in the commonly-used panel from the option card, or click  in the toolbar of Standard , implement copybase command, use short-cut key [Ctrl]+[Shift]+[C].

5.2.4.2 Cutclip



The function of Copyclip and cutclip is the same, but for copyclip, the original selected figure will be kept, yet if you use cutclip, the original one will be deleted , it will only be saved in the clipboard.

【Command】 Cutclip

【Icon】 

【Definition】 Delete selected object and save it into the clipboard for paste.

【Step】

User can implement command Cutclip in the forms of : click Cutclip button in the main menu of edit, click button  in the commonly-used panel from the option card, click the button of  in the toolbar of Standard, implement cutclip command, using short-cut keys of [Ctrl] + [X].

When the command of Cutclip is implemented, select object to be cut then confirm, the selected one will be deleted and saved into the clipboard for paste . Meanwhile, user can select a figure to be cut first , then click Cutclip.

5.2.4.3 Pasteclip

【Command】 Pasteclip

【Icon】 



【Definition】 Paste the object in the clipboard to specified position.



Copy and cut the object and save it into the clipboard, The system will use different internal format to save information in the clipboard, all applicable format will be used to save information when copy or cut. And when paste information in the clipboard, it will use the format that has the most information. For example, the content in the clipboard is selected object from CAXA Draft, when paste it in the window of CAXA Draft, it will be the same as the selected one.

In addition, user can use Special Paste to select different paste modes, for example, Windows primitive format, which includes screen vector information, what's more, it can be zoomed out or printed when the resolution hasn't been reduced. But it is unable to edit it with CAXA Draft Modify function.

Note: When copy or paste between different Windows program, the selected content will exist in the form of OLE. It will be explained in next section.

【Step】

User can implement command Pasteclip in the forms of : click Pasteclip button in the main menu of edit, click button  in the commonly-used panel from the option card, click the button of  in the toolbar of Standard, implement pasteclip command, using short-cut keys of [Ctrl] + [V].

The modes of Special paste: Click Pastespec button in the main menu of edit, click button  in the commonly-used panel from the option card, click the button of  in the toolbar of Standard, implement Pastespec command.

5.2.5 Insert Object

The abbreviation of Object Linking and Embedding is OLE, it is a mechanism of Windows. User can insert the object , such as text, table, picture etc. created by other Windows program into file, so as to meet multiple requirements, and user will create all kinds of file. The operation of OLE includes: insert object, delete, cut, copy, paste and special paste , open and edit object, object convert, object linking, check object attribute etc. The figure drawn by CAXA draft itself can be regarded as one OLE and inserted into other software that support OLE.

Following is the detail.

5.2.5.1 Insert obj



【Command】Insertobj

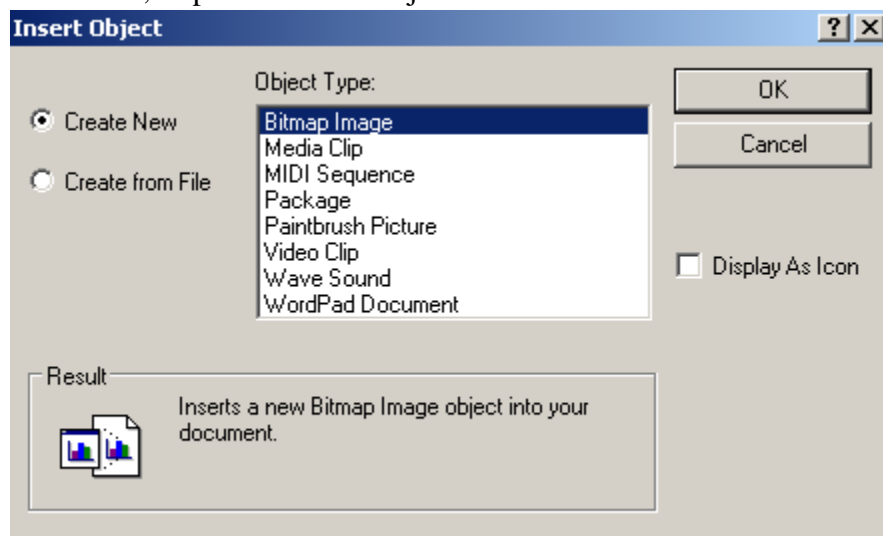
【Icon】

【Definition】 User can input information to the figure from other program that supports OLE.

Insert one OLE in the file, User can create new object, or create object from existing file, .the newly created object can be embeded object or link object.

【Step】

User can implement command Insertobj in the forms of : click Insertobj button in the main menu of edit, click button  in the commonly-used panel from the option card, click the button of  in the toolbar of Standard, implement Insertobj command.

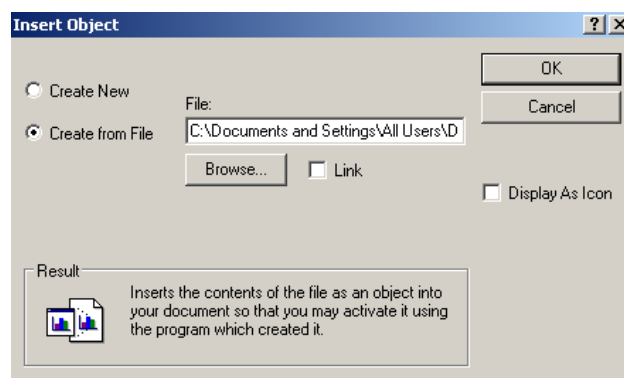


Pic5-1 insert object

Click the command of Insertobj, a dialog box will pop up , as shown in figure 5-43,

1, By default, Insert object dialog box will be shown in the form of create new object, in which OLE type registered in the object type listbox, then user can select needed object, click the button of OK, corresponding object edit window will pop up, in which user can edit . For example, select BMP image, then the window of Paintbrush will pop up, user can edit here.

2, If New is not selected in the dialog box, but Create from file is selected, the dialog box will be:



pic5-2 create object from file

3,Click the button of Browse, then Browse dialog box will pop up, then select desired file from

the file list, the file will be inserted into document as object.

4, The method for the above mentioned ones is to insert object to file, the inserted object has already been a part of CAXA Draft file. In addition to insert, user can insert object in the form of link. The real difference between link and insert is : the object to be link, actually, is not a part of CAXA Draft file, it exist in an external file, only link information will be kept in CAXA Draft file. When external file is modified, the object in CAXA Draft file will updated automatically. It is easy to link object, when file in the dialog box of figure 5-44 is selected, select the check box of Link, then click OK, that means the object will be linked to the file.

5, There is another check box of Show as Icon, if this check box is selected, the file will be shown as Icon, instead of the object content itself.

Note: The type of object can be inserted is determined by the software type in user's computer. For example, if software of Word is not installed in the computer, then document or table created by Word will be unable to be insert in CAXA Draft.

5.2.5.2 Open and edit OLE

Modify object position, size and content.

(1) Click object, there are eight black block named dimension handle around the selected object. The object size can be modified when dragging dimension handle. If dragging object, user can change object position. Click left key, dimension handle will disappear. Inspect whether the pulldown box at the lower right corner is gray, if it is gray, press ESC to resume normal select state.

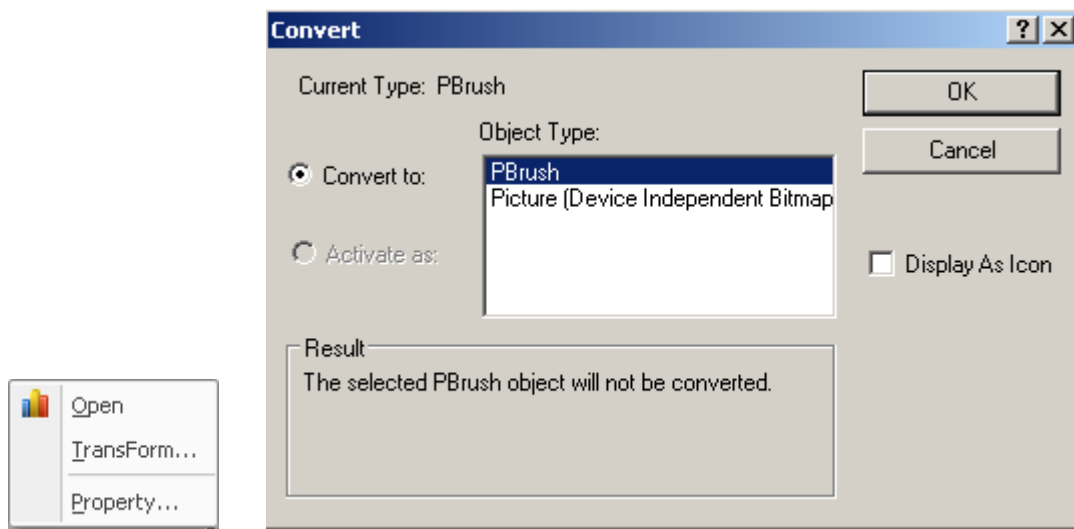
(2) There are two methods to open and edit inserted object, one is direct edit, it doesn't need to open object editor any more, because interface of editor is merged in EB interface. When the edit is finished , press ESC to return . The other method is wide-open, an edit window of object can be opened separately by this method. For linkage object, there is only one edit mode, that is "wide-open".

(3) To newly inserted object, user can proceed first edit in the form of wide-open when being inserted into file.

(4) To the inserted object, when it is selected, user can edit, open, convert or Attribute ,as shown in figure 5-45.user can edit by mode of direct edit or wide-open. To linkage object, user will edit it only by one mode of wide-open. If the option of Convert to is selected, dialog box as figure 5-46 will pop up,then the current object will be convert to another format. If select Activate as, all other selected programs will be used to start when the object is open.

Once the object is selected, user can move or zoom by dragging pinch point.

(5) In addition, double click object, user can edit by the mode of direct edit. If press CTRL while double click, user can edit by the mode of wide-open.



pic5-3 transform

pic5-4 convert type

5.2.5.3 Paste special

Paste content in clipboard as per desired type and mode to file.

(1) In Windows software that supports OLE, select and copy part content to clipboard, for example, copy a line of text in Microsoft Word, click right key at the selected object, a shortcut menu will pop up, as shown in the figure below.

(2) The origin of copied content is listed in the above dialog box.

(3) If paste is selected, then the object will be inserted into file as plain text, Microsoft Word 6.0 or Picture.

(4) If paste link is selected, then the selected text will be inserted into file as linkage object.

5.2.5.4 Link object

(1) At first, click left key to select inserted linkage object .

(2) Click option link in the menu of edit, a dialog box will popup, as shown in figure. It should be noted that, if embedded object is selected, the option of link will be gray.



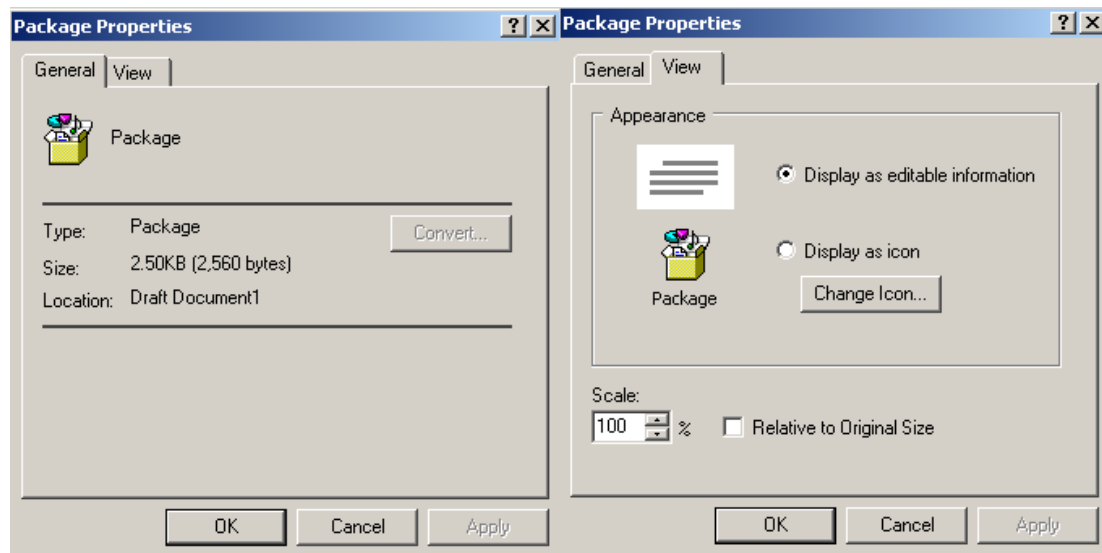
pic5 link

- (3) In the above dialog box, origin , type and update mode of linkage object are listed . If by hand is selected, user can update object by button update now, if automatic is selected, the inserted object will be updated when its origin is modified.
- (4) User can open object origin by click button open origin, in order to edit linkage object.
- (5) If button modify origin is selected, corresponding dialog box will pop up, in which user can select other files that are the same as original object type, so that user can modify origin of linkage object.
- (6) If click button break link, the linkage between object and its origin will be broken.

5.2.5.5 Object attribute

Select an object, and check its attribute, convert object attribute , modify object size, icon and display mode. If the object is inserted by the mode of linkage, user can operate link for it.

For example, select an object of BMP bitmap, click option object attribute in the menu of edit, corresponding dialog box will pop up, as shown in figure:



pic5-49 object properties 1

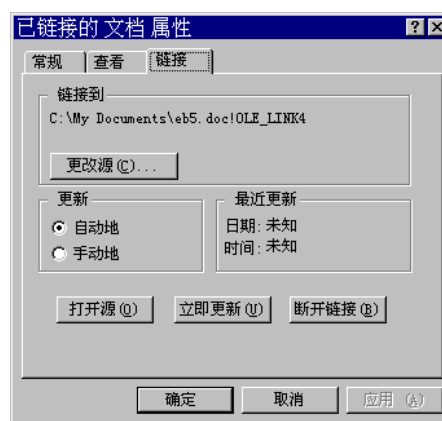
pic5-50 object properties 1

There are two options in top left corner of box, they are **regular** and **check**. Object type, size and position are listed in the option of regular.

User can convert the embedded object to other format by button switch, in order to reduce file space. If option check is selected, a relevant dialog box will pop up.

Select display mode of object, or click button modify icon to change object icon, input scale factor in the edit box to change object size. If square frame of relative to original dimension is selected, corresponding display dimension will calculated.

If linkage object is selected, there will be one more option of link in the dialog box, as shown in figure:



pic5-51 link properties

5.2.5.6 Use right key function to operate object

Use right key function to operate object

Click right key within OLE object, a shortcut key will pop up, select submenu of BMP picture object, all operation on OLE object can be realized in this submenu.

5.2.5.7 Insert graph drawn in CAXA Draft into other software

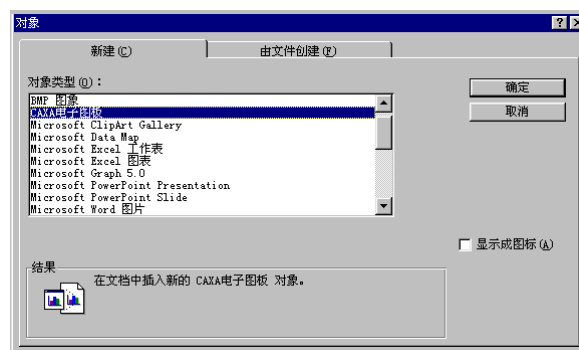
Graph drawn in CAXA Draft, as an OLE object, can be inserted into other software that supports OLE, following is the operation detail.

Insert CAXA Draft object

User can create new object or create object from existing *.exb file. The newly created object can be inserted object or linked object.

(1) At editing state of Word, move the cursor to the position that EB object is be inserted in.

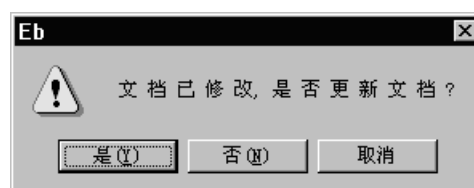
(2) Click option object in the menu of insert, a dialog box will pop up, as shown in figure:



pic5-52 insert CAXA object

(3) There are two methods to create, one is new, the other is create by file. Select the type of CAXA Draft in the list box of new option, then click button ok ,EB edit window will be opened automatically, in which user can edit desired graph.

(4) When graph is drawn, close electronic board window, a dialog box will pop up , as shown in figure.



pic5-53 update dialog box

If yes is selected, the drawn graph will be inserted into Word file as an OLE object.

(5) By dragging eight dimension handle around EB object, user can adjust object size, or double click object to edit.

Note: graph size and shape inserted into Word is determined by the size and shape in drawing area, before closing EB, user should use view all to make the drawn graph be displayed completely in drawing area.

(6) If the mode create by file is selected, user can create object from existing *.exb file. The newly created object can be embedded object or linkage object.



5.2.6 Erase and Erase all

【Command】 erase

【Icon】 

【Definition】 Delete object from figure

【Step】

User can implement command Erase in the forms of : click Erase button in the main menu of edit, click the button of  in the commonly-used panel from the option card , click the button of  in the toolbar of Standard, implement command of Erase.



Once such command is implemented, select object to be deleted and confirm, then the selected one will be deleted. If it needs to interrupt such command, press the key of ESC to exit. It also support select object first, then implement Erase.

【Command】 eraseall

【Icon】 

【Definition】 The object in the open layer which meets pick filter condition will be all erased.

【Step】

User can implement command Eraseall in the forms of : click Eraseall button in the main menu of edit, click the button of  in the commonly-used panel from the option card , click the button of  in the toolbar of Standard, implement command of Eraseall, then the following dialog box will pop up.



pic5-38 delete all

Click OK , then all object will be erased, if click Cancel , the latest operation will be cancelled.

5.3 Modify

5.3.1 Summary

In CAXA Draft, we can edit curve, block, text or label through the operation of pinch point, erase repeated line, move, copy, trim, edge, chamfer, mirror, zoom, array, stretch, break etc.

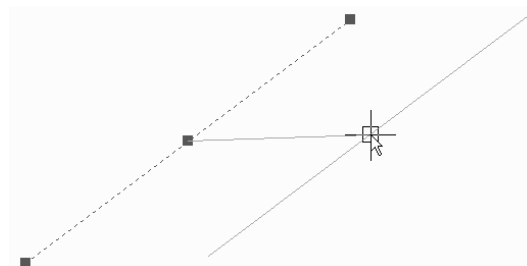
Each function can be realized via keyboard command or short-cut key, or click corresponding button in the main menu of edit or in the toolbar.

5.3.2 Pinch point edit

As for pinch point, we have introduced it in the section of 2.4.2. Pinch point edit means moving, stretching, rotating , zooming figure by dragging pinch point. Different pinch point of different figure will have different indication. For example.

【Move by using pinch point】

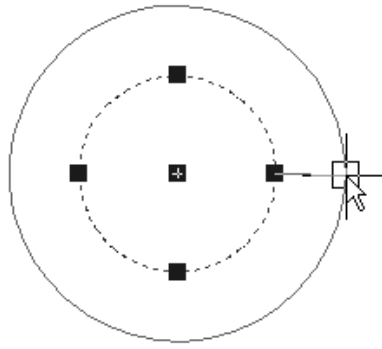
Move object by selected pinch point, the selected object will be highlighted and moved to one direction and distance, as shown in following figure, select straight line and mid-pinch point to move.



Click left key to confirm, then the line will be moved to the preview position .

【Stretch by using pinch point】

Select pinch point and move the object to new position , then stretch it, but the pinch point in text, reference block, midpoint , circle center, the object will only be moved instead of being stretched. If we select a pinch point in the circle, then drag, the circle will be stretched, as shown in following figure.



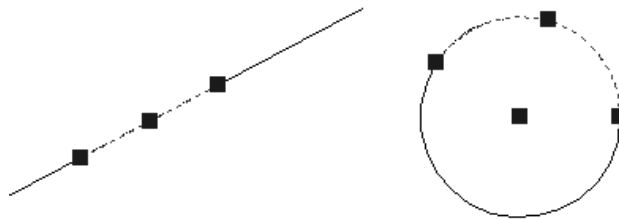
5.3.3 Erase repeated line

【Command】 eraseline

【Icon】



【Definition】 Erase element in the selected figure or completely repeated element.

This command is only useful for straight line, circle, arc, ellipse, furthermore, the object needs to erase is completely coincided or included in the remaining part. As shown in following figure.



When the command is implemented, the dash line will be erased.

【Step】

User can implement command Eraseline in the forms of : click Eraseline button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Eraseline.

Once such command is implemented, select object to be deleted and confirm, then the selected one will be deleted. It also support select object first, then implement Eraseline.

5.3.4 Move

【Command】 Move

【Icon】

【Definition】 Move selected object as per specified angle and direction.

The following immediate menu will be used for the command of move, as shown in following figure.



1. Specify two points	▼	2. Keep original	▼	3.Rotate Angle	0	4.Scale	1
-----------------------	---	------------------	---	----------------	---	---------	---

pic5-6-2 Move

Parameter introduction:

- (1) Offset mode: It means moving graphics element by two point positioning for two given point. Moving entity to a specified position as per given offset.
- (2) Graph status: It means moving entity to a specified position, and select keep original state or move as block as per request in 2: of immediate menu.
- (3) Rotate angle: When moving graph, user can specify rotate angle of entity by keyboard input
- (4) Scale: Before moving, user can specify scaling factor of graph to be moved.

【Step】

User can implement command Move in the forms of : click Move button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Move.

Once such command is implemented, select object to be deleted , set parameter in the immediate menu and confirm, then the selected one will be moved.

There is difference between two points and offset for offset mode, details is shown as follows:

- (1) Two points: Select figure, input two points by keyboard or click left key , then move.
- (2) Offset: Select figure, a base point will be given by the system, generally speaking, base point of line will be midpoint. Input offset value for X direction and Y direction, then user can move.

Use coordinate, grid capture, object capture, dynamic input or other tool to move object precisely, and convert orthogonal, polar axis or other operation status. The command of Move support first selection then move.

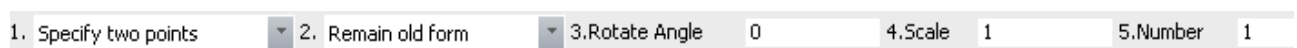
5.3.5 Copy

【Command】 copy

【Icon】 

【Definition】 Create duplicate for the selected object as per specified angle and direction.

The copy in this section means create duplication of figure only in the same CAXA Draft file, the selected object will not be saved in Windows clipboard, it is not related to paste. User use the following immediate menu to copy and move.:





pic5-6-3 immediate menu for copy

The detail parameter is shown as follows:

- (1) Offset mode: Copy and paste element by the mode of two points locating, then move and copy.
- (2) Graph status: Copy entity to a specified position, user can select original state and paste as block in 2: of immediate menu as per request.
- (3) Rotate angle: When copying or moving graph, it is allowed to specify entity rotate angle, the angle can be inputted via keyboard.
- (4) Scale: Before copy, user can specify scaling factor for copied graph.
- (5) Copy No.: Only when copy is selected, user can determine copy number in 6:of immediate menu. According to specified two point distance and number, the system will calculate space between copied graphs.

If the number in immediate menu is more than 1, the system will calculate space between copied graphs as per specified two points.

【Step】

User can implement command Copy in the forms of : click Copy button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Copy.

When copy is implemented, select object that needs to copy, set parameter in the immediate menu and confirm, then it will be copied.

The offset mode of given two points is different from given offset mode, detail is shown as follows:

- (1) By given two points: When graph is selected, input the first point and second point position by keyboard or left-click to move and copy.
- (2) By given offset mode: Select graph, a base point will be given automatically, generally speaking, base point of line will be midpoint. Input offset value for X direction and Y direction, then user can move and copy.

Use coordinate, grid capture, object capture, dynamic input or other tool to move object precisely, and convert orthogonal, polar axis or other operation status. The command of Copy support first selection then copy.

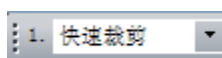
5.3.6 Trim

【Command】 trim


【Icon】 

【Definition】 Trim the object, make it terminate precisely specified by other object edge.

There are three trim modes: Quick trim, pick border and batch trim, use the following immediate menu to implement trim mode as you like.



pic5-6-3 immediate menu for trim

User can implement command Trim in the forms of : click Trim button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click

the button of  in the toolbar of Modify, implement command of Modify.

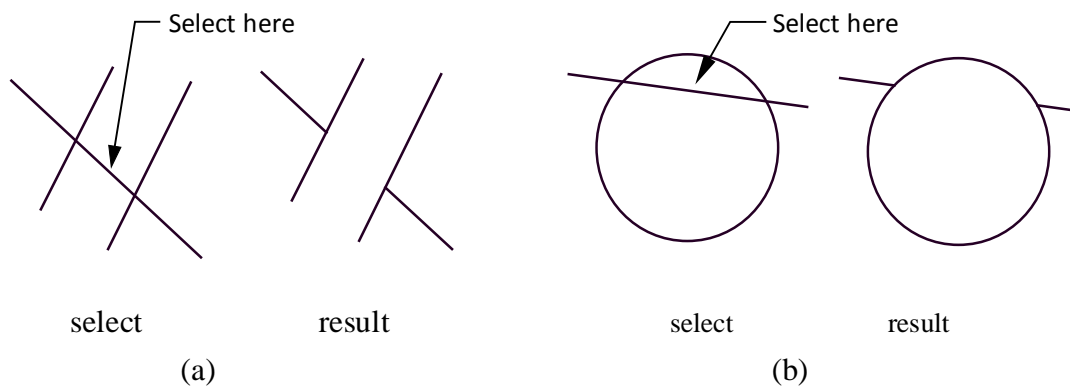
5.3.6.1 Quick trim

【Definition】 Select curve to be trimmed, the system will judge the edge automatically and respond accordingly.

User can trim between any intersected lines conveniently, the method is: select line segment by mouse directly, it will specify trim edge as per intersected curve automatically, then left-click to trim.

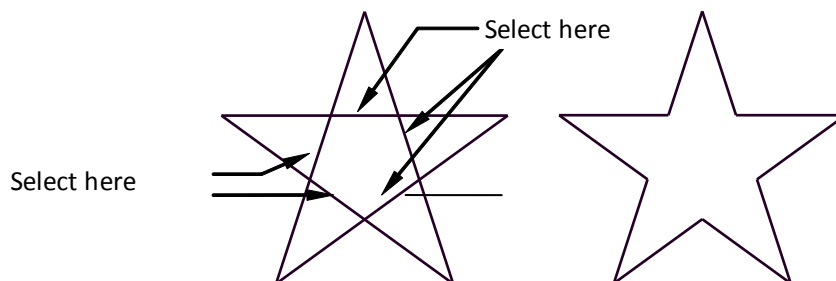
【Example】

Example one, Following example express different selected position will cause different trim result.



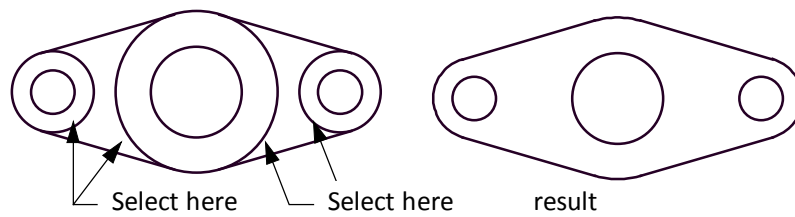
pic5-2 select position of quick trim

Example two: The figure below expresses trim lines quickly.



pic5-3 quick trim line

Example three: The figure below is example of quick trim circle and arc.



pic5-4 quick trim circle and arc

【Step】

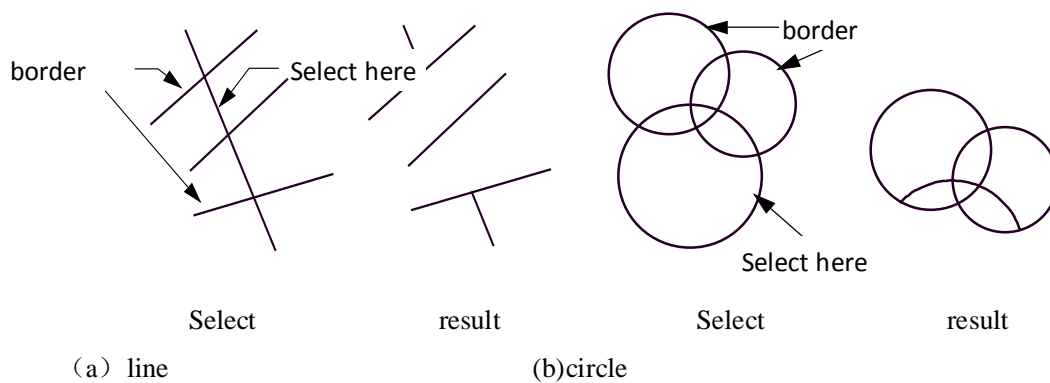
Implement the command of trim, select Quick trim or other trim modes in the immediate menu, click object to be trimmed, press ESC key to exit.

5.3.6.2 Pick border

【Definition】 To complicated intersected border, user can pick border to trim.

Pick one or more curves to form trimming border, and trim a series of curves, and border at the other side of will be kept. In addition, Scissor line can also be trimmed.

For the mode of picking border, user can trim a series of curves accurately. It is more speedy for complicated curves.

【Example】

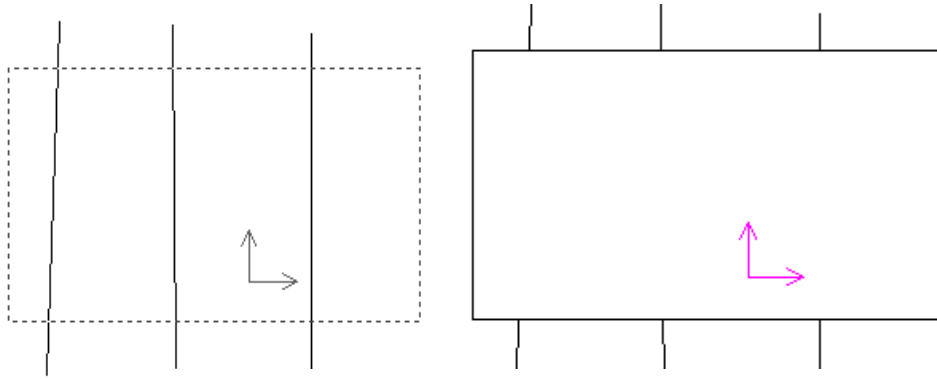
pic5-5 trim by pick border

【Step】

Implement command of Trim, select Pick border in the immediate menu, then select one or more curves as per hint, and click right key to confirm. It will then hint "pick curve to be trimmed". The selected border will be trimmed, and border at the other side of will be kept.

5.3.6.3 Batch trim

【Definition】 For multi curves, user can adopt batch trim.

【Example】**【Step】**

Implement command of Trim, select Batch trim in the immediate menu, then select scissor link as per hint, then confirm, the curve to be trimmed will be selected by window, click right key, specify trim direction, then it is finished.

Scissor link can be one curve or multi end-to-end curve.

5.3.7 Edge

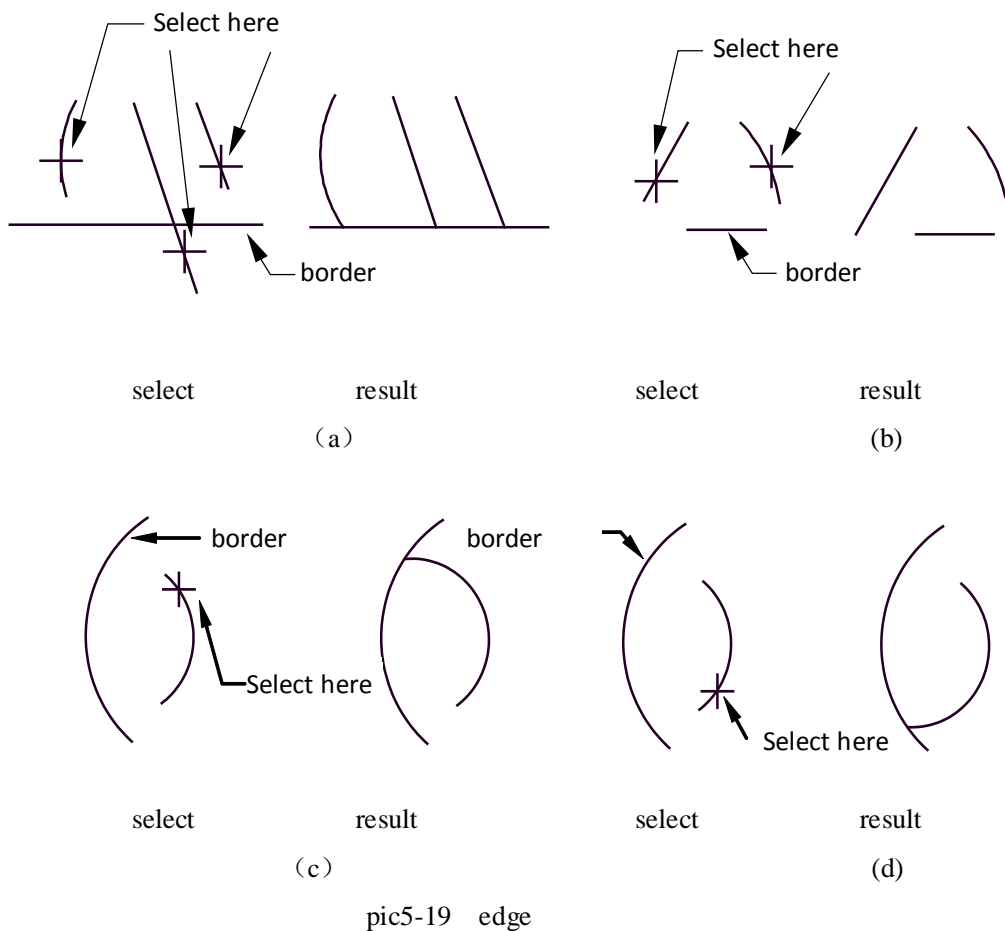
【Command】 edge

【Icon】



【Definition】 Trim or extend a series of curves regarding one curve as boundary.

If the selected curve is intersected with bound, user should operate by command of trim, then the selected curve be trimmed till the bound. If no intersection point exists, the curve will be extended to bound as per its own trend, such line direction, center and radius of arc will keep the same.

It should be noted that, there may be exception to circle or arc, because it is unable to extend it without end, the extending range is limited by radius, and only one end of arc can be extended, as shown in figure(c)and(d).



【Step】

User can implement command Edge in the forms of : click Edge button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Edge.

Implement the command of Edge, Select a curve as bound, it will hint "select curve to be edited". User should select a series of curves to be edited as per hint, and click right key to finish.

5.3.8 Corner

【Command】 corner

【Icon】 



【Definition】 Modify object, and link it in modes of fillet or chamfer, or other modes.

Fillet is divided into several modes, they round fillet, multi-fillet, chamfer, **outside chamfer or inside chamfer, multi-chamfer, sharp corner** etc, user can select any mode in the following immediate menu.



pic5-6 immediate menu for corner


User can implement command Fillet in the forms of : click Fillet button in the main menu of

Modify, click the button of  in the commonly-used Modify panel from the option card, click the button of  in the toolbar of Modify, implement command of Fillet.

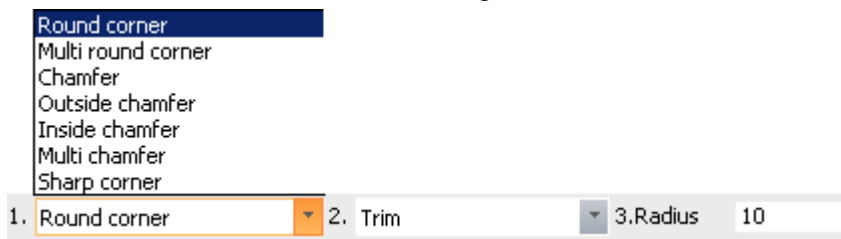
5.3.8.1 Fillet

Smooth Fillet for two arcs or lines.

【Command】 Fillet

(1) Click and select command **Fillet** in the submenu of modify, or click button  in the toolbar of edit.

(2) Click 1: in the immediate menu, an option menu will pop up, from which user can select desired chamfer form, as shown in figure.



pic5-7 immediate menu for fillet

(3) Click 2: of immediate menu, an option menu will pop up, as shown in figure.

Click any of them can switch trim mode. The meaning of option menu is introduced as follows:

- 1) Trim: Trim all redundant sides after being trimmed.
- 2) Trim first line: Only trim redundant part of initial side, initial side is the first selected curve.
- 3) Not trim: When chamfer has been executed, the original line segment keep the same, instead of being trimmed.

(4) Click 3: of immediate menu, input radius value of chamfer arc as per hint.


(5) Select first curve to be chamfered as per hint, the selected one will be highlighted, then it will hint "select second curve", when second curve is selected, there will be a smooth arc to chamfer between these two curves.

It should be noted that, user will get different chamfer result as per different selected curve position. What's more, the dimension of chamfer arc radius should be appropriate. Otherwise, user can't get correct result.

5.3.8.2 Multi-Fillet

Chamfer a series of end to end lines by given radius.

【Command】 corner

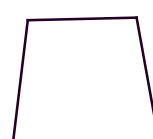
- (1) Click and select command **Fillet** in the submenu of modify, or click button  in the toolbar of edit.
- (2) Click 1: of immediate menu, and select **multi fillet**.
- (3) Click 2:radius in immediate menu, input a real number via keyboard and redefine chamfering arc radius.
- (4) Select a series of end to end lines to be handled as per instruction, these lines can be closed or unclosed. For example:



Before Multi-Fillet



after Multi-Fillet



Before Multi-Fillet




after Multi-Fillet

pic5-8 Multi-Fillet

5.3.8.3 Chamfer

Chamfer between two lines, the line can be trimmed or stretched along the angle.

【Command】 Corner

- (1) Click and select command **Fillet** in the submenu of modify, or click button  in the toolbar of edit.
- (2) Click 1: of immediate menu, select **chamfer**.
- (3) Select trim mode from 2: of immediate menu. User can refer to **round fillet** for operation method.
- (4) In the immediate menu, 3: length indicates axial direction length, 4: chamfer means angle of chamfer. User can input length and angle via keyboard as per hint. Among which, axial direction indicates length begins with intersection point along first selected line. Angle means contained angle between chamfer line and first selected line, the range is 0-180, as shown in the figure below. The length and angle are both related to first selected line, different chamfer is produced by different selection order.

5.3.8.4 outside chamfer or inside chamfer

Draw **outside chamfer or inside chamfer for three perpendicular lines. Implement the command Corner,**

- (1) Click 1: of immediate menu, select **outside chamfer or inside chamfer**.
- (2) In the immediate menu, 2: length indicates axial direction length, 3: chamfer means angle of chamfer. User can input length and angle via keyboard as per hint to change the default ones.
- (3) Select three perpendicular lines as per hint, the lines are similar to lines in the following figure, line a and line b are perpendicular to line c, they are on the same side of line c.
- (4) Result of outside or inside chamfer are not related to selection order, it is determined only by perpendicularity.

5.3.8.5 Multi Chamfer

Chamfer a series of end to end line. Implement the command Corner, then the following menu will pop up.

- (1) Click 1: of immediate menu, select **multi chamfer**.
- (2) In the immediate menu, 2: length indicates axial direction length, 3: chamfer means angle of chamfer. User can input length and angle via keyboard as per hint to change the default ones.
- (3) Select end to end lines as per hint, the method is similar to the method of multi fillet.

5.3.8.6 Sharp Corner

At the intersected point of two lines, form a sharp corner, if there is intersected point, then this point will regarded as bound, and redundant part will be trimmed. If there is no intersected point, the system will calculate intersection point automatically, and extend curves to intersection point.

Click 1: of immediate menu, select **sharp corner**, select first curve and second line as per hint, then sharp corner is finished.

It should be noted that, user will get different chamfer result as per different selected curve position.



5.3.9 Rotate

【Command】 rotate

【Icon】 


【Definition】 Rotate or rotate while copying the selected graph.

【Step】

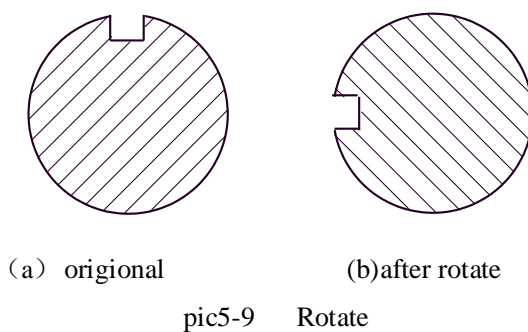
User can implement command Rotate in the forms of : click Rotate button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Rotate.

Implement command of Rotate, the following immediate menu will pop up.

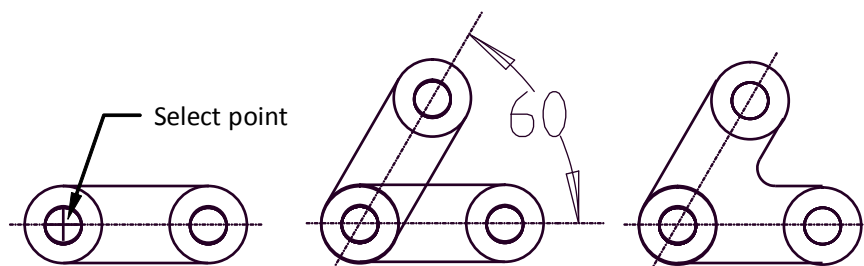


- (1) Click and select command Rotate in the submenu of modify, or click button  in the toolbar of edit.
- (2) Select entity to be rotated as per hint, single select or select by window are both applicable, the selected entity will be highlighted, click right key to confirm when selection is finished.
- (3) Then it will hint "base point", specify a rotate base point by mouse, it will hint "rotation angle". User can input angle via keyboard or move mouse to determine rotate angle. When it is determined by mouse, the selected entity will be rotated along with moving cursor. If rotate position is determined, click left key to finish rotate operation.
- (4) Click 2:Rotate in immediate menu, it will become 2: copy. User can copy as per content in immediate menu. The operation method of copy is the same as that of rotate operation, but if it is copied, the original graph won't disappear.

Example one: In the following figure, the graph is rotated instead of being copied.



Example two: The figure below is an example of rotate and copy.



pic5-10 rotate copy

【flash animation demo】



5.3.10 Mirror

【Command】 mirror

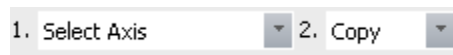
【Icon】 

【Definition】 Regard one line as axis, Mirror or copy symmetrically the selected object.

【Step】

User can implement command Mirror in the forms of : click Mirror button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Mirror.

Implement the command Mirror , then the following immediate menu will pop up.



(1) User can select entity to be mirrored as per hint, single select or select by window are both applicable, the selected entity will be highlighted, click right key to confirm when selection is finished.

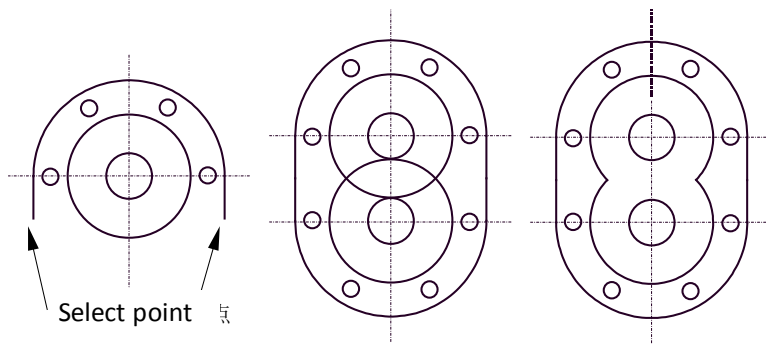
(2) It will hint "select axes", select an axes for mirror, a new graph will pop up regarding the selected axes as symmetric axes. Meanwhile, the original entity will disappear.

(3) Click 1: select axes in immediate menu, the content will be "specify two points". That means user can specify two points, connecting line of two points will be symmetric axes for mirror, the other operation is the same as previous one.

(4) If click 3: mirror in immediate menu, the content will be copied, the operation of copy is the same as that of mirror operation. But if it is copied, the original graph won't disappear.

Note: If user needs to move graph orthogonally, press F7 key or click Orthogonal button in the status bar.

Example: the figure below is example of mirror operation. Draw and select entity in figure (a), select two end points as symmetric reference for mirror operation, the result is shown in (b), then trim redundant lines , figure (c) is got.



pic5-11 mirror



5.3.11 Scale

【Command】 Scale

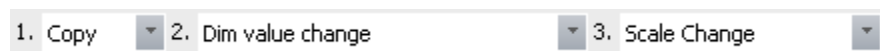
【Icon】

【Definition】 Zoom in or Zoom out select object.

【Step】

User can implement command Scale in the forms of : click Scale button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Scale.

Implement command of Scale, select object as per prompt, then click right key, following immediate menu will pop up.



pic5-12 immediate menu for scale

- (1) Select 1:Copy in the immediate menu, then when new aiming graph will be zoomed, the original graph will keep the same. If click option1:, it will convert to the option Move, then when zooming, only the newly aiming graph is produced, the original graph will disappear.
- (2) Dimension value unchangeable: Click this item, it will be Dimension change, If dimension element is included in selected element, it can control dimension change. When Dimension value doesn't change is selected, the selected dimension element won't be changed along with changing scale. On the contrary, if dimension value change is selected, it will be changed along with changing scale.
- (3) Specify a base point for scale by the mouse, then input scale modulus. According to base point and current cursor point, it will calculate scale factor when dragging mouse, and changing result will be displayed on screen dynamically,. When the input or cursor position is determined, click left key, a changed graph will be displayed on screen soon. User can input scale factor via keyboard directly.

5.3.12 Array

【Command】 array



【Icon】 

【Definition】 The aim of array is to produce several parallel graphs simultaneously.

There are three array modes, round array, rectangle array and curve array. Following is the immediate menu for array.




pic5-13 immediate menu for circular array

User can implement command Array in the forms of : click Array button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Array.

5.3.12.1 Round array

【Definition】 Regard one base point as circle center, then copy and array the select object.

【Step】

1) Click and select command array in the submenu of modify, or click button  in the toolbar of edit. User can proceed polar array as per current immediate menu and operation hint, and four graphs after being arrayed are distributed uniformly.



pic5-14 immediate menu for circular array

(2) Select entity, the selected one will be highlighted, click right key when the selection is finished. Select center point and reference point by left key, then the array result is displayed. As for the meaning of center point and reference point, user can refer to the mark in the example below.

(3) Since 2: rotate in immediate menu is selected, the graph being arrayed will be rotated automatically.

(4) Insertion point position will be calculated automatically as per content 3:uniform and 4: number in immediate menu, contained angle between each point is the same. Each array graph is listed on the circle uniformly. The number of array graph includes selected entity.

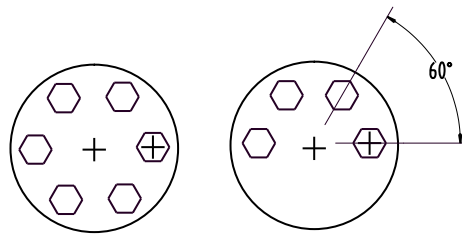
(5) Click 3: uniform in immediate menu, it will be switched to:



pic5-15 immediate menu for circular array

The meaning in immediate menu is: Proceed polar array at contained angle of 30 degree, the fill angle of array is 360 degree. Around center point, rotate counter clockwise from the selected entity position, the angle is called fill angle. Neighbor angle value and array fill angle value can be inputted via keyboard.

The next figure is example of round array operation, among which figure(a)expresses uniform mode, figure(b)expresses specified contained angle of 60 degree, array fill angle is 180 degree.



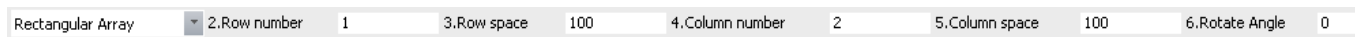
pic5-16 Round array

5.3.12.2 Rectangle array

【Definition】 By the mode of rectangle array, user can copy and array the selected entity.

【Step】

(1) User can switch 1: rectangle array to 1: Polar array.



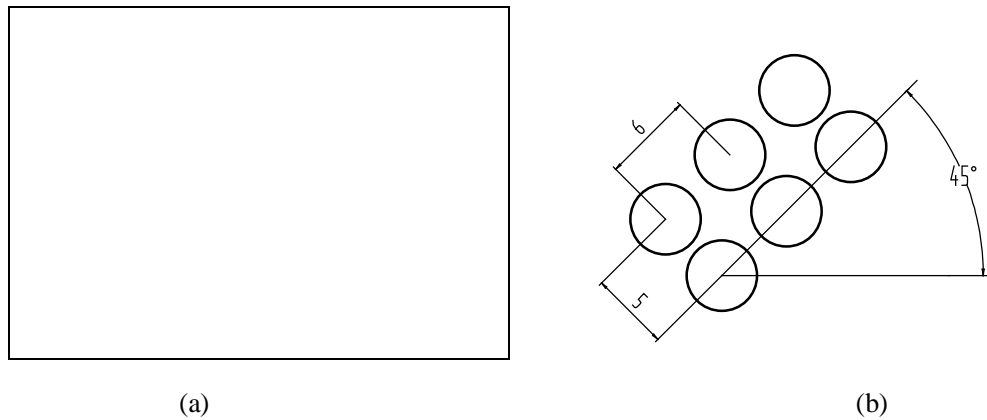
pic5-17 immediate menu for rectangle array

2) As shown in above immediate menu, row number, row space, column number, column space and default rotate angle are specified, all values can be modified via keyboard input.

(3) As for row space and column space, it means space between base point of each element, rotate angle means contained angle with X –axis positive direction.

Example:

Two rectangle array examples are shown in the figure below, in figure (a) of 5.34, the row number is 3, row space is 7, column number is 4, column space is 8, the rotate angle is 0 degree. In figure (b), the row number is 2, row space is 5, column number is 3, column space is 6, the rotate angle is 45 degree.



pic5-18 rectangle array

5.3.12.3 Curve array

【Definition】 It means producing series of identic graphs on one or several end to end curves. The structure of each identic graphs is the same, it's position is different because of the option Rotate/Unrotate.

【Step】

- (1) Implement the command Array, following immediate menu will pop up, user can click 1: option to switch.



(2) Curve selection mode:

Curve can be selected individually, and also it can be selected by chain, one or several end to end curves can be selected by chain.

For one curve selection, the array will begin from start point of it.

For chain selection, the array will begin from start point of selected curve.

Curve type:

For one curve selection, the curve can be line, arc, circle, spline, and ellipse etc. For chain selection, there are only line, arc or spline in the chain.

For one curve selection, poly line comes from AutoCad mainly. If curve in poly line is line segment, it can be loaded normally in EB, so user can regard it as curve. If there exists arc in poly line, it will be regarded as block when being loaded in EB, such poly line can not be curve.

(3) For rotation, user can select graph, specify base point, then select curve, and determine direction of newly produced graph, then a group of identic graph will be arrayed at different position of the curve evenly, the graph directions are different.

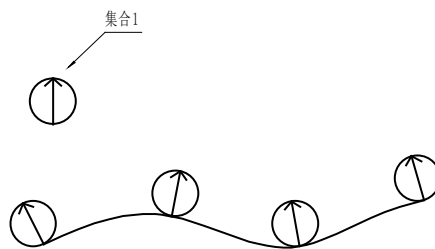
For irrational, user can select graph, specify base point, then select curve, then a group of identic graph will be arrayed at different position of the curve evenly.

(4)Array number

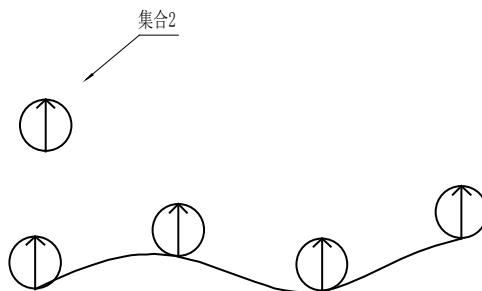
It means number of newly produced graph.

Especially, when the curve is not close, new graph will be produced at two ends of it, the total number will keep the same.

Example, the figure below is example of curve array, among which, figure (a) expresses single bus selection, and rotate is selected, the array number is 4.



pic5-19(a)



pic5-19(b)



5.3.13 Break

【Command】 Break

【Icon】 

【Definition】 Break one specified curve into two parts at specified point.

【Step】

User can implement command Break in the forms of : click Break button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Break.

(1) Select curve to be broken as per hint, the selected one will be highlighted. It will then hint "select breaking point". User should select the point carefully, click left key when it is selected. Then the original curve is broken into two individual part.

(2) In order to get accurate graph, user should use intellectual point, grid point, navigating point, or tool point menu introduced in chapter 11 to select breaking point on curves to be broken. It is allowed to set the point outside curves, the regulation is as follows:

1) If the element to be broken is line, the system will draw a vertical line from the selected pointed to the line to be broken. The foot is break point.

2) If the element to be broken is arc or circle, the system will draw a line from center point to setting point, the intersection point of line and arc is break point.

Example one: Part a curve

A curve can be parted by the operation of break in menu of modify and partition point in drawing toolbar.

Example two: Select a point outside curve



5.3.14 Stretch

【Command】 Stretch

【Icon】 

【Definition】 Keep the curve original trend direction, then Stretch or shorten them,

【Step】

User can implement command Stretch in the forms of : click Stretch button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Stretch. There are two modes introduced as follows.

5.3.14.1 Single select

【Definition】 Stretch or reduce curve under the condition of keeping it having original trend

【Step】

Implement the command stretch, then the immediate menu will pop up.

- (1) Click 1: of immediate menu , and choose single select.
- (2) Select one end of line or arc to be stretched as per hint, and click left key, then the curve disappears, when moving mouse again, a stretched curve is dragged by cursor. When the stretched curve reaches specified position, click left key. A stretched line is displayed. User can also shorten curves by the same operation.
- (3) User can drag cursor or input coordinate value to stretch. To line, length can be inputted, to arc, user can click 2: of immediate menu, switch stretching arc length, stretching angle, stretching radius and free stretch. When stretching arc length and angle, the circle center and radius will keep the same, but the center angle is changed, new center angle can be inputted via keyboard. When stretching radius, circle center and center angle will be the same, but the radius will be changed, user can input new radius value via keyboard. For free stretch, central angle, center and radius can be changed. Stretch value can be determined by absolute or increment in option 3: of immediate menu.
- (4) The command can be operated repeatedly, click right key to finish.
- (5) In addition, user can use the function of pinch point edit introduced in section 5.3.2 to stretch.

5.3.14.2 Select by window

【Definition】 Move specified part of graph within window to stretch graph.

【Step】

Implement the command Stretch, the immediate

- (1) Click 1: of immediate menu , and choose select by window.
- (2) Specify first point in series of curves window, it will hint "another point", user should drag mouse and input another point, then a window is formed. It should be noted that, user should select from right to left for window selection, i.e. the position of second point must be in the left side of first point, this point is very important. Otherwise, full selection of series of curves can be realized.
- (3) When the selection is finished, select offset in option 2 of immediate menu, it will hint "offset or position point along X-axis direction and Y-axis direction". Then drag mouse or input a position point via keyboard, series of curves in window are stretched. It should be noted that, offset of X-axis direction and Y-axis direction means offset relating to reference point, and reference point is given automatically. Generally speaking, reference point of line is mid point, to circle, arc or rectangle , the reference point is center . To combined entity, spline, the reference point is center of

rectangle included in that entity. In the figure below, Select by window, include rectangle , reference point etc. are introduced.

(4) Click 2: offset of immediate menu ,and switch to 2: two point, it will hint "first point", select series of curves by window and specify one point, it will hint "second point" , then drag mouse to stretch series of curves. When second point is determined, stretching is finished. As shown in the following figure, stretching length and direction are decided by two determined points.

(5) There are three options in 3: of immediate menu, they are non-orthogonal, X direction, Y direction, by which user can limit position of stretching point. For non-orthogonal, the direction is unlimited, user can input value or select by cursor to determine. For X direction, user can stretch horizontally, for Y direction, user can stretch vertically.

5.3.15 Explode

【Command】 explode

【Icon】 

【Definition】 Polyline, mark, pattern filling or block reference can be combined and converted to single element.



Explode combined object by Polyline, mark, pattern filling or block reference etc, then convert to single element. For example, explode polyline to simple line segment and arc. When exploding block reference or related mark, it will become object duplicate that can be formed to block or mark.

When mark or pattern filling is exploded, all relative relation will lose, the mark or filling will be replaced by single objected, such as line, text, dot, and 2D object.

When exploding polyline, all related width information will lose. All straight line and arc will be placed along the center line of the original polyline. If user explodes block that includes polyline, then you should explode polyline individually. If explodes a circle, the width will be 0.

For most object, the explode result is invisible.

【Step】

User can implement command Explode in the forms of : click Explode button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Explode.

Implement the command Explode, then select object to be exploded and confirm.

5.4 Property edit

CAXA DRAFT supplies property toolbar, in order that user can edit object layer, color, linear, line weight etc, most of which are base property.

5.4.1 Property option panel



【Command】 properties

【Icon】 

【Definition】 Edit object property by property option panel.

Base property includes layer, color, linear, line width, own property includes circle center, radius, diameter etc.

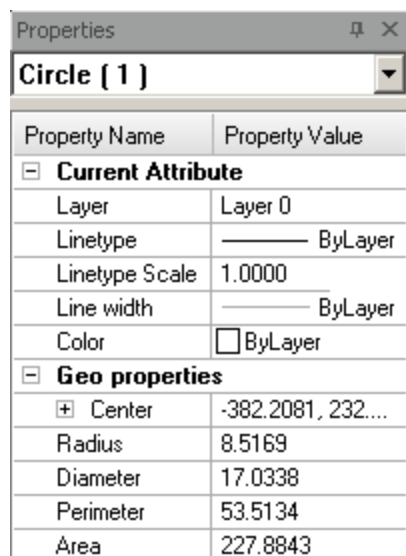
【Step】

User can implement command Property in the forms of : click Property button in the main menu of Modify, click the button of  in the commonly-used Modify panel from the option card , click the button of  in the toolbar of Modify, implement command of Property, select object then click right key to select Property.

Once property command is implemented, the property option panel is open, select object that needs to edit, then modify in the option panel. When the property option panel is open, user can select and edit object directly.

Or select object needs to edit, then implement command of property.

Following is Circle property edition status.



pic5-20 Property option panel

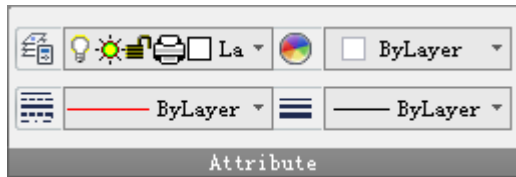
Use the command propertiesclose to close property option panel, or you can execute one more time by other open modes.

5.4.2 Property tool bar

CAXA DRAFT supplies property toolbar, in order that user can edit object layer, color, linear, line weight etc. as shown in following figure.



Property function area in Common function option card, as shown in following figure.



Select object first, then select corresponding property from the tool bar of property or function panel.

5.4.3 Match

【Command】 match

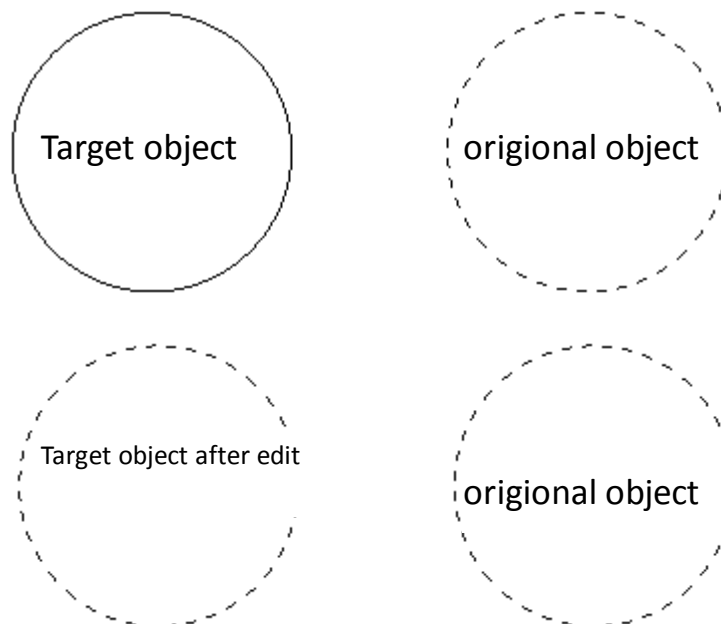
【Icon】

【Definition】 Copy some feature of one object to other object.

【Step】

User can implement command Match in the forms of : click Match button in the main menu of Modify, click the button of in the commonly-used Modify panel from the option card , click the button of in the toolbar of Modify, implement command of Match.

Implement command Match, select original object as per prompt, then select aiming object that needs to modify. Following is an example of Match.



Match

By matching, user can modify layer, color, linear, line weight or other own properties, such text, mark, etc.

Chapter 6 Dimension

Dimension is very import to drawing, the image size and other information can be expressed by dimension.

The dimension function of CAXA Draft is rich and smart, which includes size dimension, coordinate dimension, text dimension, engineering dimension etc. All these dimension can be edited and set parameter to meet request under various condition.

6.1 Summary

Dimension in CAXA Draft is divided into several groups, they are:

- (1) Size dimension, including linear dimension, diameter dimension, radius dimension, angle dimension, arc dimension etc, among which , linear dimension is divided into several kinds according to dimension form, they are horizontal dimension, vertical dimension, parallel dimension, Basic dimension and continuous dimension.
- (2) Coordinate dimension, including origin dimension, quick dimension, free dimension, alignment dimension, hole position dimension, lead-out dimension, auto-list.
- (3) Text dimension, including text, lead-out note, technique requirement.

6.2 Size dimension

【Command】 dim

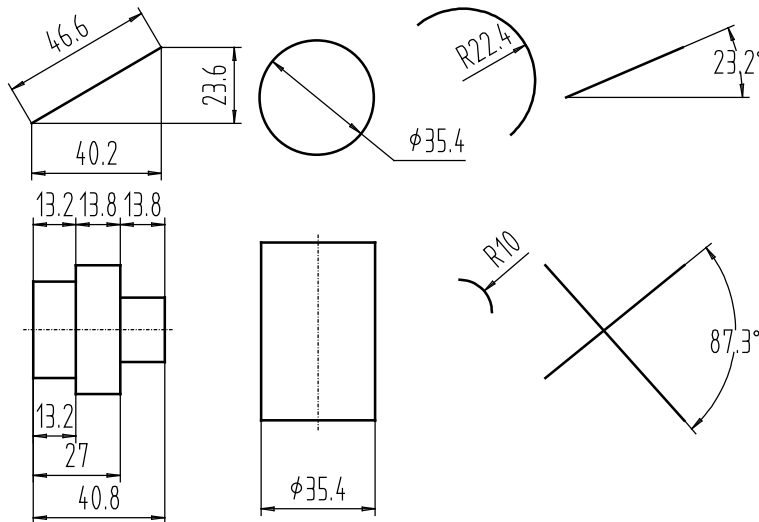
【Icon】 

【Definition】 Add dimension to current image

Size dimension includes Basic dimension, datum dimension, continuous dimension, three point angle dimension, angle continuous dimension, semi dimension, major arc dimension, radial dimension, taper dimension, curvature radius dimension. All these dimension can be implement by the command Dimension, and switch options in the immediate menu, or execute it individually.



Click command Dimension, an immediate menu will pop up, in which switch options to execute the above commands.

Following is an example of common dimension.



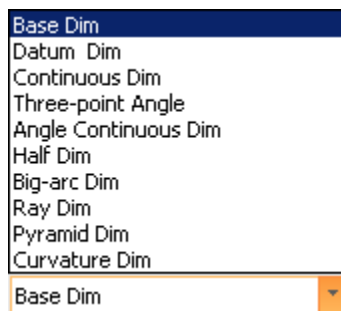
Pic6-1 dimensions example

【Step】

User can implement size dimension in forms of : click button dim in the main menu of dimension, click  in the tool bar of Dimension, click the button of  in the dimension panel from the option card, implement command of dim.

Click command Symexchange, following dialog box will pop up.

Size dimension function is realized via immediate menu, click command, following immediate menu will pop up.



Pic6-2 quick menu of dimensions

Then click option **【1.】** in the above immediate menu, select dimension mode, then select object to be dimension. Following is detail introduction of size dimension.

6.2.1 Basic dimension

【Command】 powerdim

【Icon】 

【Definition】 Quick create linear dimension, diameter dimension, radius dimension, angle dimension or other basic dimension.

There are a lot of size dimension types, it can judge intellectually the dimension type according to selected object.

【Step】

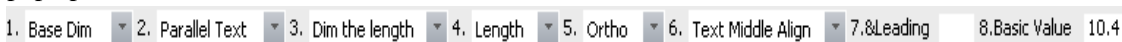
User can implement basic dimension in forms of : click button powerdim in the sub-menu of size dimension, or click command powerdim directly.

Click command powerdim, select object to be dimension as per prompt, then defined dimension parameter and position. The definition and operation method is different from that of individual object and two objects. Following is the detail.

6.2.1.1 Dimension individual object

(1) Dimension for straight line

Click command Powerdim, then select line, a dimension preview prompt and a dialog box will pop up.



Pic6-3 quick menu of dimension line

Parameter in immediate menu is explained as follows:

- 【1: 】 other size dimension modes can be selected here.

【2: text parallel】 Set the text dimension position related to that of dimension line the text can be set to parallel or horizontal or ISO standard.

- Dimension of line length

In the immediate menu, if Dimension length is selected for option 【3: 】 , and length is selected for option 【4: 】 , then the dimension will be the line length.

When option 5 is orthogonal ,the line length in horizontal orientation or vertical orientation will be marked. Switch it to parallel, the line length will be marked.

【5: Orthogonal】 Dimension the line length along with X axis or Y axis, when it is converted to 【5: parallel】 , the dimension line will be parallel to that line. Such dimension is the actual

length of that line.

【6: text center align】 Dimension text is center aligned , when it is converted to 【6: text dragging】 , the dimension text will be moved along with moving cursor.

【7: Prefix】 Add prefix to dimension text, such as “R”or “φ” etc.

【8: Basic dimension】 It is line length, the value in the edit box is default, user can input dimension value in the edit box.

● Line diameter dimension

Switch option 4 to diameter, i.e. mark diameter, the mark mode is the same as that of line length, the difference is that user should add prefix Φ before diameter dimension value.

● Mark angle between line and coordinate axis.

Switch option 3 to mark angle in immediate menu, then angle between line and coordinate is marked. The immediate menu is as above figure.

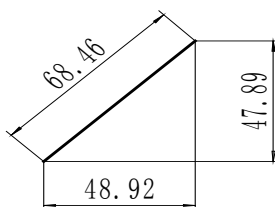


Pic6-4 angle of line and X ray

User can mark angle between line and X axis or Y axis by switching option 4 in immediate menu, angle dimension apex is start point of line nearing selected point.

【5: Degree】 The unit for dimension is degree, user can convert to 5: degree minute second】

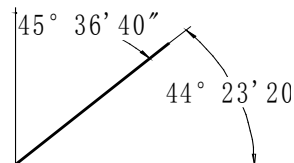
● Followings are example for line dimension.



(a) length



(b) diameter



(c) angle

Pic6-5 example of line dimension

(2) Circle dimension:

Click command Powerdim, select circle needs dimension as per prompt, following immediate menu will pop up.



There are 3 options in 3: of immediate menu, they are :diameter/radius/circle, i.e. user can select three mark modes respectively, they are mark diameter dimension, mark radius dimension,

lead out dimension borderline from circle to mark diameter dimension.

When marking diameter or circle diameter, there will be a prefix ϕ before dimension value. When marking radius, there will be a prefix R before dimension value.

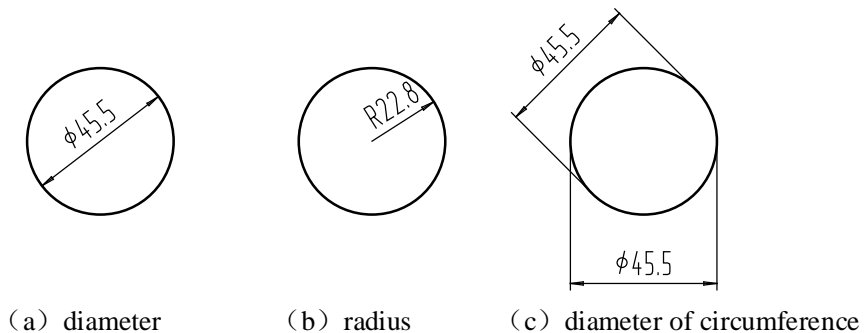
When selecting circle diameter, the immediate will be switched to :

1. Base Dim 2. Parallel Text 3. Diameter of circumference 4. Text Middle Align 5.&Leading %c 6.Dim Value 8.3

In the option【5: orthogonal】 of the immediate menu, when parallel is selected, another option of rotate angle will be added to the immediate menu, by which user can specify dimension line skewness.

Mark positions of dimension line and dimension text are determined dynamically by mark point.

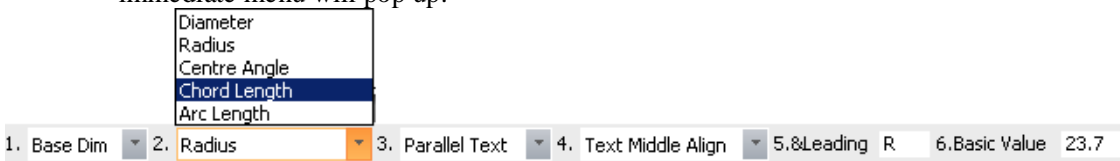
Figure is an example of circle mark.



Pic6-6 example of circle dimension

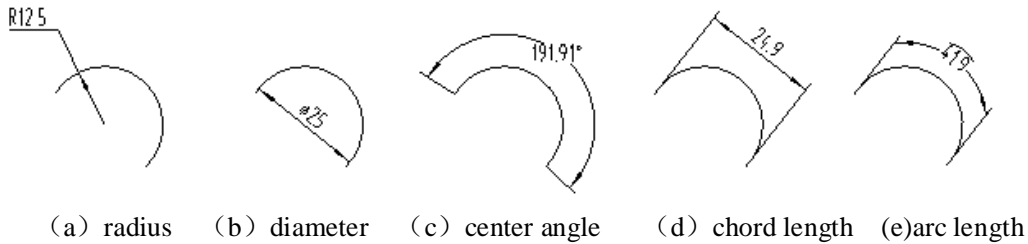
(3) Arc dimension:

Click command Powerdim, select arc needs dimension as per prompt, following immediate menu will pop up.



There are 5 options in 2: of immediate menu, they are radius/diameter/central angle/chord length/arc length, user can select as request to dimension arc. Then specify dimension line position as per prompt, the dimension position can be specified dynamically along with the dimension point.

Figure is an example of circle dimension.



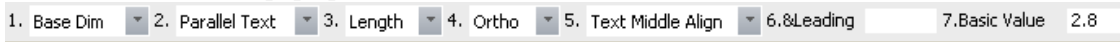
Pic6-7 example of arc dimension

6.2.1.2 Dimension two objects :

(1) Point and point dimension:

To mark distance between two points, user can select point and point respectively, such as screen point, isolated point, or other feature point.

Click command Powerdim, select first point and second point as per prompt, following immediate menu will pop up.



Select each option in the above immediate menu as per drawing requirement, then specify dimension position as per prompt.

(2) Dimension of point and line:

Select point and line respectively.

Click command Powerdim, select first point as per prompt, then select any point in the line, following immediate menu will pop up.



Select each option in the above immediate menu as per drawing requirement, then specify dimension position as per prompt.

(3) Dimension of point and circle or dimension of point and arc:

To mark distance from point to circle center, user should select point and circle or arc respectively. The immediate menu is the same as that of point and point mark. It should be noted that, if user select point at first, this point can be arbitrary point, such as screen point, isolated point, or various controlling point of end point, mid point etc. Yet if user select circle or arc first, the point can't be screen point.

(4) Dimension of circle and circle

To mark distance between two circle centers, user should select circle and circle, or circle and arc, or arc and arc respectively. The immediate menu is the same as that of point and point mark.

(5) Dimension of line and circle or arc

To mark distance from circle center of circle or arc, user should select line and circle or arc respectively

1. Base Dim 2. Parallel Text 3. Text Middle Align 4. Centre 5. Ortho 6.&Leading 7.Dim Value 5.5

There are two options in 3: of immediate menu, they are circle center/tangent point, when circle center is selected, user can mark shortest distance from circle center to line. When tangent point is selected, user can mark distance from tangent point to line.

(6) Dimension of line and line

When two lines are selected, distance or angle between two lines will be marked according to relative location, parallel or non parallel.

If parallel, distance or corresponding diameter between two lines will be marked, the immediate menu will be as follows:

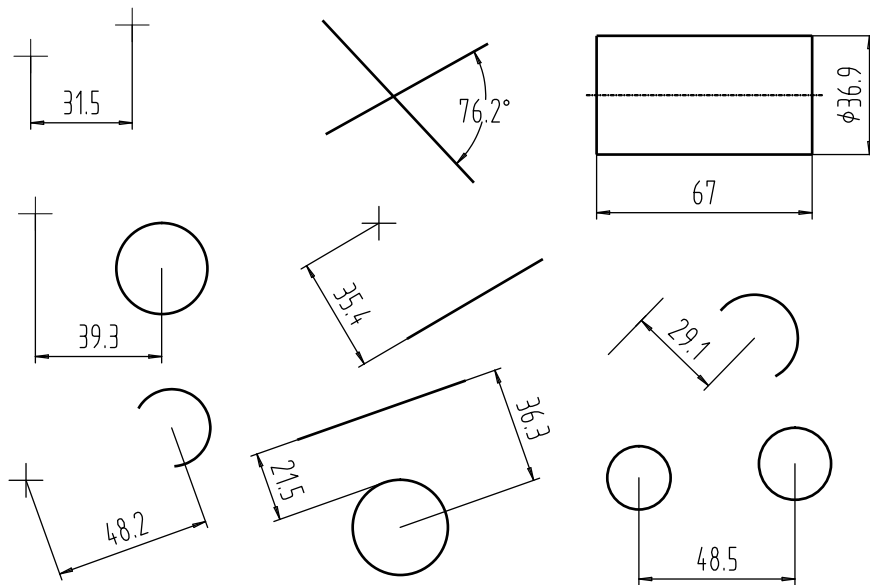
1. Base Dim 2. Parallel Text 3. Length 4. Text Middle Align 5.&Leading 6.Basic Value 12.9

Length/diameter are in 3: of immediate menu, when length is selected, distance between two lines will be marked, when diameter is selected, corresponding diameter of two lines will be marked, and there will be a prefix Φ before dimension value automatically. If the select two lines are intersected, the angle between two lines will be marked, the immediate menu is shown as follows:

1. Base Dim 2. DMS 3.&Leading 4.Basic Value 167°d17'52"

Convert option **【2: degree】** to **【2: degree minute second】** in the immediate menu.

Following is example of selecting two object dimension.



Pic6-8 Example of two object dimension

6.2.2 Base line dimension

【Command】 basdim

【Icon】

【Definition】 Lead out several dimensions from one basic point.

【Step】

User can implement Base line dimension in forms of : click button powerdim in the sub-menu of Base line dimension, or click command dim then select Base line dimension in the immediate menu, or click command basdim directly.

Click command basdim, operate as per instruction, several dimension can be created continually, the operation method for selecting an existing dimension or lead-out is different. Following is the detail.

If a marked linear dimension is selected, the selected linear dimension is regarded as first reference dimension, and dimension reference borderline will be determined as per selected point location, corresponding immediate menu is displayed as follows:

1. Base Dim	2. DMS	3.&Leading	4.Basic Value	167%d17"52"
-------------	--------	------------	---------------	-------------

Meaning of options in immediate menu are introduced as follows:

Option 2:text parallel/text horizontal/ISO standard, it is used for control text orientation.

Option3:Dimension line offset, it means spacing between dimension lines. The default is 10mm, it can be modified.

【4: prefix】, Add prefix to dimension.

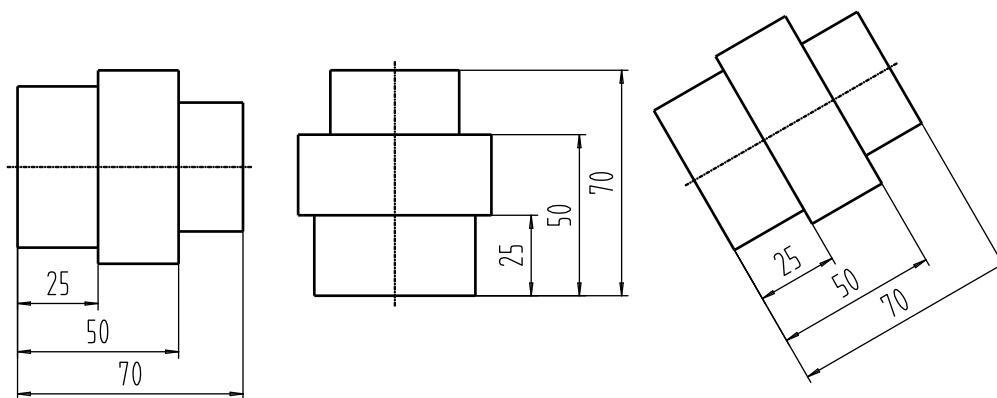
【5: basic dimension】 by default, it is actual dimension value, user can input new value if necessary.

(2) If first lead-out point is selected:

1. Datum Dim 2. Parallel Text 3. Offset of dim line 10 4. &Leading 5. Basic Value Calculate value

it will be lead-out point of dimension reference borderline, select second lead-out point , specify dimension linear position, then the first datum dimension between two leadOut point will be dimensioned, user can select second lead-out point repeatedly as per prompt, then a group of reference dimension can be marked.

In the immediate menu, option 【3: orthogonal】means dimension line is parallel to coordinate axis, it can be converted to 【3: parallel】 , then the dimension line will be parallel to the two point line. Following is the example of Base line dimension.



Pic6-9 example of baseline dimension

6.2.3 Continuous dimension

【Command】 contdim

【Icon】

【Definition】 A serials of end to end dimension.

【Step】

User can implement 【Continuous dimension】 in the form of: click contdim button in the sub-menu of dimension, click command dimension and select contdime in the immediate menu, or

click command contdim directly.

Click command contdim, operate as per instruction, several dimension can be created continually, the operation method for selecting an existing dimension or lead-out is different. Following is the detail.

1) If a dimension linear is selected, the selected linear dimension is regarded as first dimension of continuous dimension, and dimension reference borderline will be determined as per selected point location, user can mark subsequent continuous dimension along the other orientation, and corresponding immediate menu is displayed as follows::



When second lead-out point is given, it will hint "second lead-out point" repeatedly, user can select proper second point by repeated selection to mark a group of continuous dimension.

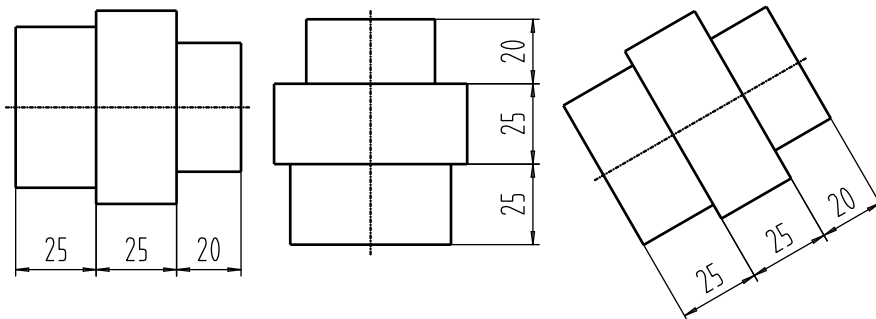
(1) If first lead-out point is selected, it will be lead-out point of dimension reference borderline, the system will hint "select second lead-out point", when another one is selected, the immediate menu will be:



First dimension of two lead-out points at X axis orientation, Y axis orientation, or along two points orientation can be marked, and the system will hint "second lead-out point" repeatedly.

User can select proper second point by repeated selection to mark a group of continuous dimension. Following figure is example of continuous mark.

Continuous dimension example.



Pic6-10 example for Continuous dimension

6.2.4 Three point angle dimension

【Command】 3parcdim

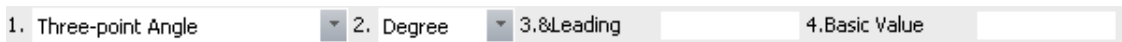
【Icon】 

【Definition】 Create a 3-point angle dimension.

【Step】

User can implement 3parcdim in the form of: click 3parcdim button in the sub-menu of dimension, click command dimension and select 3parcdim in the immediate menu, or click command 3parcdim directly.

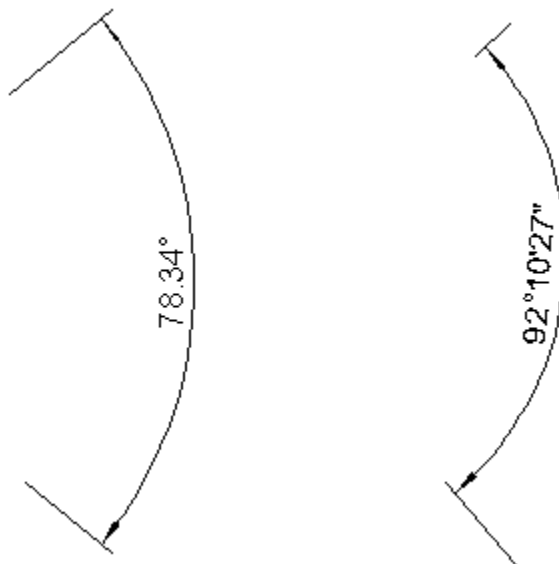
Click command 3parcdim, following immediate menu will pop up.



Click option 【2: degree】 , user can select 【degree,minute,second】 in this option.

User can select **apex**, **first point**, **second point** in turn, and confirm locating point of dimension line at proper position,

The inclination between line passing first lead-out point and peak point, as well as line passing second lead-out point and peak point, is three point angle dimension, following is the example of 3parcdim.



Pic6-11 Three point angle dimension

6.2.5 Angle continuous dimension

【Command】 continuearcdim

【Icon】 

【Definition】 Create a serials of angle dimension continually.

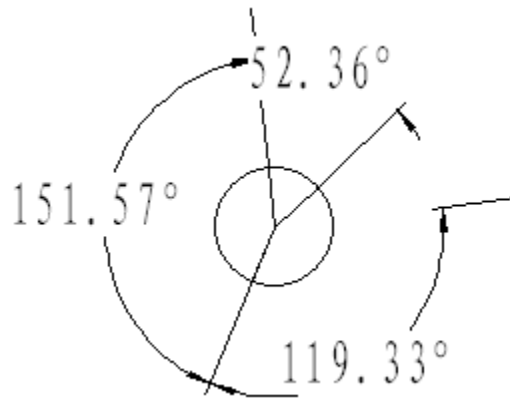
【Step】

User can implement continuearcdim in forms of : click button continuearcdim in the sub-menu of size dimension, or click command dim then select continuearcdim in the immediate menu, or click command continuearcdim directly.

Click command continuearcdim, operate as per prompt, several dimension can be created continually, the operation method for selecting an existing dimension or lead-out is different. Following is the detail.

- (1) If dimension point is selected, it will hint "select first dimension element or angle dimension", "start point", "end point", "dimension line position", "select next element", "dimension line position", user can select as per number of dimension angles, then click right key to get a shortcut menu, select exit button to exit.
- (2) If dimension line is selected, it will hint "select first dimension element or angle dimension", "select another line, "dimension line position", "select next element", "dimension line position", user can select as per number of dimension angles, then click right key to get a shortcut menu, select exit button to exit.

The following figure is after dimension:



6-13 Angle continuous dimension

6.2.6 Half dimension

【Command】 halfdim

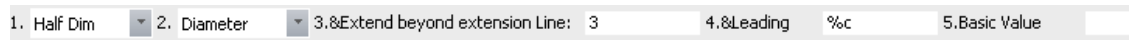
【Icon】

【Definition】 Create half dimension

【Step】

User can implement halfdim in forms of : click button halfdim in the sub-menu of size dimension, or click command dim then select halfdim in the immediate menu, or click command halfdim directly.

Click command halfdim, following immediate menu will pop up.



Click option 【1: halfdim】 to convert to other dimension command.

Click option 【2: diameter】 to convert to diameter or length dimension.

Click option 【3: extend length】 to set extend dimension line length of halfdimension,

Click option 【4: Prefix】 to input prefix for dimension text, when option 【2: 】 is diameter, the symbol %c will be added automatically to diameter.

When all parameter is set in the immediate menu, operate as per prompt.

(1) Select line or first point

When a line is selected, it will hint "select a parallel line or second", if a point is selected, it will then hint "select line or second".

(2) Select second point or line

If two points are selected, twice over distance of first point to second point is dimension value. If a point and a line are selected, Twice over shortest distance from point to line is dimension line. If two parallel lines are selected, twice over distance between two lines is dimension value. Measured value of dimension value is displayed in immediate menu, it can also be inputted, When second element is inputted, it will hint "dimension line location".

(3) Determine dimension line location

Drag dimension line dynamically by moving cursor, and determine dimension line location at proper position, then the mark is finished.

Dimension borderline lead-out point of semi mark is always leaded out from element selected for the second time. The following figure is example of semi mark. Among which figure(a)is mark mode of two selected points. Figure (b) is mark mode of selected point for the first time and selected line for the second time. Figure (c)is mark mode of two selected parallel lines. And figure (d)is mark mode of selected line for the first time and selected point for the second time.

6.2.7 Big arc dimension in measure dimension

【Command】 arcdim

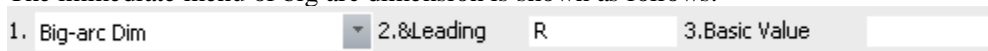
【Icon】 

【Definition】 Create big arc dimension.

【Step】

User can implement arcdim in forms of : click button arcdim in the sub-menu of size dimension, or click command dim then select arcdim in the immediate menu, or click command arcdim directly.

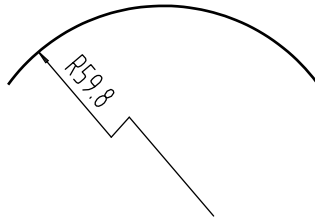
The immediate menu of big arc dimension is shown as follows:



When arc is selected, the arc dimension value is displayed in the immediate menu, user can also input dimension value, the immediate menu will be as the above one:

When first lead-out point, second lead-out point and locating point are specified in turn, a big arc mark is finished.

The figure is an example of big arc mark.



Pic6-15 Big arc dimension in measure dimension

6.2.8 Radial dimension

【Command】 radialdim

【Icon】

【Definition】 Create radial dimension

【Step】

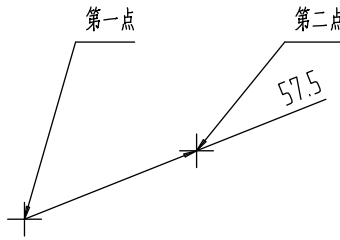
User can implement radialdim in forms of : click button radialdim in the sub-menu of size dimension, or click command dim then select radialdim in the immediate menu, or click command radialdim directly.

The immediate menu of radialdim is shown as follows:



The default dimension value is distance from first point to second point, it can also be inputted.

Drag dimension line dynamically by moving cursor, and determine text locating point at proper position, then radial dimension is finished.



Pic6-16 Radial dimension

6.2.9 Gradient dimension

【Command】 gradientdim

【Icon】

【Definition】 Create taper or gradient dimension.

【Step】

User can implement gradientdim in forms of : click button gradientdim in the sub-menu of size dimension, or click command dim then select gradientdim in the immediate menu, or click command gradientdim directly.

The immediate menu of gradientdim is shown as follows:



Click option 【2: Taper】 to convert to taper or gradient, the default value of gradient is ratio between relative axis height and line length, it will be expressed in the form of 1:X. The default dimension value for taper is double over the value of gradient.

Click option 【3: forward】 to convert to reverse, it is used to adjust symbol direction of taper or gradient.

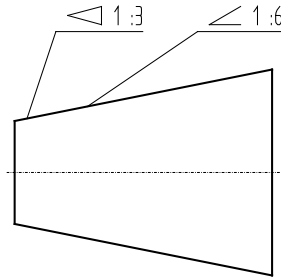
Click option 【4: add down lead】 to control add down lead or not.

Click option 【5: text without frame】 to set add frame or not for the dimension text.

Confirm all parameter in the immediate menu.

When axial line is selected, it will hint "select line", after the line is selected, default dimension value is shown in immediate menu, it can also be inputted. Then it will hint "locating point".

Drag dimension line dynamically by moving cursor, and input text locating point at proper position, then taper mark is finished.



Pic6-17 Gradient dimension

6.2.10 Curvature radius dimension

【Command】 curvradiusdim

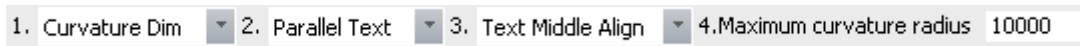
【Icon】

【Definition】 Dimension curvature radius for spline.

【Step】

User can implement curvradiusdim in forms of : click button curvradiusdim in the sub-menu of size dimension, or click command dim then select curvradiusdim in the immediate menu, or click command curvradiusdim directly.

The immediate menu of curvradiusdim is shown as follows:



Click option 【2:】 to select 【text horizontal】 or 【text parallel】 .

Click option 【3:】 to select 【text center align】 or 【text dragging】 .

Click option 【4:】 to set dimension prefix.

Click option 【5:】 to set basic dimension.

When all parameter in the immediate menu is confirmed, select spline needs dimension, then specify dimension line position.

Following is the example of curvature radius dimension.



Pic6-18 Curvature radius dimension

6.3 Coordinate dimension

【Command】 dimco



【Icon】 

【Definition】 Dimension coordinate origin, select dimension value of point or center point of circle.

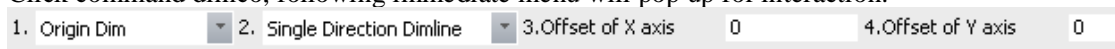
Coordinate dimension includes origin dimension, quick dimension, free dimension, alignment dimension, hole site dimension, automatic list in coordinate dimension. All these commands can be executed via command dimco, then convert in the immediate menu. Or implement such command respectively.

Click command dimco, an immediate menu will pop up, in which user can convert to the above mentioned command.

【Step】

User can implement coordinate dimension in forms of : click button dimco in the main menu of dimension, or click command  in the dimension panel of dimension option card, click button  in the tool bar of size, or click command dimco directly.

Click command dimco, following immediate menu will pop up for interaction.



Pic6-27 immediate menu for Coordinate dimension

Click option 【1.】 in the immediate menu to select dimension mode, then select object to dimension. Following is concrete introduction of those dimensions.

6.3.1 Origin dimension

【Command】 origindim

【Icon】 

【Definition】 Dimension X value and Y value for the origin in current coordinates.

【Step】

User can implement origindim in forms of : click button origindim in the sub-menu of coordinate dimension, or click command dimco then select origindim in the immediate menu, or click command c origindim directly.

The immediate menu of origindim is shown as follows:

(1) Input second point or length. Dimension line begins from origin, the second point can determine locating point of mark dimension text.

According to the position of moving cursor, user can determine marking coordinate in X-axis at first or in Y-axis at first. When second point or length is inputted, it will hint "second point or length", if just one coordinate axis is needed to mark, user can press right key or press enter key to finish. If another coordinate axis is needed to mark, user should go on to input second point or length for operation.

(2) The format of origin mark is determined by option in immediate menu. Each option in immediate menu will be introduced as follows:

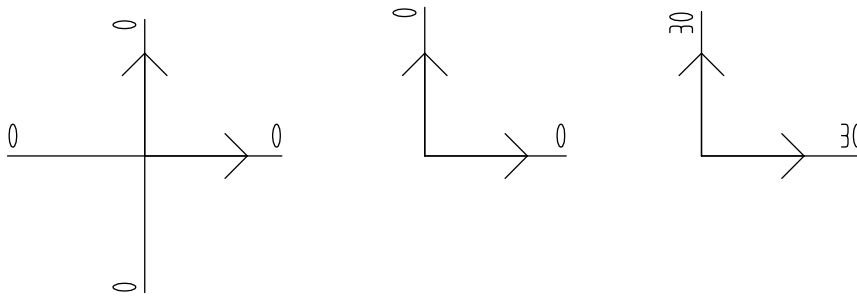
Dimension two-way/Dimension one way: Dimension two-way means dimension begins from origin, which can be extend at double ends. Dimension one-way means dimension begins from origin, which can be extended at one end that is near to dragging point.

Text two-way/Text one way: When dimension two-way is selected, text two-way means dimension value can be marked at double ends of dimension line. Text one-way means dimension value will be marked at one end that is near to dragging point.

X-axis offset: X coordinate value of origin.

Y-axis offset: Y coordinate value of origin.

The following figure is an example of origin dimension.



Pic6-28 example of Origin dimension

6.3.2 Fast dimension

【Command】 fastdim

【Icon】

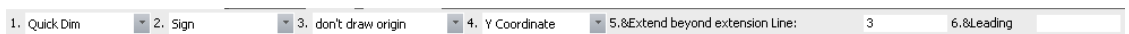
【Definition】 It is used to mark X coordinate value or Y coordinate value of any mark point in current coordinate system.

The dimension format is determined by immediate menu, once mark point is inputted, the mark is finished.

【Step 】

User can implement fastdim in forms of : click button fastdim in the sub-menu of coordinate dimension, or click command dimco then select fastdim in the immediate menu, or click command c fastdim directly.

The immediate menu of fastdim is shown as follows:



Each option in immediate menu will be introduced as follows:

Sign/positive sign: When dimension value is equal to calculated value, select sign, the marked dimension value will take actual value, if it is negative number, user should keep negative sign. If positive sign is selected, the marked dimension value will take absolute value.

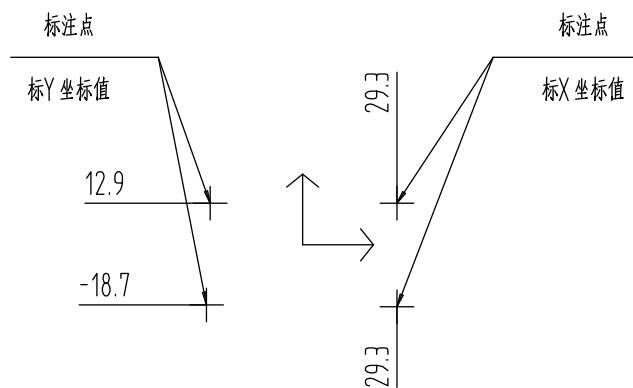
Y coordinate/X coordinate: It is used to control marking Y coordinate value or X coordinate value.

Extending length: It is used to control length of dimension line. Length of dimension line is extending length plus length of text string. The default is 3mm, user can input value by keyboard when shortcut keys **ALT+4** are pressed.

【Prefix】: Add prefix

Dimension value: When it is Y coordinate in option 3 of immediate menu, the default dimension value is Y coordinate value of mark point. Otherwise, it is X coordinate value of mark point. User can use shortcut keys **ALT+5** to input dimension value, sign is useless then.

The following figure is an example of Rapid mark.



Pic6-29 example

6.3.3 Free dimension

【Command】 freedim

【Icon】

【Definition】 It is used to dimension X coordinate value or Y coordinate value of any dimension point in current coordinate system, the dimension format is determined by user.

【Step】

User can implement freedim in forms of : click button freedim in the sub-menu of coordinate dimension, or click command Dimco then select freedim in the immediate menu, or click command c freedim directly.

The immediate menu of freedim is shown as follows:



Each option meaning in the immediate menu.

Negative sign/positive sign: select sign ,the marked dimension value will take actual value, if it is negative number, user should keep negative sign. If positive sign is selected, the marked dimension value will take absolute value.

【Draw/not draw origin coordinates】: Decide to draw origin coordinates or not

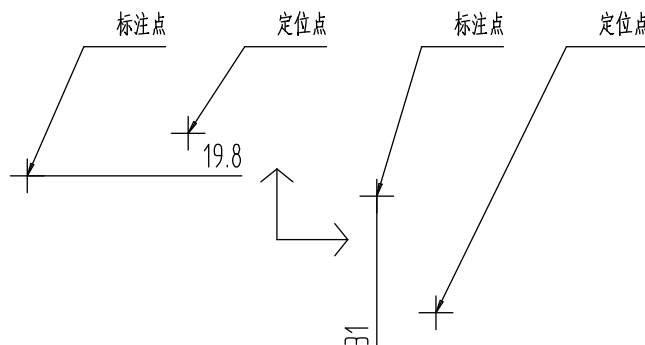
【Prefix】: Set prefix for the dimension.

Dimension value: the default dimension value is X coordinate value or Y coordinate value of mark point. User can use shortcut keys **ALT+3** to input dimension value, sign is useless then.

When parameter in the immediate menu is confirmed, specify dimension point as per prompt first, then the dimension point X coordinate value or Y value will be displayed in the immediate menu (the dragging point will decide its coordinate value).

Then Give locating point, Drag dimension line along X-axis or Y-axis by moving cursor, and press left key at proper position. Other point input modes can be used to specify locating point, such as keyboard, tool point etc.

Following figure is free dimension example.



Pic6-30 example

6.3.4 Alignment dimension

【Command】 aligndim

【Icon】 

【Definition】 As for alignment dimension, it is a group of dimension, the first coordinate dimension is its reference, create continuously a group of dimension with parallel line and aligned text.

【Step】

User can implement aligndim in forms of : click button aligndim in the sub-menu of coordinate dimension, or click command Dimco then select aligndim in the immediate menu, or click command aligndim directly.

The immediate menu of aligndim is shown as follows:



Options in immediate menu will be introduced as follows:

Sign/positive sign: select sign ,the marked dimension value will take actual value, if it is negative number, user should keep negative sign. If positive sign is selected, the marked dimension value will take absolute value.

Dimension line closed/open:It is used to control whether drawing dimension line or not when it is alignment mark.

Arrow closed/open:it will pop up only when dimension line is open, and it is used to control whether draw arrow at one end of dimension line.

【Draw/not draw origin coordinates】: Decide to draw origin coordinates or not

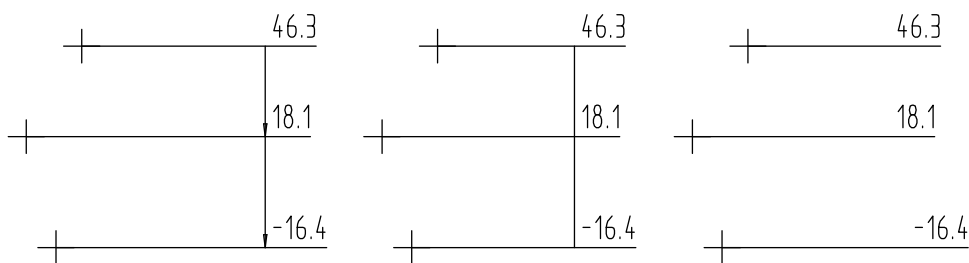
【Prefix】: Set prefix for the dimension.

Dimension value:The default is mark point coordinate value. User can use shortcut keys **ALT+4** when dimension line is closed, or use shortcut keys **ALT+5** when dimension line is open, to input dimension value, sign is useless then.

When all parameter in the immediate menu is confirmed, create first coordinate dimension, The dimension method is the same as that of free mark.

When first coordinate dimension is finished, it will only hint "dimension point" for subsequent coordinate dimension, user can select a series of mark points, then coordinate mark for a group of dimension text alignment is finished.

Alignment mark format is determined by options in immediate menu. When **dimension line** open in option 3 is selected, another option of **arrow closed/arrow open** will be added to immediate menu, as shown in following figure:



Pic6-31 example

6.3.5 Hole site dimension

【Command】 hsdim

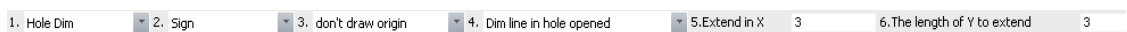
【Icon】

【Definition】 It is to dimension X coordinate value and Y coordinate value of circle center or point.

【Step 】

User can implement hsdim in forms of : click button hsdim in the sub-menu of coordinate dimension, or click command Dimco then select hsdim in the immediate menu, or click command hsdim directly.

The immediate menu of hsdim is shown as follows:



Options in immediate menu will be introduced as follows:

Negative sign/positive sign: select sign, the marked dimension value will take actual value, if it is negative number, user should keep negative sign. If positive sign is selected, the marked dimension value will take absolute value.

Dimension line in hole open / closed: When marking circle center coordinates, it can control whether drawing dimension borderline within circle or not.

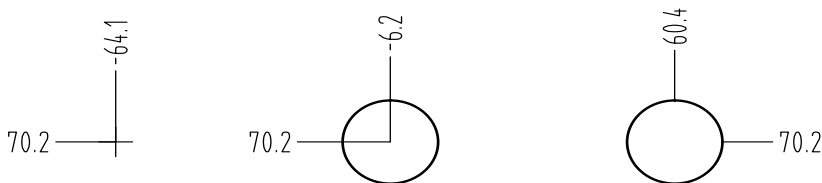
Stretch length along X-axis: Along X-axis orientation, It is used to control length of dimension borderline exceeds outside circle, or extending length of dimension borderline from mark point. The default is 3mm, it can be modified.

Extending length along Y-axis: Along Y-axis orientation, It is used to control length of dimension borderline exceeds outside circle, or extending length of dimension borderline from mark point. The default is 3mm, it can be modified.

【Draw/not draw origin coordinates】: Decide to draw origin coordinates or not

When parameter in the immediate menu is confirmed, select circle or point as per prompt to create hole site dimension.

Following is the example of hole site dimension.



Pic6-32 example

6.3.6 Down-lead dimension

【Command】 downleaddim

【Icon】

【Definition】 When dimension lines or text in coordinate dimension are too dense, it is necessary to lead out its dimension value.

【Step】

User can implement downleaddim in forms of : click button downleaddim in the sub-menu of coordinate dimension, or click command Dimco then select downleaddim in the immediate menu, or click command downleaddim directly.

The immediate menu of downleaddim is shown as follows:



Automatic fold and manual fold are two mark modes of lead-out mark.

(1) Automatic fold

Input dimension point and locating point in turn as per instruction, then the mark is finished. The mark format is controlled by options in immediate menu.

Options in immediate menu will be introduced as follows:

Sign/positive sign: select sign, the marked dimension value will take actual value, if it is negative number, user should keep negative sign. If positive sign is selected, the marked dimension value will take absolute value.

Automatic fold and manual fold: it is used to switch lead-out mark mode.

Clockwise fold/counterclockwise fold: it is used to control bend line direction.

L: it is used to control length of first bend line.

H: it is used to control length of second bend line.

【Draw/not draw origin coordinates】: Decide to draw origin coordinates or not

【Prefix】: Set prefix for the dimension.

Dimension value: The default is mark point coordinate value, user can use shortcut keys **ALT+7** to input dimension value, sign is useless then.

(2) Manual fold

Switch 3: of immediate menu to manual fold, the immediate menu will be as follows:

1. Derivate Dim 2. Sign 3. draw origin 4. Break Handcraft 5.&Leading 6.Basic Value Calculate value

As per system instruction, user can input mark point, first lead-out point, second down-lead point and locating point in turn, then the mark is finished.

Options in immediate menu will be introduced as follows:

Sign/positive sign: select sign, the marked dimension value will take actual value, if it is negative number, user should keep negative sign. If positive sign is selected, the marked dimension value will take absolute value.

Automatic fold / manual fold: it is used to switch lead-out mark mode.

【Draw/not draw origin coordinates】: Decide to draw origin coordinates or not

【Prefix】: Set prefix for the dimension text.

Dimension value: The default is mark point coordinate value. User can use shortcut keys ALT+4 to input dimension value, sign is useless then.

Following is an example of lead-out dimension.



Pic6-33 example

6.3.7 Automatic list in coordinate dimension

【Command】 autolist

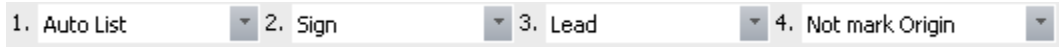
【Icon】

【Definition】 It means coordinates of mark point, circle center or spline interpolation point are listed by table automatically.

【Step】

User can implement autolist in forms of : click button autolist in the sub-menu of coordinate dimension, or click command Dimco then select autolist in the immediate menu, or click command autolist directly.

The immediate menu of autolist is shown as follows:



(1) Mark Spline interpolation point coordinate

If first mark point is inputted, and spline is selected, the immediate menu will be as follows:



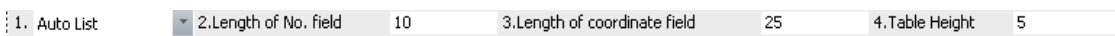
Options in immediate menu will be introduced as follows:

Negative Sign/positive sign: select sign, the marked dimension value will take actual value, if it is negative number, user should keep negative sign. If positive sign is selected, the marked dimension value will take absolute value.

With lead-out line/no lead-out line: it is used to control whether add lead-out line between selected point and symbol.

Symbol: It is mark on lead-out line..The default is A. User can use shortcut keys **ALT+4** to input desired symbol.

When sequence number is inserted, the immediate menu will be as follows:



Pic6-34 immediate menu

Once locating point is inputted, the mark is finished. If total line number in table is more than set line number in immediate menu, user should input locating point of each table respectively.

Options in immediate menu will be introduced as follows:

Sequence length: It is used to control length of sequence number in one column.

Coordinate length: It is used to control length of X coordinate and Y coordinate in the column.

Width:It is used to control width of each row.

Line number:It is used to control exporting line number at best in one time. If total line number is 25, and the set line number is 15, then export two tables, the line number of first table is 15, line number of second table is 10.

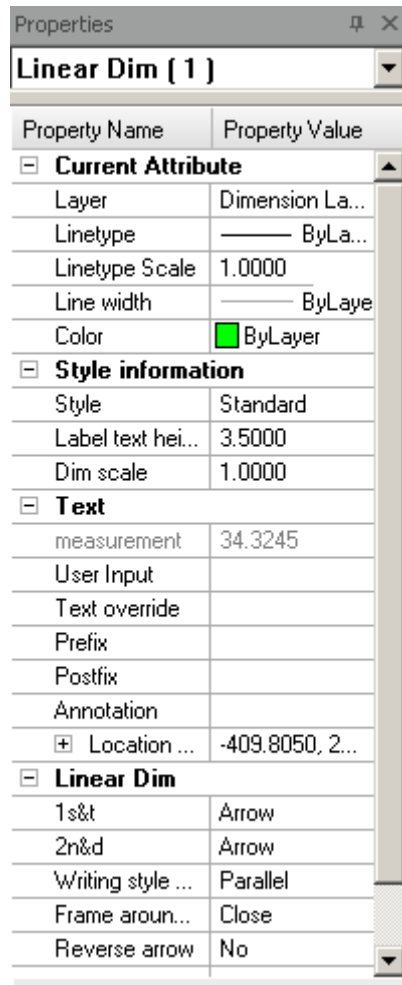
(2) Mark or point and circle center coordinates

When mark point or circle, arc is selected, it will hint "sequence number interpolation point".

When sequence number interpolation is inputted, it will hint "input mark point or select circle or arc" repeatedly.

When a series of mark points are inputted, press right key or press enter, an immediate menu will pop up, as shown in figure 6.26, following operation step is the same as that of selecting spline. The difference is , if there is circle or arc, one column of diameter Φ will be added to the table when the table is exported, but the list box won't be updated.

The following figure is an example of Automatic list.



Pic6-26 properties of linear dim

In the above option panel, user can modify property information, style, text, arrow forms and text frame or other parameters.

6.4 Text dimension

6.4.1 Summary

Text is usually needed to indicate all kinds of information of the drawing, such as note, technical requirement etc.

In CAXA Draft, text dimension includes character, down lead notes, technical requirement etc, following is the details.




6.4.2 Text

【Command】 text

【Icon】 

【Definition】 Create text object in current drawing.

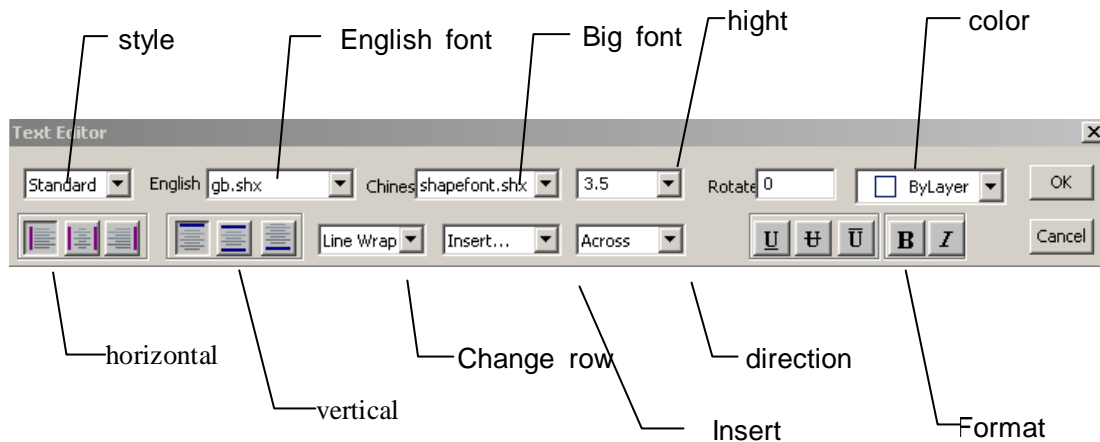
【Step】

User can implement command text in the forms of: Click button  in the main menu of drawing, click button  in the tool bar of drawing , Click button  in the Dimension panel of option card, click command text.

There are three modes to create text, such as two specified points, searching border, select curve. Following is the details.

(1) Specify two point

Click command Text, an immediate menu will pop up, in which select **【specify two points】** , specify first point and second point that forms a rectangle for the text as per prompt, following text editor dialog box will pop up, as shown in following figure.



Pic6-36 text editor

Set text parameter, input text in the dialog box of inputting text, click OK, the parameter meaning of text editor is shown as follows:

【Style】: Click Style to select style for the to be created text, style convert is effective for the whole text paragraph. If new style is applied to current text being edited, character format for font, text height, bold or italic style will be replaced. The underline and color property will be remained

in the new style.


【Font】 On the right side of English or Chinese, user can specify font or change selected character font for the newly input text.


【Angle】 On the right side of inputting box, user can set rotate angle for the newly input text, or change rotate angle for the selected text. For landscape, it means , anticlockwise measured, inclination between the text extending direction and X axis positive direction, For portrait, it means , anticlockwise measured, inclination between the text extending direction and Y axis negative direction , the rotate unit is angle.


【Color】 Specify color for new text, or change the selected text color.


User can specify associated color with the open bylayer or byblock for the text, or select one color from the color list. Or click button other to open dialog box of select color.


【Text height】: Set text height for new text or selected text.

【Bold】: Click  to open or close Bold for new text or selected text. This option is only used to character with TrueTYPE font.

【Italic】: Click button  to open or close italic format for new text or selected text. This option is only used to character with TrueTYPE font.

【Underline】: Click button  to open or close underline for new text or selected text.




【Midline】: Click button  to open or close midline for new text or selected text.。

【Upline】: Click button  to open or close upline for new text or selected text.。

【Text orientation】: Set text writing direction portrait or landscape.

【Insert symbol】: Click button **【insert】** to insert all kinds of special symbols, such as diameter symbol, angle symbol, negative or positive symbol, upper or lower mark, deviation, roughness, dimension special symbol etc.

【Newline】: Set text auto newline, contract text or manual newline. Auto newline means. For auto newline, when the text reaches the right side or bottom at the specified area, the Chinese character, word, number or symbol will be newlined as one unit, so the first or last character can be avoided, and a complete word can be got. For contract text, when specified word parameter exceeds specified area, the system will modify automatically the text height, word width and character space, to make sure that the word will be in the specified area completely. For manual newline, when inputting text, press enter key to newline.

【Vertical alignment】: Click button 、、 to set text vertical alignment, vertical alignment includes top alignment, center alignment, bottom alignment.

【Horizontal alignment】: Click button 、、 to set text horizontal alignment,

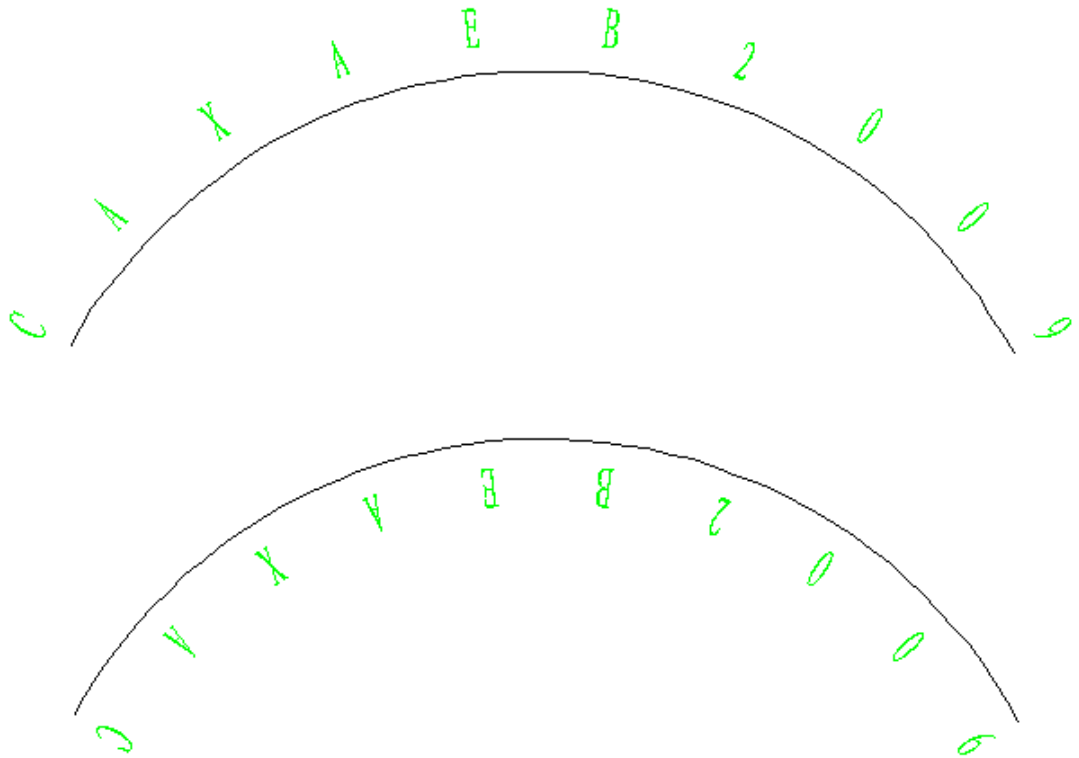
horizontal alignment includes left alignment, horizontal center, right alignment.

(2) Search border

Click command text, select “search border” in the immediate menu, specify one point in the border, then specify border space modulus, text inputting dialog box and text editor will pop up, as shown in following figure, text editor method is the same as that of Specifying two point.

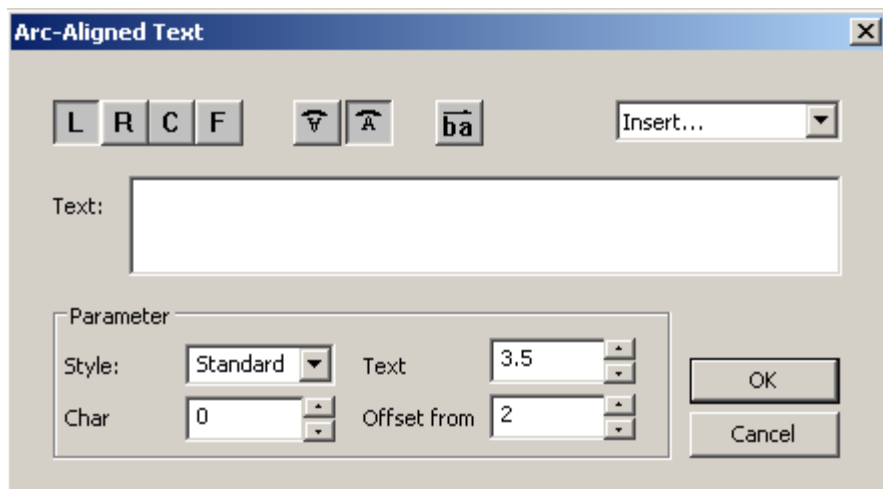
(3) Select curve

Click command Text, and select “ select curve in the immediate menu, then select curve as per prompt, then it will hint the direction of select text dimension, if the direction is different, the dimension result will be different. As shown in following figure.



Pic6-37 curve text

Specify text direction and first point, then select end point, following dialog box of curve text parameter will pop up.



Pic6-38 arc text

【Detail text】 On the right side of the edit box, input text, click option Insert to insert all kinds of symbol,, the parameter meaning is shown as follows.

【Alignment】: Click **L** to set text left alignment, click **R** to set text right alignment, click **C** to set text center alignment, click **F** to set fair alignment.

【Text orientation】: Click button 、 and to set text writing orientation.

【Style】: Select text style.

【Character space】: Set character space for the text.

【Text height】: Set text height.

【Curve offset】: Set offset distance between text and curve.

When above parameter is set, input detail text, click OK to create curve text object.

(4) Insert symbol

In order to input commonly used symbol and special format conveniently, CAXA Draft has regulated some indicating method, all those methods have an initial mark of %.

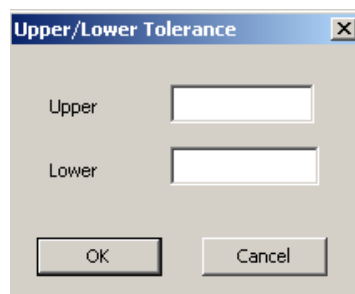
Select "%" in the pulldown list box is equal to inputting "%%" in the edit box, it is mainly used for outputting character string "%p", "%c" etc. For example, the inputted character string is "%p%c%", the output will be "%p%c%".

Select **【φ】** in the pulldown list box is equal to inputting "%c" in the edit box, it is used to output **【φ】**.

Select **【°】** in the pulldown list box is equal to inputting "%d" in the edit box, it is used to output **【°】**.

Select "±" in the pulldown list box is equal to inputting "%p" in the edit box, it is used to output "±".

Then following Deviation dialog box will pop up.



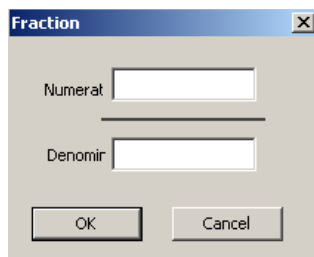
Pic6-39 dialog box

Input upper deviation and lower deviation in corresponding box respectively, then press **enter** key or click button **OK** to finish the input. The upper deviation must be more than lower deviation. It is equal to inputting format "%upper deviation% lower deviation%b". Sign must be added to upper deviation and lower deviation, when it is 0, it doesn't need input. E.g. input 0.005 in the upper deviation edit box, and input -0.004 in the lower deviation edit box, click button **OK**, character string "+0.005%-0.004%b" is added to the current position of edit box, if the character string 12 is in front of that character string, and there is no character string after it, the complete character string will be "12%+0.005%0.004%b". the text is generated as follows, see figure.

12^{+0.005}_{-0.004}

Pic6-40 result

Select option fraction in the pulldown box, a dialog box will pop up, as shown in figure.



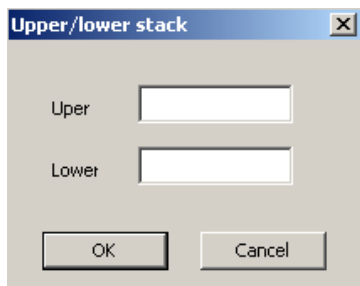
Pic6-41 dialog box

Input molecule in the molecule edit box, and input denominator in the denominator edit box, press enter key or click button **OK** to finish. It is equal to input format "%&molecule/denominator%b". For example, input 1 as molecule and input 6 as denominator in corresponding edit box, click button **OK**, character string "%&1/6%b" will be added to current position of edit box. If the character string 12 is in front of that character string, and there is no character string after it, the complete character string will be "12%&1/6%b", the text is generated as follows, as shown in figure.

$$12\frac{1}{10}$$

Pic6-42 result

Select option "superscript and subscript " in the pulldown box, a dialog box will pop up, as shown in figure.



Pic6-43dialog box

Input superscript and input subscript in corresponding edit box respectively. Then press enter key or click button ok to finish it.

As for the last option **other character**, mapping table will pop up, user can select character to be inserted. To other option, the system will insert corresponding text directly. Also, to get above mentioned special format and symbol, user can input by himself according to regulated format instead of using combination box.

6.4.3 Down-lead note

【Command】ldtext

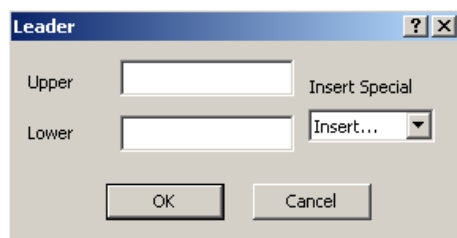
【Icon】

【Definition】It is used to lead out remark, composed of text and outgoing line, the exit point may have arrow. Chinese and western character are both applicable.

【Step】

User can implement command ldtext in the forms of: Click button in the main menu of dimension, click button in the tool bar of Dimension, Click button in the Dimension panel of option card, click command ldtext.

Click command ldtext, following dialog box will pop up.



Pic6-44dialog box

Input upper illustration and lower illustration in corresponding dialog box, if one line needed, user can input upper illustration only. Click button **OK**, and step into next operation, click button **Cancel** to finish such command.

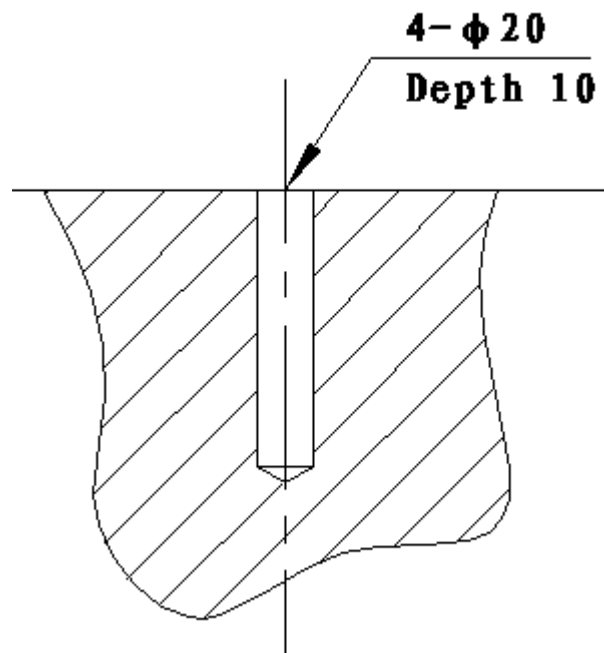
After clicking button **OK**, the following immediate menu will pop up.

1. Default text direction 2.&Extend beyond extension Line: 3

Input first point as per hint, it will then hint "second point".

When second point is inputted, mark of down-lead illustration is finished.

The following figure is an example of lead-out illustration.



Pic6-45 example

6.4.4 Technique requirement library




【Command】 speclib

【Icon】 

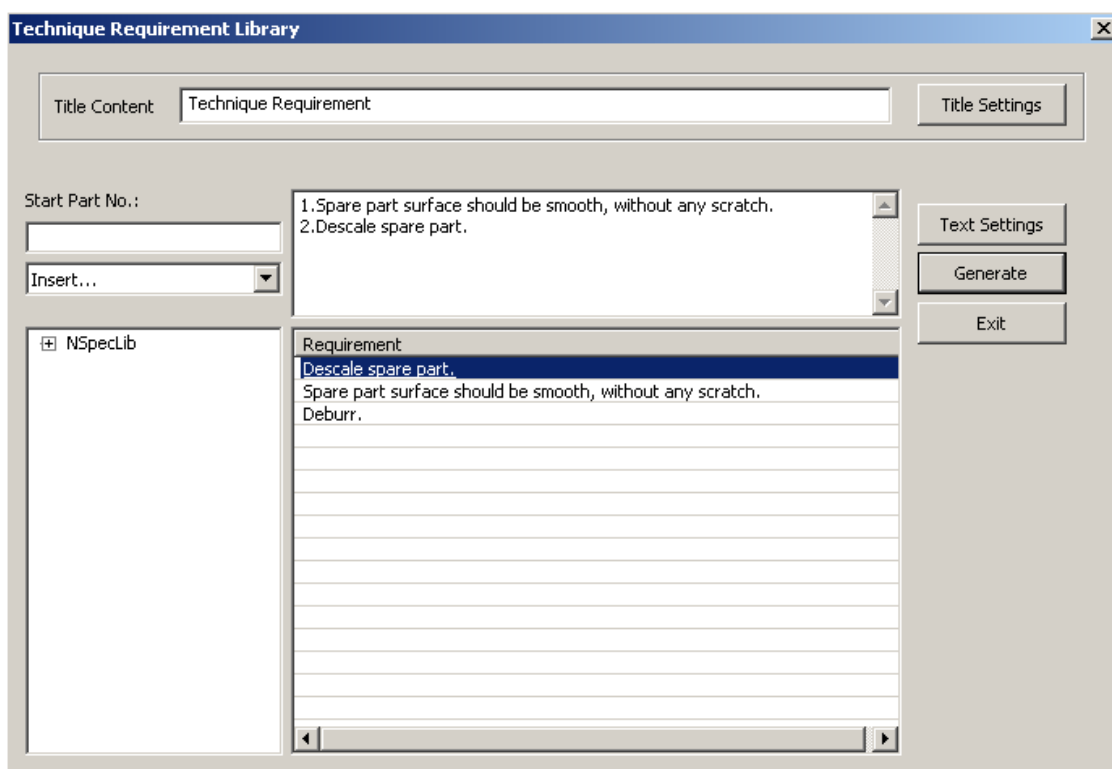
【Definition】 To produce technique requirement text quickly.

CAXA Draft records commonly used Technique requirement text in sort by data-base file, the Technique requirement text can be inserted in the drawing area, and the text can also be added, deleted or modified.

【Step】

User can implement command speclib in the forms of: Click button  in the main menu of dimension, click button  in the tool bar of Dimension, Click button  in the Dimension panel of option card, click command speclib.

Click command speclib, following dialog box will pop up.



Pic6-46 Technique requirement library

Technique requirement types are listed in the left corner box, and all text option of current type are listed in the right bottom corner table. If there is desired text, drag it from table to the edit box at proper position by pressing left key directly, or edit text in the edit box directly.

Click Setup and the dialog box character label parameter set up will pop up, modify parameter to be adopted for Technique requirement text. The usage of combined box at top right corner is the same as that of character label and edit dialog box. When it is edited, click button generate, specify the area for Technique requirement text as per instruction, the system will create Technique requirement text automatically. It is clarified that the font parameter is set for Technique requirement text, but the title Technique requirement can be modified by clicking button setup beside the title.

The management for Technique requirement library is progressed in this dialog box. Select different types in the left bottom box, the corresponding content will be shown in table at right bottom corner. Modify the content in the table directly if necessary. If new text is needed, input in the row with mark * in left side of end table. To delete some text, click left key in left selection area of corresponding row, and press button delete. If a new type is needed, select the last option add new type in the list box , input name of new type, and add text to the table for new type. If delete a type, select it and press button delete, the message box will pop up, choose yes, then the type and its text are deleted from the data base. Double click type name can modify it. When all management work is done, click button exit , and exit dialog box will pop up.

6.4.5 Text finding and replace




【Command】 textoperation

【Icon】 

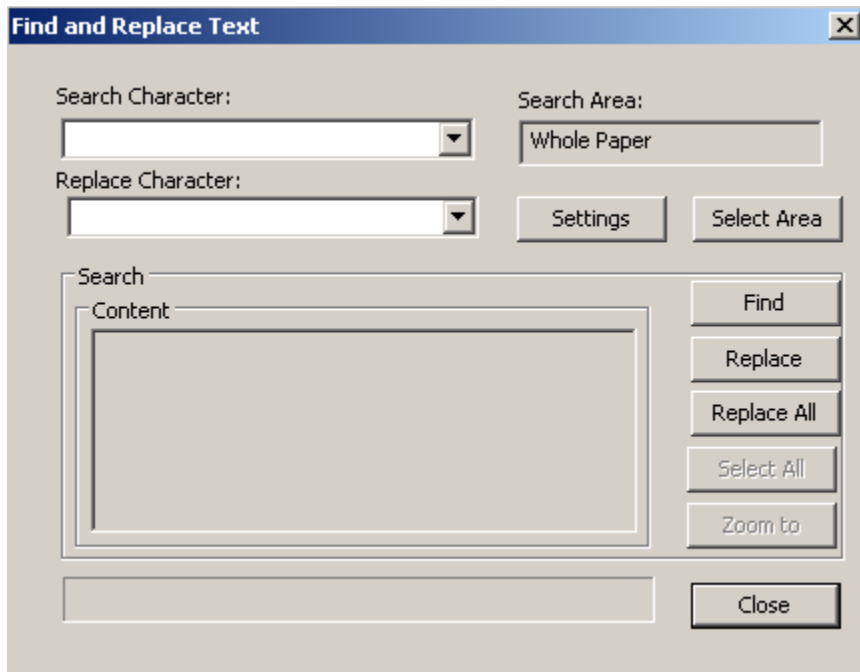
【Definition】 Find and replace text in the current drawing.

Text find and replace can find and replace character in the dimension.

【Step】

User can implement command textoperation in the forms of: Click button  in the main menu of dimension, click button  in the tool bar of Dimension, Click button  in the Dimension panel of option card, click command textoperation.

Click command textoperation, following dialog box will pop up:



Pic6-47 Text finding and replace

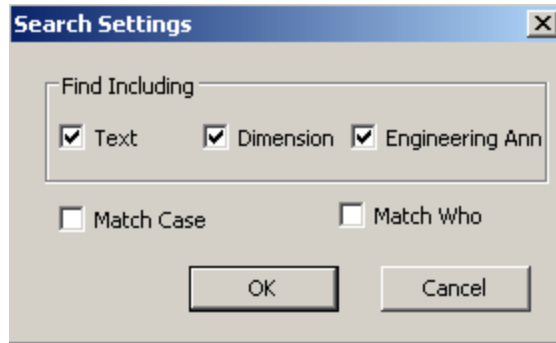
Parameter in the above dialogue box is introduced as follows:

Find text: Input text to be found or replaced

Replace text: Input text for replacement.

Search range: Default search range is the whole graph, user can modify search scope by clicking select range.

Set find: Click **set find**, following dialog box will pop up, user can limit replacing content by four options of including text, including size, including dimension, case-sensitive, including match case and find whole words only. Eg. If select “including text”, then the select scope will include text in the image. But it is unable to find title bar, list or character in border.



Pic6-48 search settings

User can operate as per request by selecting “Find”, “Replace”, “Replace all”, “Select all”, “Display find content”.

6.5 Engineering dimension




6.5.1 Geometrical tolerance

【Command】 fcs

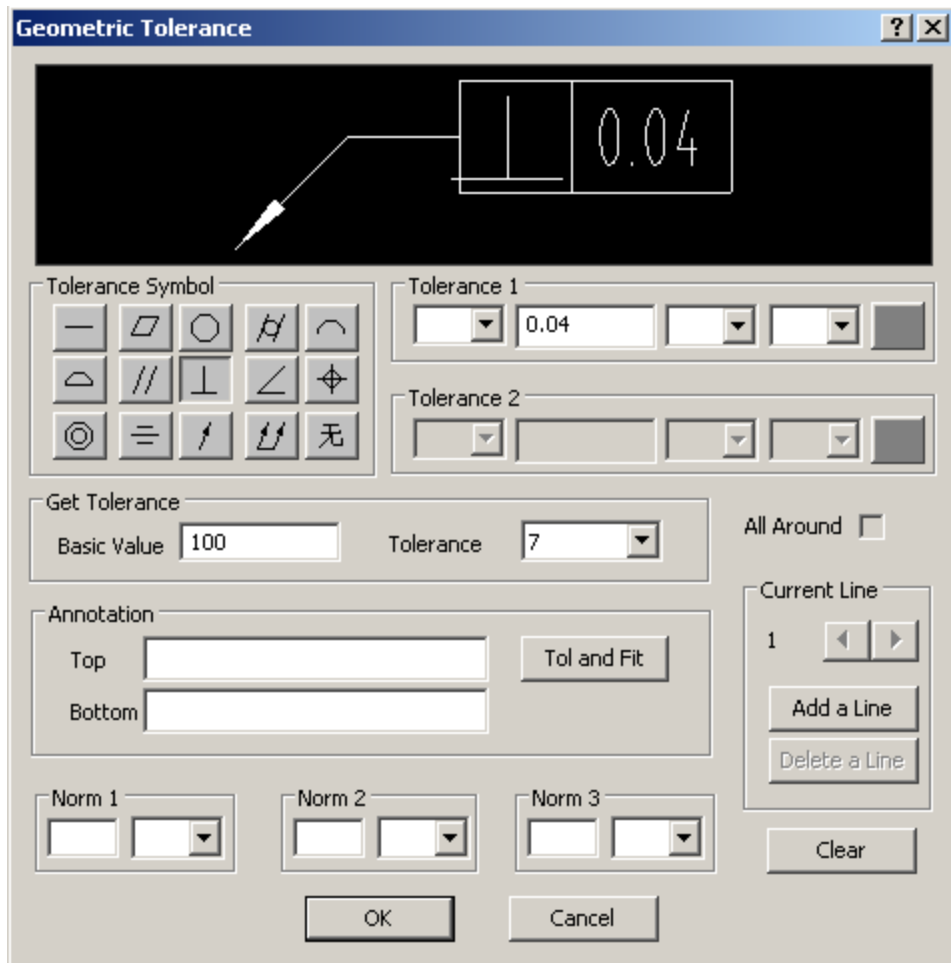
【Icon】 

【Definition】 Dimension geometrical tolerance.

【Step】

User can implement command fcs in the forms of: Click button  in the main menu of dimension, click button  in the tool bar of Dimension, Click button  in the Dimension panel of option card, click command fcs.

Click command fcs., following dialog box will pop up



Pic6-49 tolerance

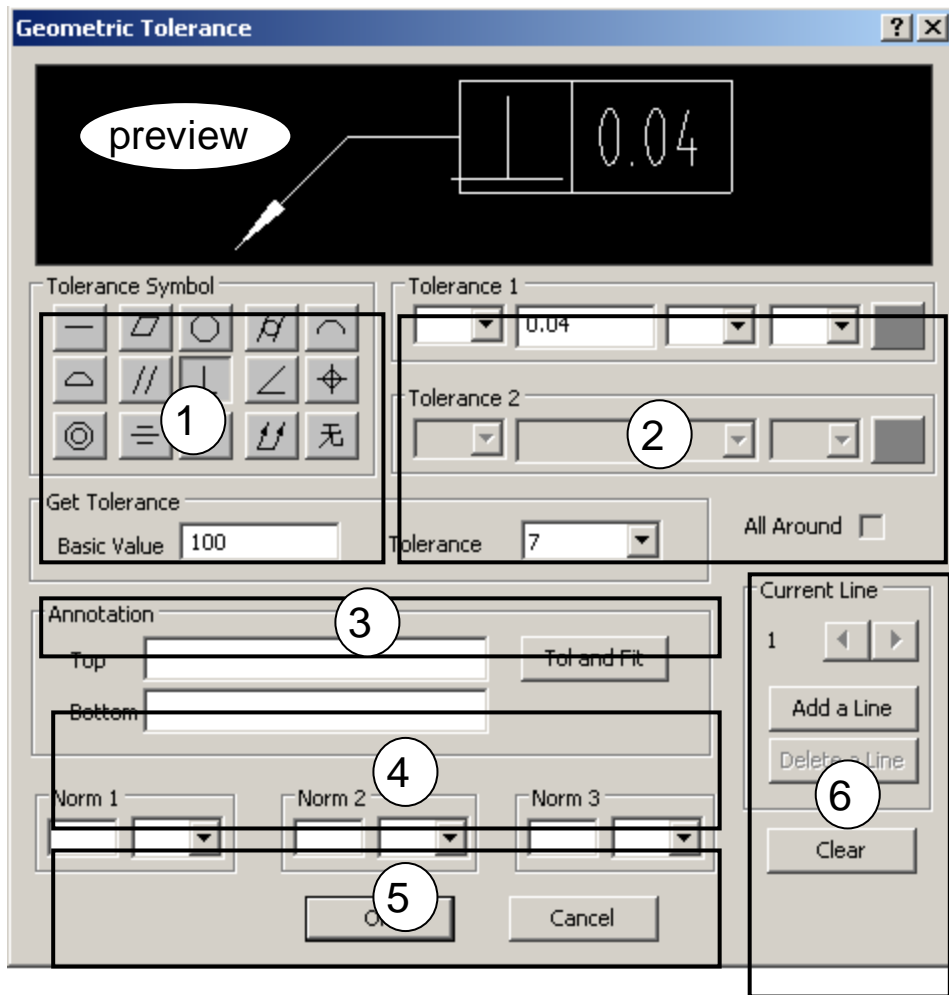
Select tolerance code and set each parameter in the above dialog box, then click OK, select horizontal dimension or vertical dimension in the immediate menu.

Then select dimension element and input turning point of down-lead line as per prompt.

Content in geometrical tolerance dialog box is introduced as follows:

User can fill in each item conveniently by this dialog box, what's more, several lines can be filled in, and line deletion is allowed.

The dialog box is divided into following areas:



Pic6-50 tolerance

(1) **Preview area:** The fill in and layout result will be displayed on the upper dialog box.

(2) **Line parameter input area:** it is at the middle of dialog box, divided into three parts.

1) Geometrical tolerance symbol area: it is leftward, in which **straightness**, **flatness**, **roundness** and so on symbol button are listed ,click one button of them, corresponding graphics will be displayed in the preview area, as shown in figure, in which the **position symbol** is display.

2) Geometrical tolerance value area: it is at the first right line which includes:

- Tolerance value 2: select diameter symbol? or symbol S.
- Value input box: user can input geometrical tolerance value in it, such as 0.1.
- Shape restrict: a list box will pop up, in which user can select option of blank, (-), (+), (>) or (<). Among which, (-) means sunkening towards inside material is allowed, (+) means heaving towards outside material is allowed, (>) means reducing from left to right is allowed, (<) means reducing from right to left, in the figure, suffix (-) is selected.
- Relative principle: a list box will pop up, in which user can select option of blank, (P), (M), (E), (L) or (F). Among which (P) means extending tolerance strip, (M) means maximum entity requirement, (E) means including requirement, (L) means minimum entity requirement, and (F) means nonrigid part free status condition. In the figure, suffix (M) is selected.

Every content after modification will be displayed in the preview area.

3) Datum symbol area: it is at the lower dialog box. There are three groups, user can input datum symbol and select corresponding mark, such as P, M, or E etc. In the figure, A, B, C is selected.

(3) Line management area: it is at the lower right corner of dialog box, which includes three items:

1) Indicate current line number: if mark only one line geometrical tolerance, the indication number is 1, if several lines of geometrical tolerance are needed to mark, user can use this option to indicate current line. And current line can be switched via button on the right.

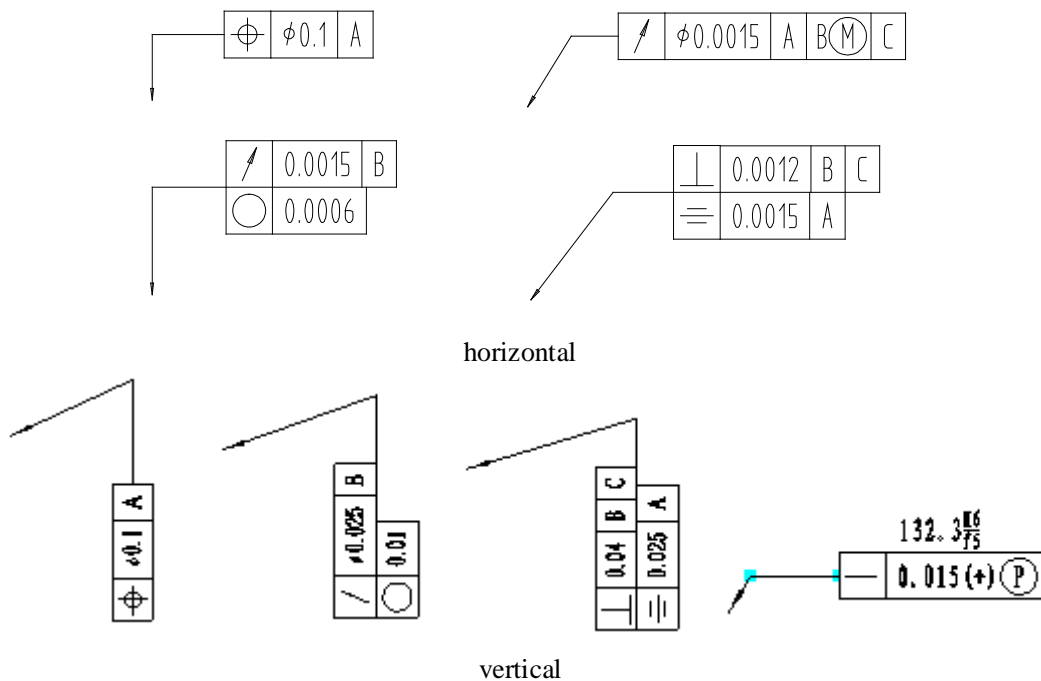
2) Add line: on the basis of one line of geometrical tolerance marked, user can click button add line to mark new line. The method is the same as that of first line mark.

3) Remove line: If click this button, the current line will be removed, the whole geometrical tolerance mark will be adjusted automatically once again.

(4)Tolerance inquiry:When tolerance symbol, tolerance grade are selected, and basic dimension is inputted, the system will give tolerance value automatically.

(5)Annotations:Click button dimension and fit, tolerance input dialog box will pop up, user can add tolerance annotations at place of geometrical tolerance.

Geometrical tolerance mark example:



Pic6-51 example

6.6 Dimension Style

Different drawing standard and environment have different dimension requirement, user can set parametric to control each dimension appearance via Dimension style.

Dimension style is a collection of every kind of dimension setting, by which user can control dimension appearance, such as arrow mode, text position and size tolerance etc. User can create dimension style to specify dimension format quickly, and make sure those dimension style be up to industry standard.

When creating dimension, the setting in current dimension style will be applied.

If the setting in dimension style is modified, all dimension style of the graph will be updated automatically.

Dimension style in CAXA Draft includes: text style, size style, down-lead style, geometric tolerance style. Roughness style, welding symbol style, basic code style, cutting symbol style etc. Following is the detail introduction of them.

6.6.1 Text style





【Command】 textpara

【Icon】 

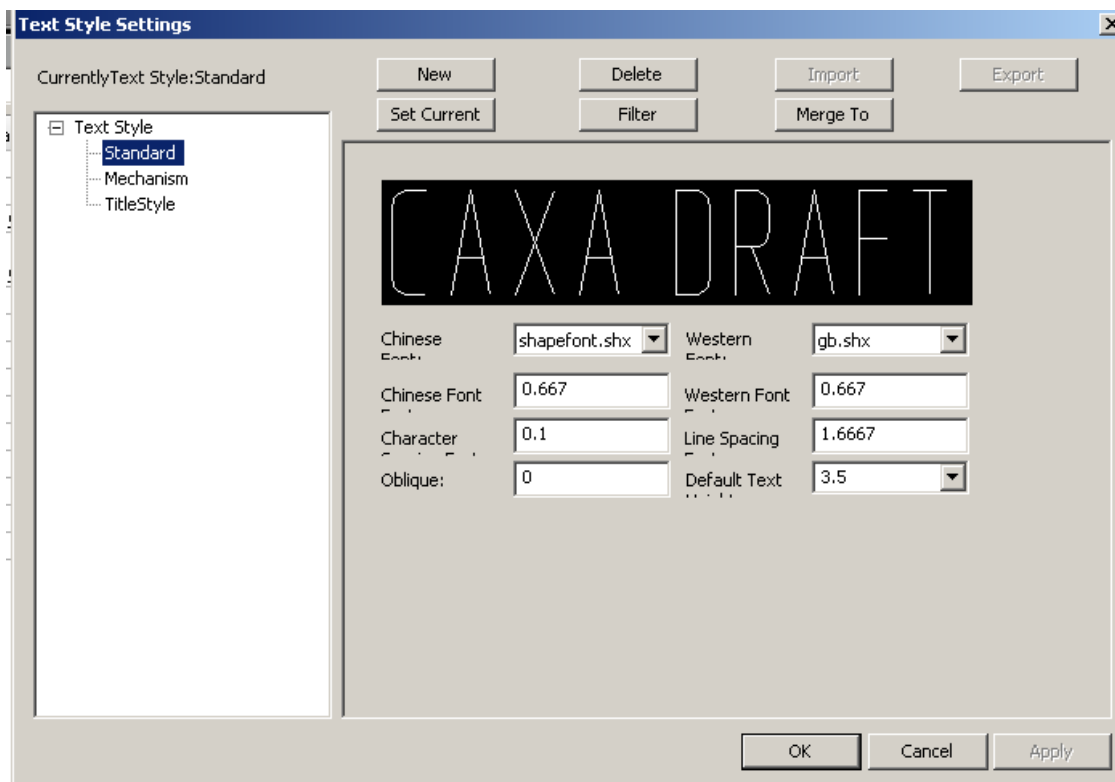
【Definition】 Set parametric for text to control text appearance.

By parametric in text style, user can control text height, text font, text direction, angle etc.

【Step】

User can implement command textpara in the forms of: Click button  in the main menu of Format, click button  in the tool bar of Setting tool , Click button  in the Dimension panel of option card, Click button  in the style management, or click command textpara.

Click command textpara, following dialog box will pop up



Pic6-52 text style dialog

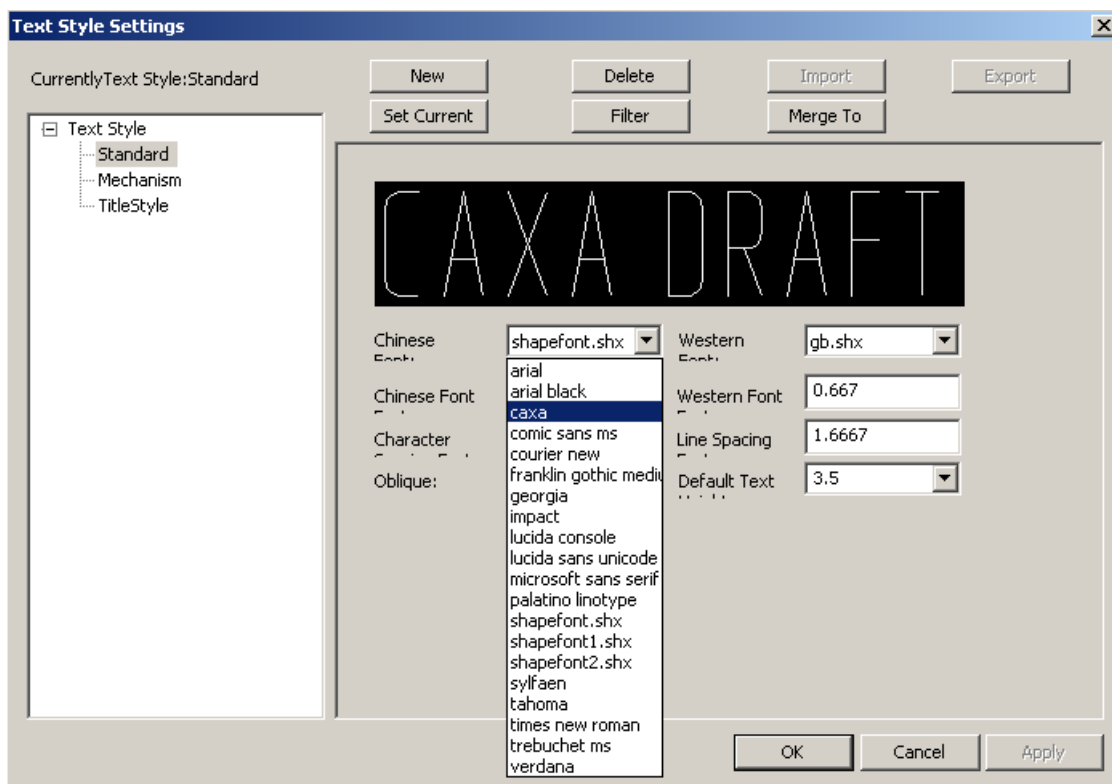
All used text styles in current file are listed in Text style, there is a default style in the system, which is called Standard, this style can't be deleted, but can be edited.

Click New, delete, set current, merge or other button in the text style dialog box, to create ,delete , set current , merge or other operation.

Select one text style, put in parametric for font, width modulus, character space, gradient, text height, and user can preview in the dialog box.

Explanation to text parameter:

- **Chinese font:** Chinese font styles can be selected. In addition to TrueType font supported by Windows, CAXA Draft also supports using monoline font.



Pic6-53 change font

Followings are examples of different text result for the selected fonts.

CAXA EB 2009

fsong-GB2313

CAXA EB 2009

shx font

Pic6-54 different font

Western font: The select mode is the same as that of Chinese, only western is limited, monoline font can also be selected..

Chinese font factor: western font factor: When the factor is 1, character size scale will be consistent with font style described in of TrueType font file, if the factor is not 1, character width will be zoom in or out by corresponding times on such basis.

Character spacing factor: It is the ratio between space and set text height , it should be noted that space is between two neighbor characters in the same row or rank.

Line spacing factor: The ratio between space and set text height, it should be noted that space is between two horizontal neighbor lines.

Column pitch factor: The ratio between space and set text height, it should be noted that space is between two vertical neighbor column.

Rotate angle: When writing horizontally, the counterclockwise angle between positive X-axis and text extending direction of one row. When writing vertically, the counterclockwise angle between negative Y-axis and text extending direction of one column. The angle unit is degree.

Default text height: set default height for words to be created, the text height can be modified when they are created.

When parametric in text style is modified, click OK or Apply to confirm the modification.

6.6.2 Dimension style





【Command】 dimpara

【Icon】 

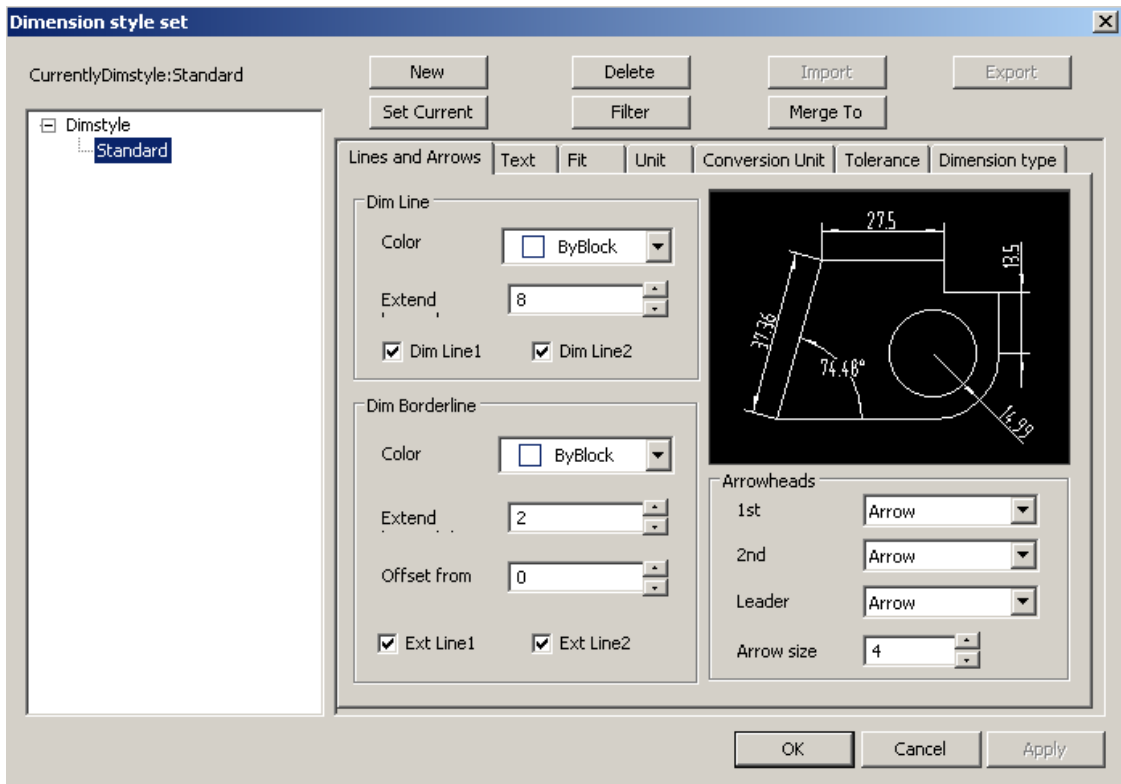
【Definition】 Set parametric for size dimension to control dimension appearance.

By dimension style, user can control its arrow style, text position, dimension tolerance, alignment mode etc.

【Step】

User can implement command dimpara in the forms of: Click button  in the main menu of Format, click button  in the tool bar of Setting tool, Click button  in the Dimension panel of option card, Click button  in the style management, or click command dimpara.

Click command Dimpara, following dialog box will pop up.



Pic6-55 dimension style dialog box

User can create, delete, set current ,merge dimension style in the above dialog box.

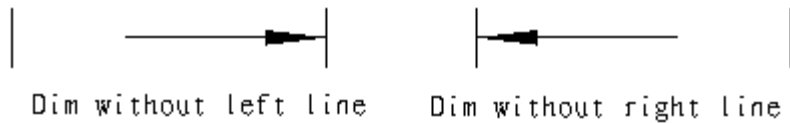
Click button **new** or select an existing dimension style, corresponding dialog box will pop up, in which user can modify options of "line and arrow", "text", "adjust", "unit", and conversion unit, tolerance, dimension style " etc. They will be introduced as follows:

- (1) **Line and Arrows:** user can set color and style for dimension , dimension borderline and arrow, following is the detail introduction.

Dimension line:each parameter that controls dimension line.

- Color :Set dimension line color, the default is **ByBlock**.
- Stretch length:When dimension line is at outside of dimension borderline, distance between the outside of dimension borderline and dimension line is outdrop length. The default is 6mm.

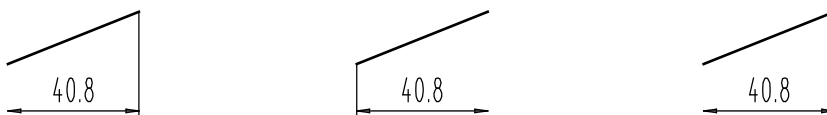
- **Dimension line:** it is divided into left dimension border and right dimension border. User can set on or of for left and right dimension line. The default is on.



Pic6-55 example

Dimension borderline: Parameters that control dimension borderline.

- **Color:** Set color for dimension borderline, the default is **ByBlock**.
- **Lead-out point form:** Set lead-out point form for dimension borderline, user can select **dot**, the default is **NULL**.
- **Exceed dimension line:** dimension borderline extends to the outside of dimension terminal, this distance is extending length, the default is 2.0mm.
- **Start point offset:** The length between dimension borderline and marked element, the default is 0mm.
- Add dash line box to forced mark dimension, user can choose output or non-output when printing.
- **Borderline:** it is divided into left borderline and right borderline, user can set on or of for left and right dimension line. The default is on, as shown in figure 6-61.



Pic6-56 example

- **Arrow relative:** user can set size and mode of dimension arrow. The default mode is arrow, what's more, user can select mode skew line or dot. When marking, the outward or inward arrow can be set according to requirement.

【Leftward arrow】: to control leftward arrow style of dimension line. By default, it is arrow, user can select diagonal, dot, hollow arrow or other shapes.

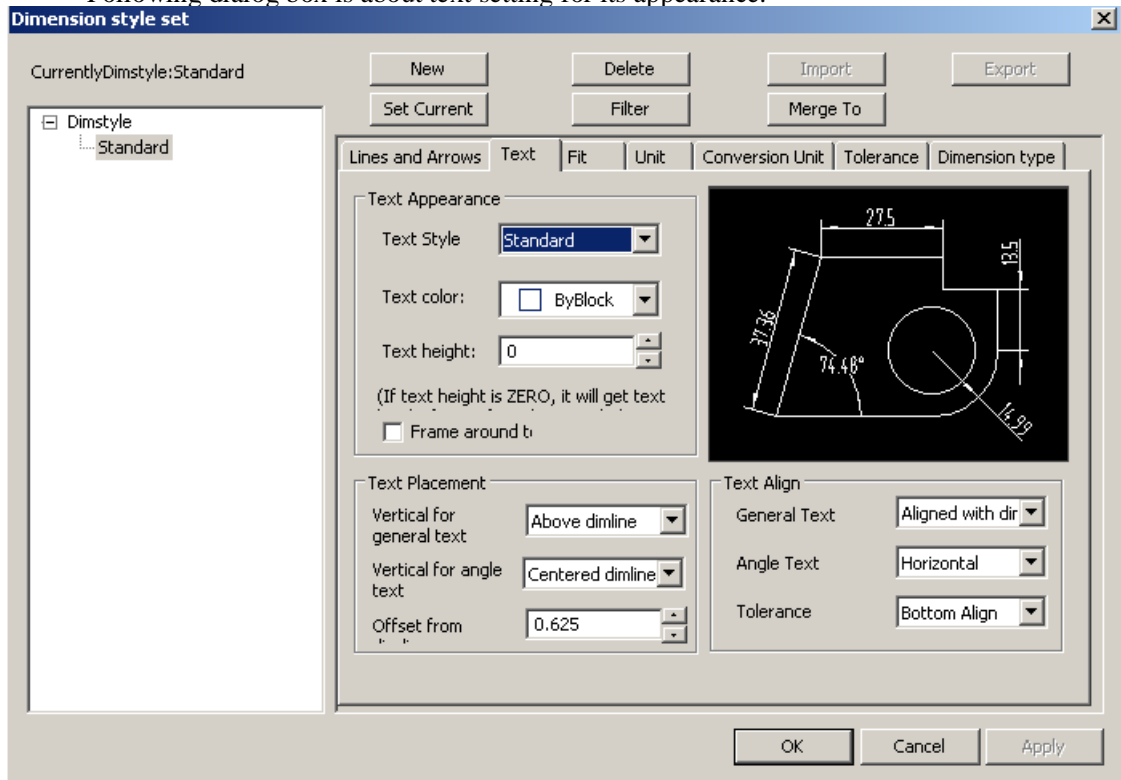
【Rightward arrow】: to control rightward arrow style of dimension line. By default, it is arrow, user can select diagonal, dot, hollow arrow or other shapes.

【Down-lead arrow】: to control down-lead arrow style of dimension line. By default, it is arrow, user can select diagonal , dot, hollow arrow or other shapes.

【Arrow size】: To control arrow size.

(2) 【Text】: Set text appearance, text position, text alignment etc.

●Following dialog box is about text setting for its appearance.

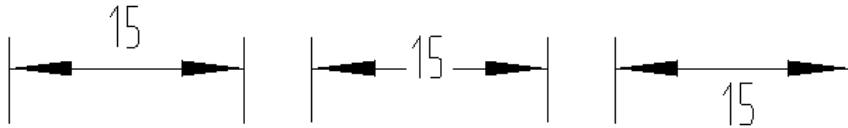


Pic6-57 text settings

- **Text style:** it is relative to text style of software, which will be explained concretely in latter section of text style.
- **Text color:** set font color of text, the default is **ByBlock**.
- **Text height:** control dimension text height, the default is 3.5mm.
- **Drawing text border:** add border to mark text.

Text Vertical position: Control position relations between dimension text and dimension line.

- **Text position:** Control text position relative to dimension line. For Text vertical position, click pulldown arrow on the right, the following text position will pop up: above dimension line, at the mid of dimension line, under dimension line, as shown in figure.



Pic6-58 example

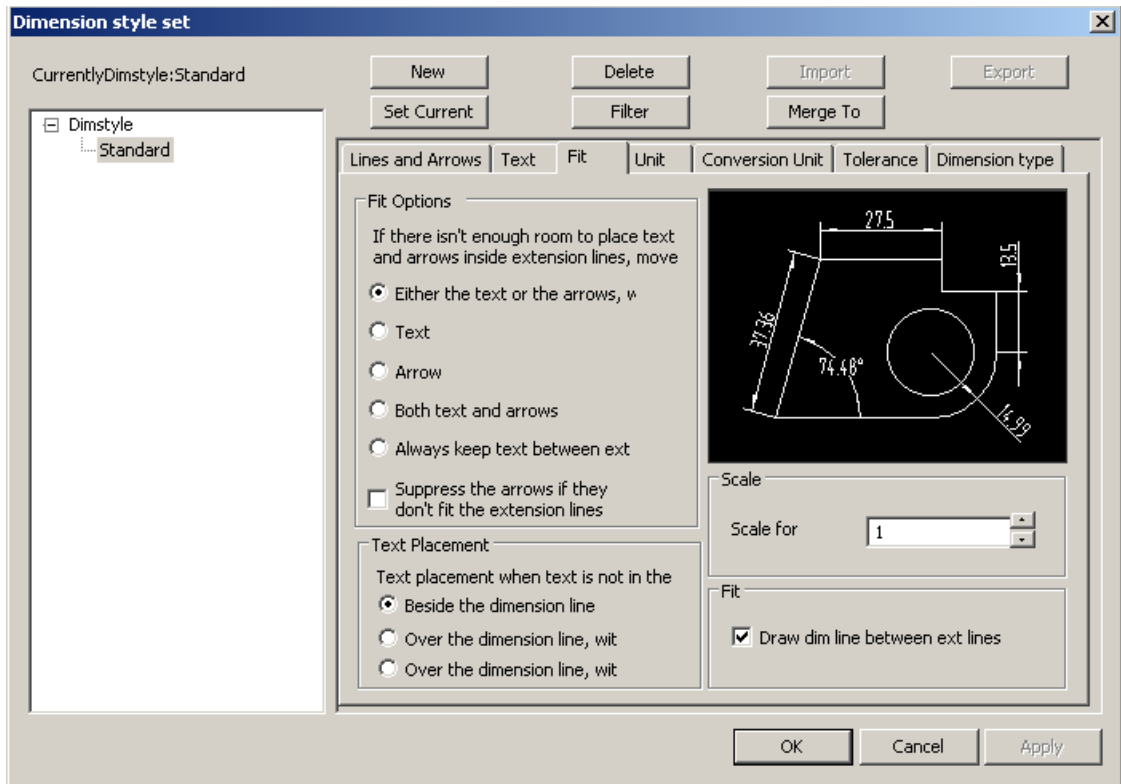
- **Distance between dimension line and text:** Control text to dimension line position, the default is 0.625mm.

Text alignment: it won't be introduced here.

【Text alignment mode】: set alignment mode for basic dimension text, such as parallel to dimension line, keep horizontal, or ISO standard.

【Tolerance alignment mode】: set alignment mode for tolerance text, such as align top, align middle, align bottom.

- (3) 【Fit】: Set relations between text and arrow, in order to get best dimension line. Fit option of dimension style is shown in following figure.



Pic6-59 fit

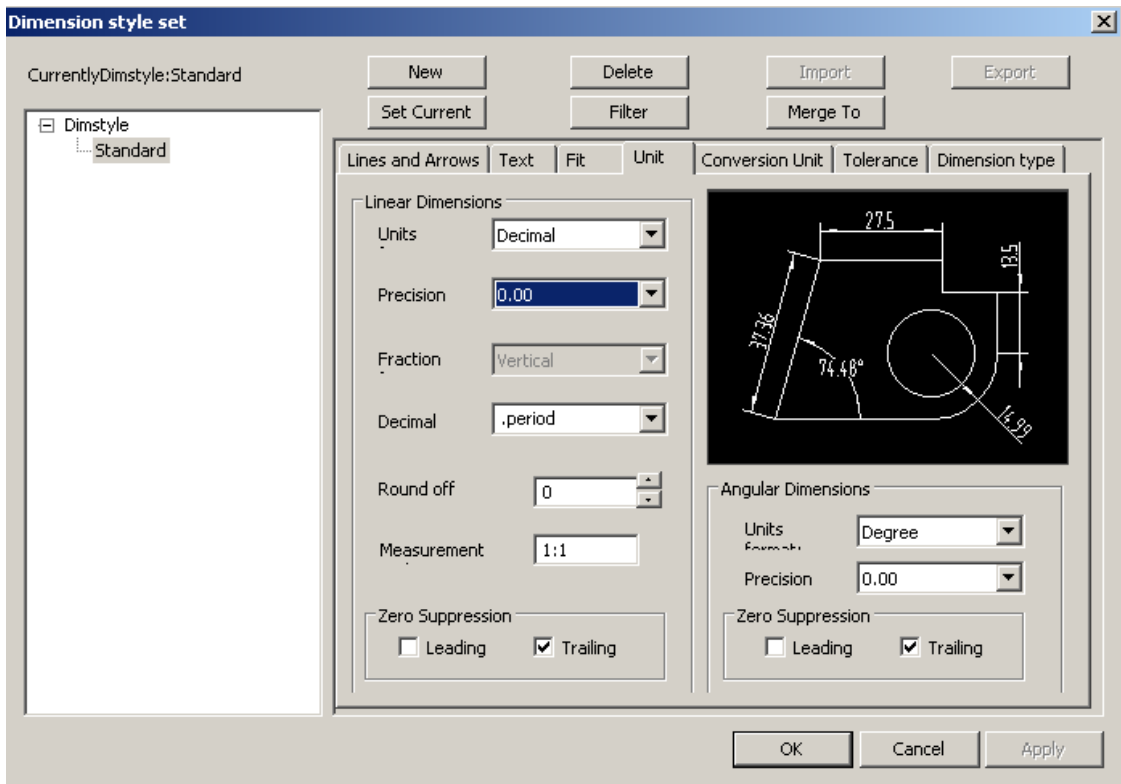
【Adjust option】: When there is not enough space for text and arrow with border line, user can set to move text or arrow out of border line, so that the text, arrow will be within the border line and best result can be got, if text is not within border line, it will not draw arrow.

【Text position】: if default space is not enough for text, user can put the text near dimension line, above dimension line without down-lead line, above dimension line with down-lead line.

【Dimension total scale】: Zoom in or zoom out dimension text and arrow as per input scale.

【Optimize】: User can set to draw dimension line between dimension border.

(4) Unit: Set dimension accuracy, please find the following figure.



Pic6-60 unit

- Linear dimension: Set parametric for line dimension format and accuracy.

【Unit system】: Set all current unit format for the dimension ext angle dimension.

【Accuracy】: Set decimal places to be shown in the dimension, accuracy is based on selected unit or angle format.

【Fraction format】: Set fraction format as vertical or horizontal, only when fraction is selected from unit system can parametric be set.

【Decimal spacer】: Decimal expression is divided into 3 modes: dot, comma, space.

【Decimal accuracy】: Set rule of rounding for dimension value except angle dimension. If input 0.25, then all dimension distance will be taken or left regarding 0.25 as unit. If input 1.0, the dimension distance will be taken or left near to integer. Digit number behind decimal dot is decided by the setting of accuracy.

- **Measure scale:** ratio between mark dimension and actual dimension. For example, when the scale is 2, to a circle with diameter of 5, the result of mark diameter is $\Phi 6$, the default is 1.

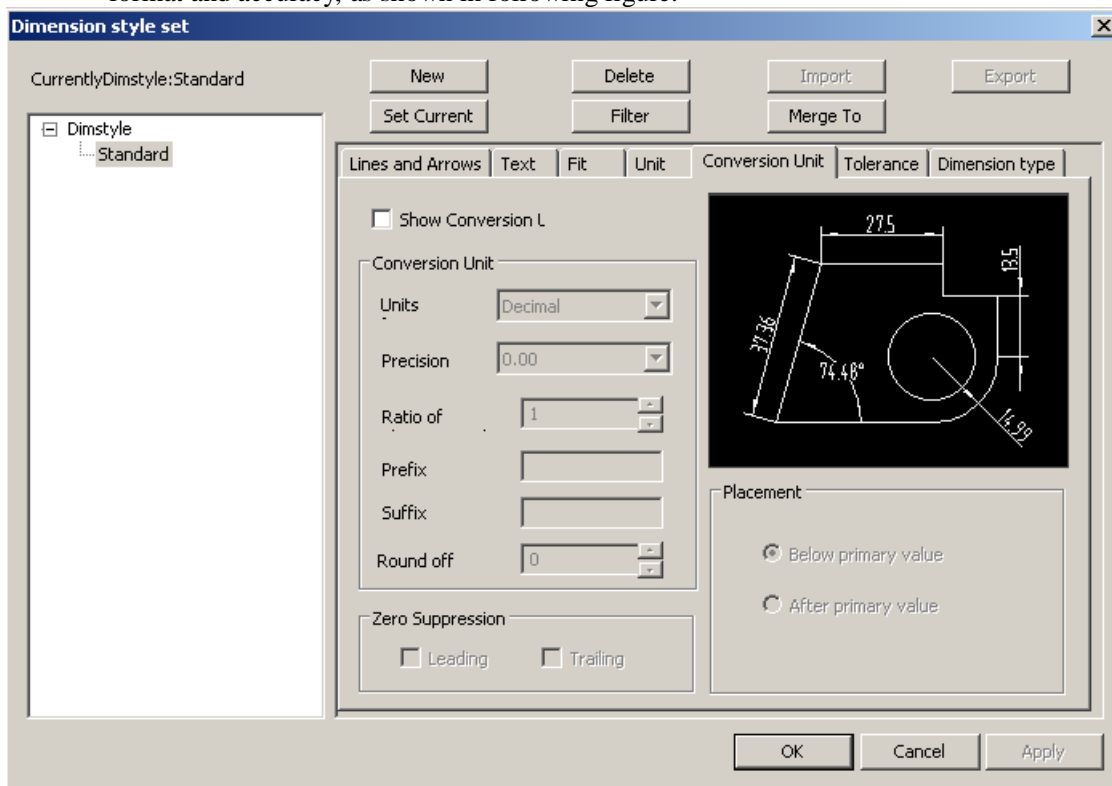
- **Zero compression:** in dimension mark, 0 before or after decimal will be removed. Such as dimension value 0.901, the accuracy is 0.00, if user selects prefix, the mark result will be .90, if suffix is selected, the mark result will be 0.9.
- Angle dimension

System of units: Angle mark unit includes degree and degree, minute and second.

Accuracy: Angle mark accuracy, which is accurate to 5 digits after radix point.

Zero compression: Control whether forbid outputting leader zero and follow-up zero.

(5) Conversion unit: Specify conversion unit display for dimension value, and set its format and accuracy, as shown in following figure.



Pic6-61 conversion unit

When “Display conversion unit” is selected, user can set conversion unit, such as unit system, accuracy, zero compression, display position.

- **Conversion unit:** Display and set current unit format of all dimension type except angle.

【Unit system】: Set unit format of conversion unit.

【Accuracy】: Set decimal digit number of conversion unit.

【Conversion scale modulus】: Specify one multiplier which will be used to conversion factors of main unit and conversion unit. Eg. Convert inch to millimeter, input 25.4, this value will not affect angle dimension, what's more, it will not be applied to taken or left value, positive or negative value.

【Dimension prefix】: It includes prefix in the conversion dimension text. User can input text or use control code to display special symbol. Eg. Input control code %c to display diameter symbol.

【Dimension suffix】: It includes suffix in the conversion dimension text. User can input text or use control code to display special symbol. The input suffix will take the place of all default suffix.

【Decimal accuracy unit】: set rule of rounding for all conversion unit of dimension type except angle dimension. Input 0.25, then all dimension value will be left or taken regarding 0.25 as unit. If input 1.0, the dimension distance will be taken or left near to integer. Digit number behind decimal dot is decided by the setting of accuracy.

- **Zero compression:** control whether forbid inputting leader zero and follow-up zero.

【Prefix】: Do not output leader zero of all decimal dimension. Eg. Make 0.5000 be .5000.

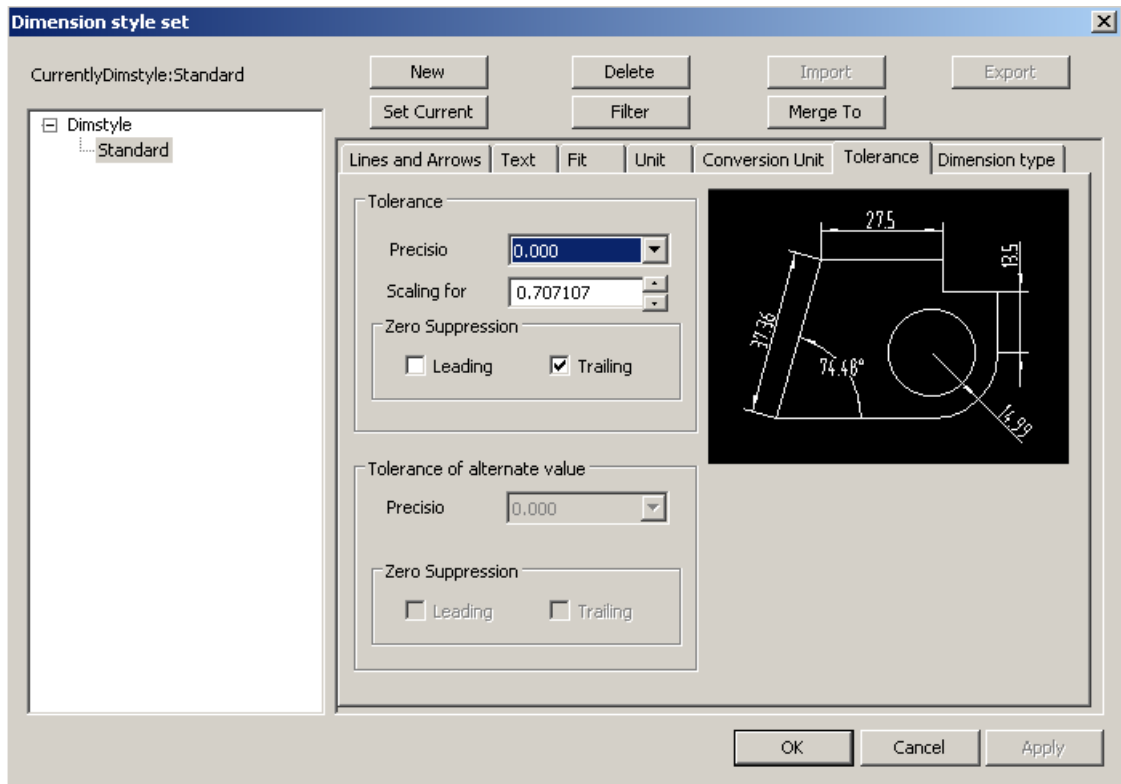
【Suffix】: Do not output follow-up zero of all decimal dimension. Eg. Make 12.5000 be 12.5, make 30.0000 be 30.

- **Display position:** Control position of conversion unit from dimension text.

【Behind main unit】: Put conversion unit behind main unit of dimension unit.

【Under main unit】: Put conversion unit under main unit of dimension unit.

(6) **Tolerance:** Control tolerance format and display of dimension text. As shown in following figure.



Pic6-62 tolerance

- **Tolerance:** Control tolerance format and display for the dimension text.

【**Accuracy**】: Dimension deviation accuracy, it can be calculated it accurately to five place of decimal.

【**Height scale**】: Set current tolerance text relative to basic dimension height scale.

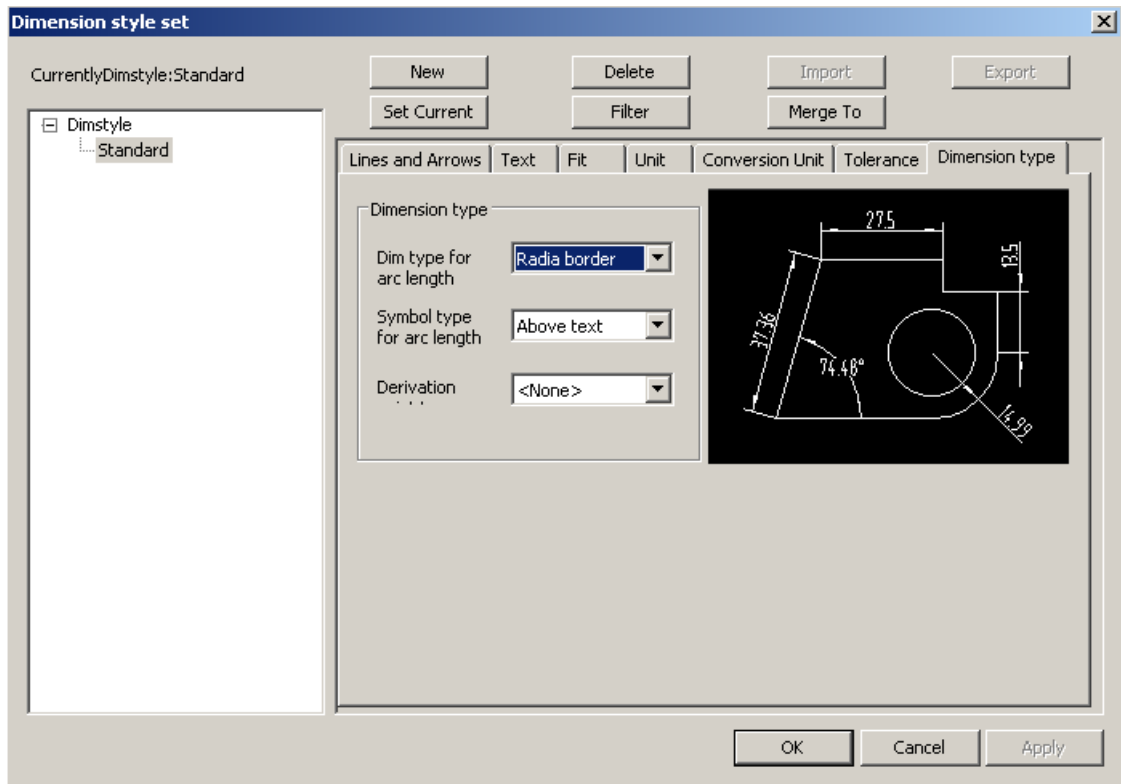
【**Zero compression**】: control whether forbid inputting leader zero and follow-up zero.

- **Conversion value tolerance:** set conversion tolerance unit format.

【**Accuracy**】: Display and set decimal digit number of conversion unit.

【**Zero compression**】: control whether forbid inputting leader zero and follow-up zero.

(7) **Dimension form:** Control arc length dimension and down-lead point or other parametric, as shown in following figure.



Pic6-63 dimension type

【Arc length dimension form】: set arc length dimension mode , border line to be vertical to span, or border line emitting.

【Arc length symbol form】: set arc length dimension symbol to be upon text or under text.

【Down-lead form】: Set dimension down-lead mode to be None or dot.

6.6.3 Down-lead line style

【Command】ldtype

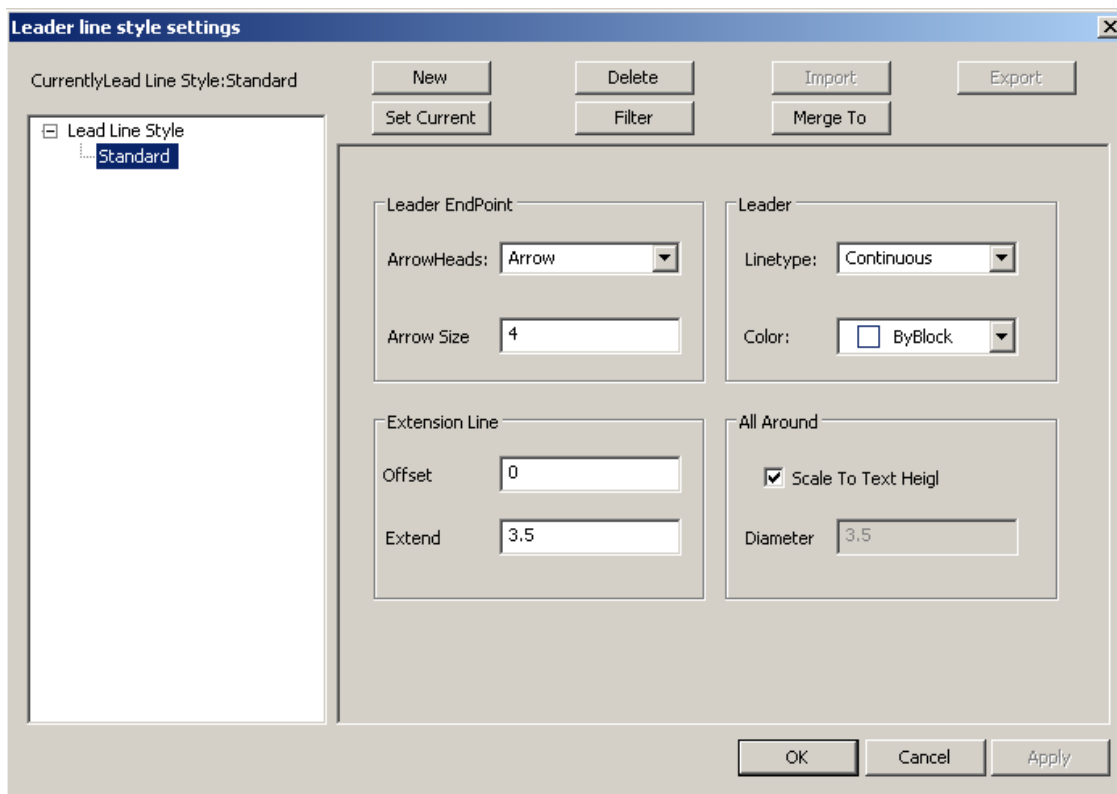
【Definition】Set all parametric for down-lead line.

Down-lead line of geometric tolerance, roughness, basic code, cutting symbol will all use command LDTYPE.

【Step】

User can execute command Ldtype in the forms of : click button lead line in the main menu of format, or click button lead line in Style management, or click command Ldtype directly.

Execute command Ldtype, following dialog box will pop up.



Pic6-64 Down-lead line style dialog box

- (1) Lead out start point: set lead line start point and arrow size.

【Arrow form】: Set arrow form for lead line to arrow, none, diagonal, dot, hollow arrow, rectangle arrow etc.

【Arrow size】: Set arrow size of lead line.

- (2) Dimension limit

【Deviation distance】: Set deviation distance for lead line dimension limit.

【Exceed distance】: Set exceed distance for lead line dimension limit.

- (3) Down-lead line

【linear】: Set lead line linear.

【Color】: Set lead line color.

- (4) Whole perimeter symbol

【Zoom text height】: Set text height for whole perimeter symbol.

【Diameter】: Specify diameter size of whole perimeter symbol directly.

6.6.4 Geometric tolerance form

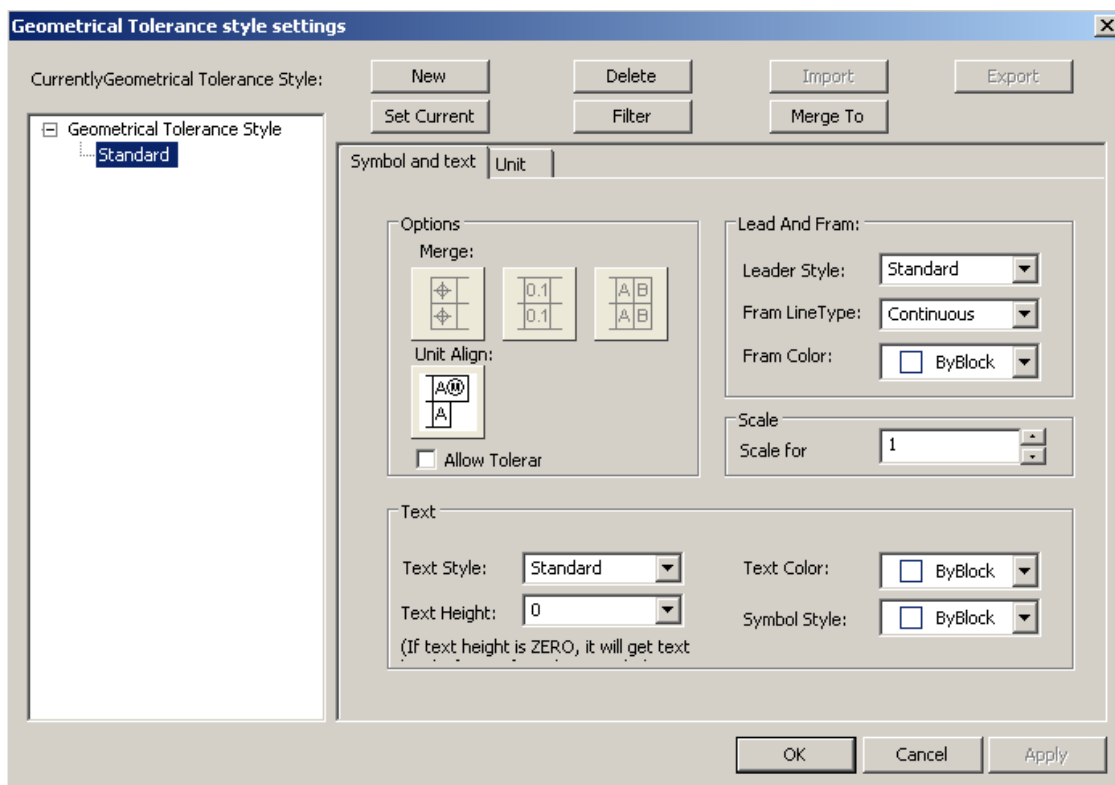
【Command】 fctype

【Definition】 Set all parametric of geometric tolerance.

【Step】

User can execute command fctype in the forms of : click button fctype in the main menu of format, or click button fctype in Style management, or click command fctype directly.

Execute command fctype, following dialog box will pop up.



Pic6-65 Geometric tolerance form dialog box

(1) Symbol and text: Set parametric of geometric tolerance symbol and text.

- Option: Set alignment and merge parametric for geometric tolerance.

【Merge】: if there are several lines of geometric tolerance, user can set whether merge its symbol, value, basic parametric when they are the same. Only when set align unit, can user set whether merge or not.

【Align unit】: Set whether unit align for geometric tolerance.

【Allowed tolerance 2】: set whether allow negative tolerance

- Lead line and frame: set parametric for lead line and frame.

【lead line style】: set lead line style cited by geometric tolerance.

【Frame linear】: Set frame linear.

【Frame color】: Set frame color for geometric tolerance.

- Text: Set text parametric for geometric tolerance.

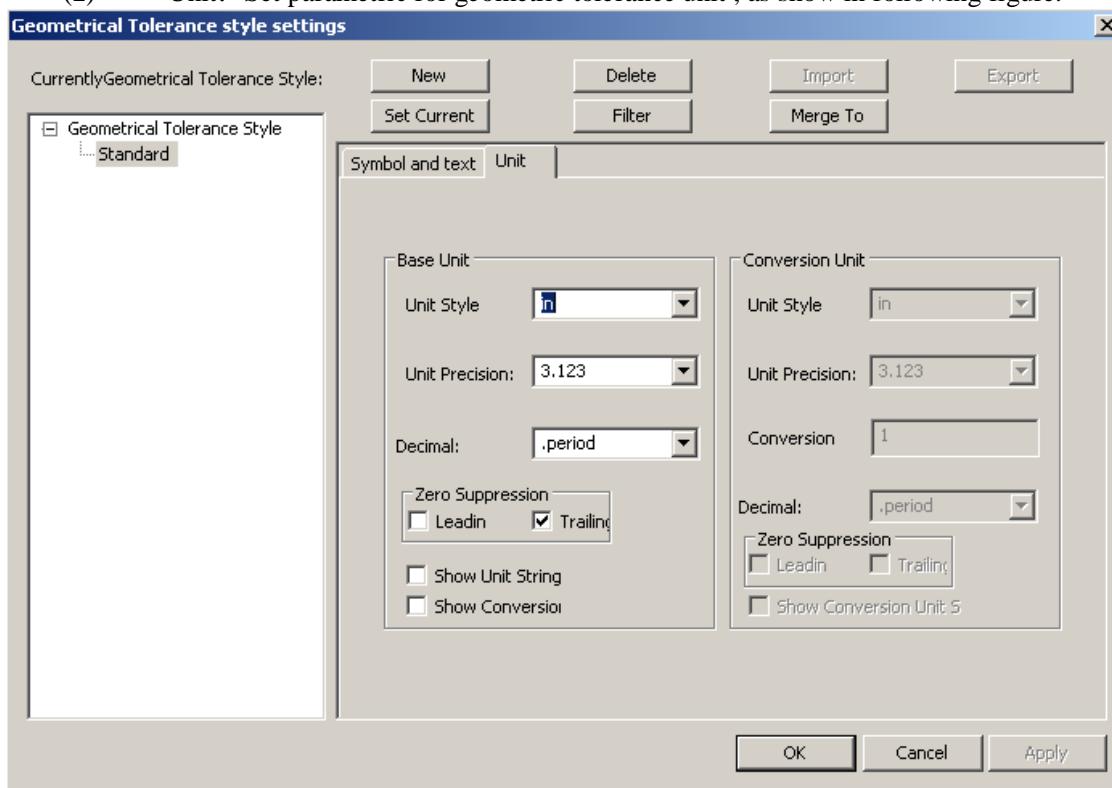
【Text style】: Set text style cited by geometric tolerance text.

【Text height】: set geometric tolerance text height.

【Text color】: set geometric tolerance text color.

【Symbol color】: set geometric tolerance symbol color.

(2) Unit: Set parametric for geometric tolerance unit , as show in following figure.



Pic6-66 unit of Geometric tolerance form

- Basic unit: set parametric for basic unit of geometric tolerance.

【Unit format】: Set basic unit format for geometric tolerance, it can be in, m, mm etc.

【Unit accuracy】: set decimal digit number of basic unit for geometric tolerance.

【Decimal form】: Set decimal form of basic unit, it can be dot, comma or space.

【Zero compression】: Set zero compression of basic unit.

【Display unit character string】: Whether display unit character string.

【Display conversion unit】: whether display conversion unit , when Display conversion unit is selected, user can set relative parametric for the conversion unit.

- Conversion unit: Set parametric of conversion unit.

【Unit format】: set conversion unit format, it can be in、 m、 mm etc.

【Unit accuracy】: set conversion unit decimal digit number.

【Conversion scale】: Set conversion unit conversion scale.

【Decimal form】: Set conversion unit decimal form, it can be dot, comma, or space.

【Zero compression】: Set conversion unit zero compression

【Display conversion unit character string】: Set to display conversion unit character string or not display.

6.7 Edit dimension

Dimension includes: size dimension, coordinate dimension, text dimension, engineer dimension, which are introduced in Section 6.2 to section 6.5. The already created dimension needs to edit position and its content.

In CAXA Draft, user can edit dimension object in several ways, eg. Dimension edit, pinch point edit, feature option panel, double click edit, dimension drive or other command.

- Most dimensions can be modified via command Dimension edit. When executing dimension edit command, the system can identify type of dimension object automatically, and user can edit then.
- Use double click edit to select dimension object and edit it.
- Select dimension object, use Pinch point edit to drag and edit pinch point.
- Select dimension object, user option panel of Feature to modify dimension parametric.
- Use command dimension drive to edit dimension and its related graph.
- In the course of edit, click right key and use dimension attribute setting to edit parametric, or user can edit it in immediate menu.

Following is introduction of dimension edit modes, and two examples are cited for complicated introduction.

The edit method for Coordinate dimension and engineer symbol dimension is the same as that of text edit and dimension edit.




6.7.1 Dimension edit command

【Command】 dimedit

【Icon】 

【Definition】 Select dimension object to be edited.

【Step】

User can execute command Dimedit in forms of: click button  in the main menu of Modify, or click button  in the tool bar of Edit tool, or click button  in the option card of Dimension, or execute command Dimedit directly.

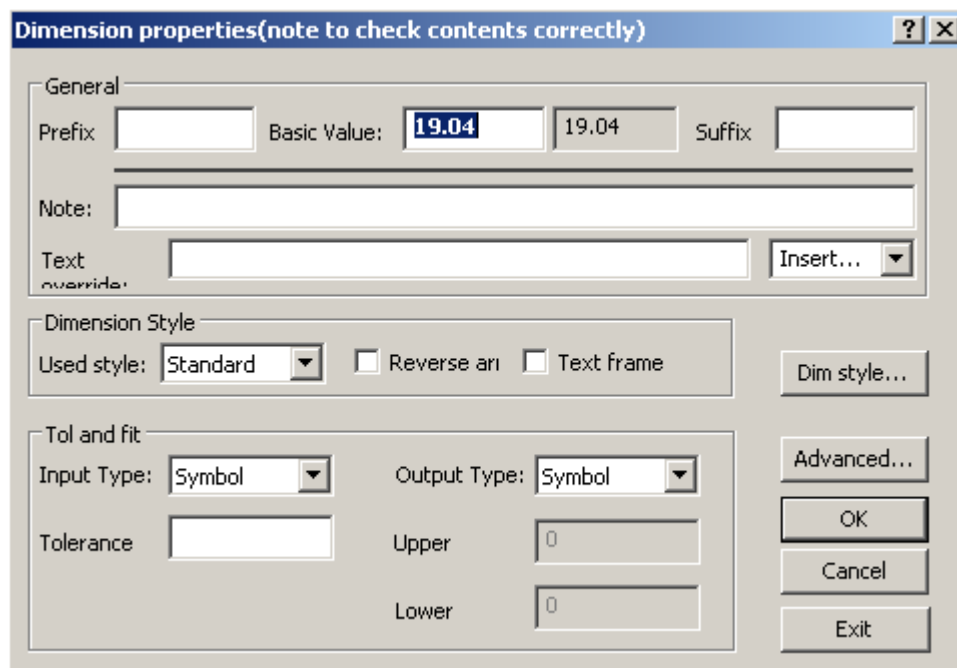
Once command Dimedit is activated, select dimension to be edited to enter into edit status. Then user can edit by immediate menu, dimension attribute setting, pinch point edit etc.

TO most dimension object, user can double click to execute DIMEDIT automatically.

6.7.2 Dialog box of dimension property setting

In addition to sized for the dimension, it is needed to add dimension tolerance, special symbol and set some special parameter. All these contents can be added and set conveniently in CAXA Draft. And dimension tolerance can be associated with basic dimension, in order to speed up modification efficiency.

Click right key when creating dimension, to enter in dialog box of Dimension property setting, as shown in following figure.



Pic6-67 Dialog box of dimension property setting

Following is the introduction to the above dialog box.

(1) Basic information setting

Prefix: it is used to describe or limit dimension value, for example, %c indicates diameter, 6- indicates number.

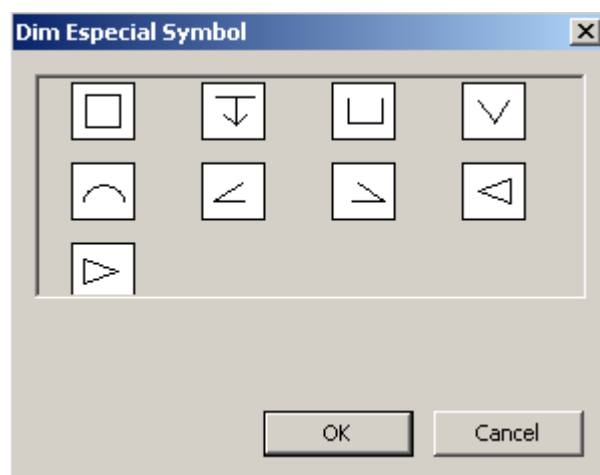
Base line dimension edit box: The default is actual measured valued, user can input value in it.

Suffix: it is used to describe or limit dimension value too.

Note: it is used to explain dimension etc.

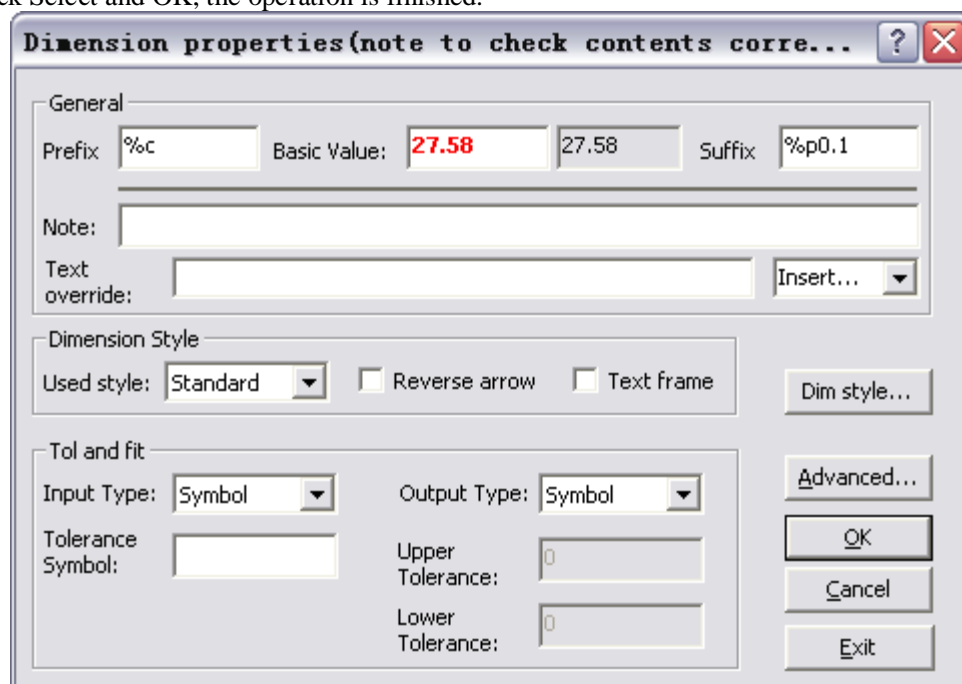
【Text replace】: When filling in content in this edit box, contents of prefix, basic size, and suffix will not be displayed, the newly filled characters will be the final content.

【Insert】: Click insert combo box, a sub-menu will pop up, user can insert all kinds of special symbols, such as diameter symbol, angle, fraction, roughness etc. Click option dimension special symbol, following dialog box will pop up.



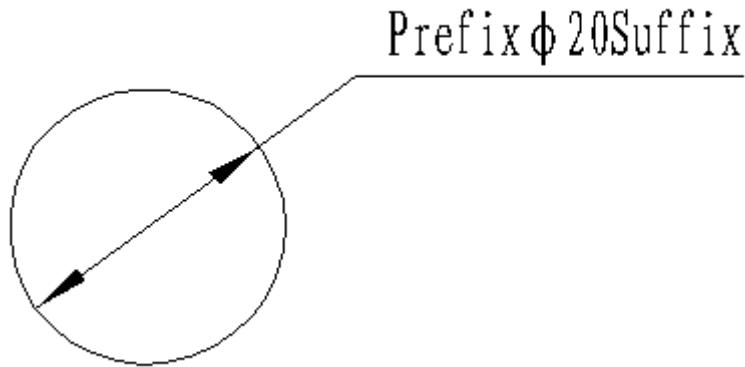
Pic6-68 especial symbol

Click Select and OK, the operation is finished.



Pic6-69 example

Fill parameter in the above dialog box, the following dimension will be created.



Pic6-70 result

(2) Dimension style setting

Click combo box on the right side of using style, and select style for creating dimension, and set arrow direction and text frame. Then click dimension style to activate dimension style dialog box, then user can set detail parameter in the box.

(3) Tolerance and Cooperation setting

Tolerance symbol edit box: When **symbol** is selected in the box of input form, user should input tolerance symbol name in this edit box, such as H7, h6, K6 etc. it will inquire table according to Base line dimension and symbol name automatically, and the inquired upper deviation and lower deviation will be displayed in corresponding edit box respectively. Or click button **advanced option**, and select proper tolerance code directly, as shown in the following figure.

When **coordination** is selected in the box of input form, user should input coordination name in this edit box, such as H7/h6, H7/k6, H7/s6 etc. it will mark according to inputted coordination when exporting. Or click button **advanced option**, and select proper tolerance code directly, as shown in the next figure. When the input form is deviation, the edit box will be gray, user can input at the position of upper and lower deviation.

Visual inquiry for tolerance and fit

Tolerance inquiry | Fit inquiry

☒ Hole tolerance ☐ Axes tolerance

A	B	C	D	E	F	G	H	J	Js	K	M	N	P	R	S	T	U	V	X	Y	Z
							H1		Js1												
							H2		Js2												
							H3		Js3												
							H4		Js4	K4	M4										
						G5	H5		Js5	K5	M5	N5	P5	R5	S5						
					F6	G6	H6	J6	Js6	K6	M6	N6	P6	R6	S6	T6	U6	V6	X6	Y6	Z6
			D7	E7	F7	G7	H7	J7	Js7	K7	M7	N7	P7	R7	S7	T7	U7	V7	X7	Y7	Z7
		C8	D8	E8	F8	G8	H8	J8	Js8	K8	M8	N8	P8	R8	S8	T8	U8	V8	X8	Y8	Z8
A9	B9	C9	D9	E9	F9		H9		Js9			N9	P9								
A10	B10	C10	D10	E10			H10		Js10												
A11	B11	C11	D11				H11		Js11												
A12	B12	C12					H12		Js12												
							H13		Js13												

Base Tolera

Common
Priority margin

OK Cancel

Pic6-71 inquiry tolerance

Visual inquiry for tolerance and fit

Tolerance inquiry | Fit inquiry

☒ Hole-base ☐ Axes-base

Datum	Hole	Axis																										
		c	d	e	f	g	h	js	k	m	n	p	r	s	t	u	v	x	y	z								
		Clearance Fit						Transition Fit						Wringing Fit														
H6						H6/f5	H6/g5	H6/h5	H6/js5	H6/k5	H6/m5	H6/n5	H6/p5	H6/r5	H6/s5	H6/t5												
H7						H7/f6	H7/g6	H7/h6	H7/js6	H7/k6	H7/m6	H7/n6	H7/p6	H7/r6	H7/s6	H7/t6	H7/u6	H7/v6	H7/x6	H7/y6	H7/z6							
H8					H8/e7	H8/f7	H8/g7	H8/h7	H8/js7	H8/k7	H8/m7	H8/n7	H8/p7	H8/r7	H8/s7	H8/t7	H8/u7											
				H8/d8	H8/e8	H8/f8	H8/g8																					
H9			H9/c9	H9/d9	H9/e9	H9/f9	H9/g9																					
H10			H10/c10	H10/d10				H10/h10																				
H11	H11/a11	H11/b11	H11/c11	H11/d11				H11/h11																				
H12		H12/b12						H12/h12																				

Base: 19.04 Fit: Fit best

OK Cancel

Pic6-72 fit inquire

Upper deviation edit box: When **symbol** is selected in the box of input form, the inquired upper deviation value is displayed in this edit box, user can input upper deviation value in corresponding box.

Lower deviation edit box: When **symbol** is selected in the box of input form, the inquired lower deviation value is displayed in this edit box, user can input lower deviation value in corresponding box.

Input form combination box: Symbol, deviation and coordination are three options of input form, by which user can control tolerance input mode. When **symbol** is selected in the box of input form, it will inquire deviation according to symbol name automatically, and the inquired upper deviation and lower deviation will be displayed in corresponding edit box respectively. When **deviation** is selected, user should input deviation value. When **coordination** is selected, user should input coordination

symbol in edit box of **symbol**, such as **H7/h6**, regardless what the export form is, it will mark according to symbol when exporting, as shown in following figure.

Dimension properties(note to check contents correctly)

General

Prefix: Basic Value: Suffix:

Note:

Text override:

Dimension Style

Used style: ☐ Reverse ani ☐ Text frame

Tol and fit

Input Type: Output Type:

Upper: Upper:

Lower: Lower:

Max clearance:0.042 Min clearance0.02

Fit System

☒ Hole-bas ☐ Axes-bas

Margin Tolerance

Hole Margin: Axes Margin:

Fit Method

☒ Clearance F ☐ Transition F ☐ Wringing Fil

Pic6-73 tolerance fit

Output form combination box: Symbol, deviation, (deviation) and symbol(deviation) are options of export form. by which user can control tolerance export form.

When **【Input form】** is **【Deviation】** 和 **【Symmetry】**, the output form will only have **【Deviation】** and **【(deviation)】**. When **【Input form】** is **【Cooperation】**, **【Output】** will only have 2 codes.

For example, When the export form is symbol, it will mark symbol, e.g. $\Phi 50K6$. When the export form is deviation, it will mark deviation, e.g. $\Phi 50^{+0.003}_{-0.013}$, When the form is (deviation), it will mark deviation value like $\Phi 50^{(+0.003)}_{(-0.013)}$, When the form is symbol (deviation), the mark will be $\Phi 50K6^{(+0.003)}_{(-0.013)}$.

6.7.3 Immediate menu input

During the course of size dimension or dimension edit, if "Basic Dimension or Prefix" pops up in the immediate menu, dimension tolerance can be inputted in the edit box with special character directly.

In the course of inputting dimension value, some special symbols like "?" (can be inputted dynamically via keyboard), angle symbol "°", upper and lower deviation of tolerance etc, user can use corresponding symbol of prefix and suffix to realize it.

Diameter symbol, it is expressed by %c, e.g. input %c40, the mark will be $\varnothing 40$.

Angle symbol, it is expressed by %d, e.g. input 30%d, the mark will be 30° .

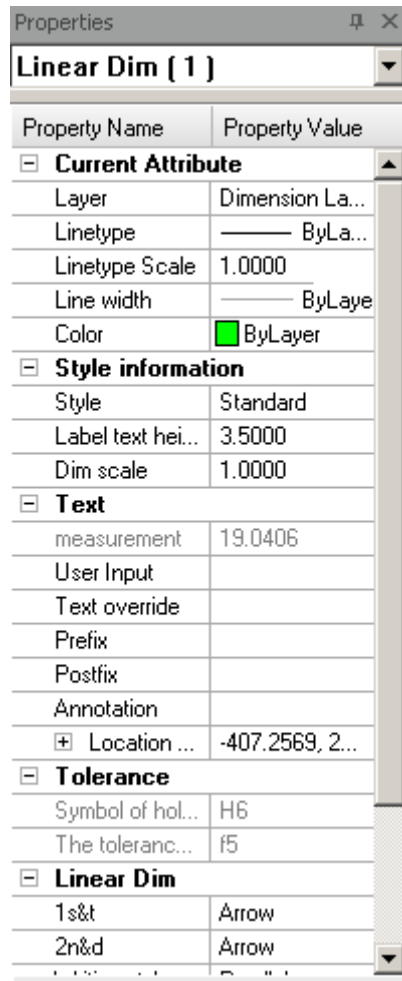
Tolerance symbol "±": it is expressed by %p, e.g. input 50%p0.5, the mark will be 50 ± 0.5 , text height of deviation value is the same as that of dimension value.

Upper, lower deviation value: the format is: dimension value plus%, plus upper deviation value, plus %, and plus lower deviation, then plus %b. Deviation value must have symbol, when deviation value is 0, it will be omitted, and text height of deviation value will be selected automatically in the system, which will be smaller by one size than dimension value, what's more, upper deviation and lower deviation will be recognized and located automatically, in order to make the mark format be up to national standard. For example, input 50%+0.003%-0.013%b, the mark will be $50 \begin{smallmatrix} +0.003 \\ -0.013 \end{smallmatrix} b$.

Suffix after upper, lower deviation value: the suffix is %b, Subsequent text height will be resumed automatically to text height of dimension value to mark.

6.7.4 Feature option panel edit

When it is dimensioned, select dimension, press right key, click option Feature to open Feature option panel, user can modify content and parameters in the option panel. e.g. when a linear dimension is selected, the feature option panel will be shown as follows:



Pic6-74 properties of linear dim

6.7.5 Dimension Drive

【Command】 drive

【Icon】

【Definition】 Select dimension object to be edited.

Dimension drive is a set of local parameterization function provided by this system. When user has selected a part of entity and relative dimensions, the system will build topological relations between entities according to dimensions. If user select desired dimension and modify its value, relative entities and dimensions will be changed accordingly. But topological relations between elements will keep unchanged, such as

tangency, contiguous etc. In addition, the system will handle graphics over-restricted or lack of restriction automatically.

To a large extent, user can adjust, modify dimension when graph is finished.

【Operation Step】

Select drive object, which is the desired part to be modified, as per system instruction, it will analysis entity and dimension of selected part. Besides selecting graphics entity, it is necessary to select dimension. Because engineering drawings depend on dimension mark to avoid ambiguous, the system analysis relations between elements according to dimension.

For example, there is a skew line, and its horizontal dimension is marked, when other dimensions are driven, its slope and vertical distance may have corresponding change. But its horizontal distance will keep the marked value. In a similar way, if user drive the horizontal dimension, then its horizontal length will be changed, which will be consistent with dimension value after being driven. To local parameterization function, it's very important to select parameterization object. In order to get desired drive result, it is necessary to do essential dimension mark before selecting drive object, and define relations between part to be driven and part won't be driven.

Generally speaking, if one entity hasn't necessary dimension mark, the system will judge restriction relations between entities according to default rules of contiguous, intersection and tangency etc.

Then user can specify a proper datum mark, any dimension indicates relative restriction relations between two or more graphics objects. If users drive this dimension, one end must be fixed, and the other end must be movable. The system will judge which end be fixed according to dimension to be driven and datum mark, in order to drive the other end. Usually, user should select special location point as datum mark, such as circle center, start point , center point and intersection point etc.

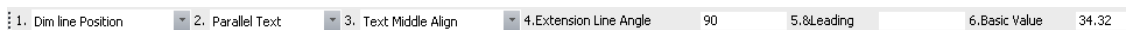
On the basis of previous two steps, User can drive one dimension. Select dimension to be driven, input new dimension value as per instruction, then the selected entity will be driven. Before exiting that drive object, user can drive multiple dimensions continually.

6.7.6 Dimension edit

Following is detail edit method for linear dimension, radius dimension or diameter dimension, or angle dimension.

(1) Linear dimension edit

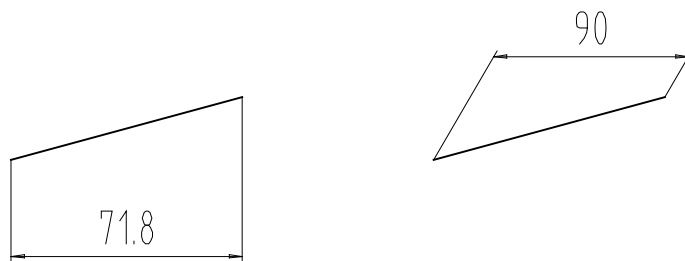
Select a linear dimension, an immediate menu will pop up.



There are four options in 1: of immediate menu, they are dimension line location, text position, text content and arrow shape. The default is dimension line location.

●Text location edit

User can modify text orientation, angle border and dimension value in above immediate menu. Among which, border angle means angle between dimension borderline and horizontal line. When new location point of dimension line is inputted, the edit operation is finished. Figure is an example of dimension line location of linear dimension. The border angle is changed from 90 degree to 60 degree. Dimension value is changed from 71.8 to 90.



Pic6-75 example

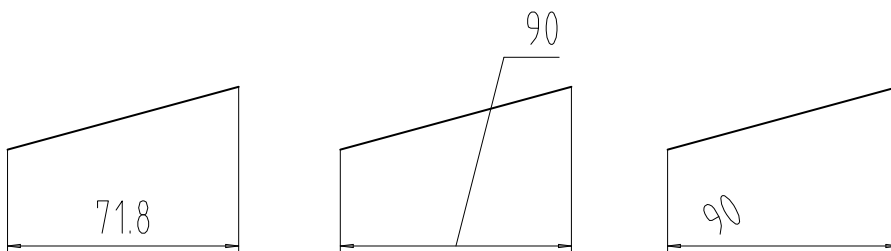
Text location edit

When editing text location, it just needs modifying location point of text, text angle and dimension value. Dimension line and dimension borderline won't be changed.

Switch 1: in immediate menu to text location, corresponding immediate menu will pop up:



Select whether add leader line or not in 3: of above immediate menu, then modify text angle and dimension value. When new text location point is inputted, the edit operation is finished. The following figure is an example of editing linear dimension text location.

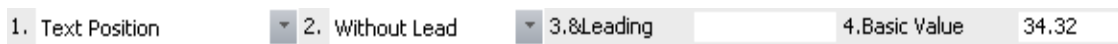


Pic6-76 example

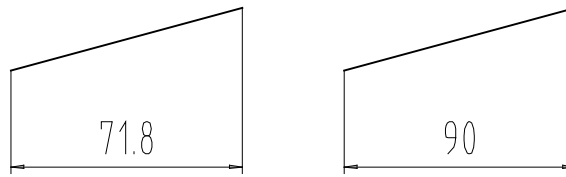
Text edit

It means modifying text content only, the others will remain unchanged.

Switch 1: in immediate menu to **text content**, corresponding immediate menu will pop up:



Modify dimension in immediate menu, and click left key to finish editing. The following figure is an example of editing linear dimension text.



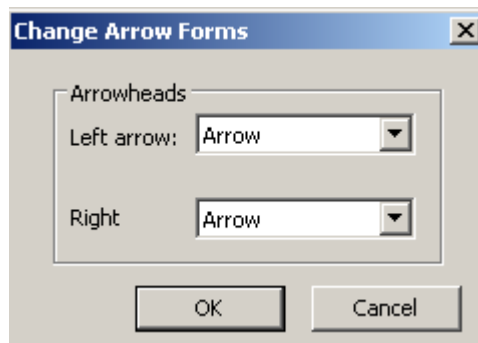
Pic6-77 example

- Modify arrow shape

Modify shape of leftward arrow and rightward arrow, user can select in the pop up menu of “Arrow shape editor”.

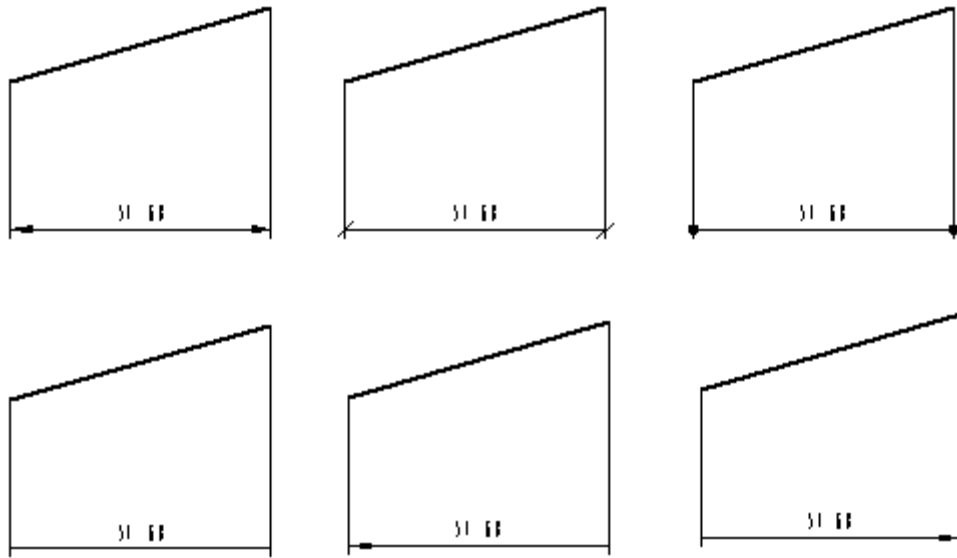
1. Arrow Shape 

Click OK when it is selected, then the modification is finished.



Pic6-78 change arrow forms

Following size is dimensioned with different arrow shapes:



Pic6-79 example

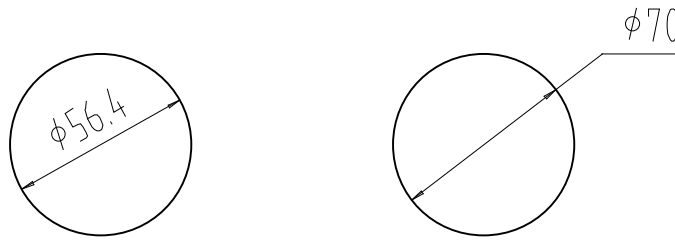
(2) Edit diameter dimension or radius dimension

There are two options in 1: in above immediate menu, they are dimension line location and text location. The default is dimension line location.

☐ Edit dimension line location of diameter dimension or radius dimension.

User can modify text orientation and dimension value in above immediate menu.

When new dimension line location is inputted, the editing operation is finished. The following figure is an example of editing dimension line location of diameter dimension. In which text parallel is changed to text horizontal, and dimension value is changed to $\Phi 70$.



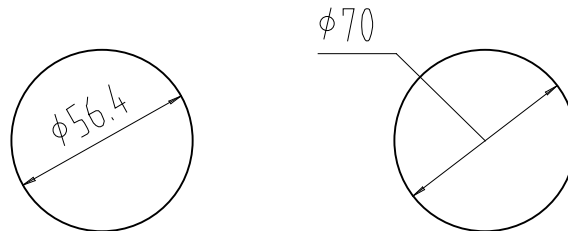
Pic6-80 example

- Edit text location of diameter dimension or radius dimension.

Switch 1: in immediate menu to text location, corresponding immediate menu will pop up:



User can select whether add leader line or not in above immediate menu, modify text angle and dimension value. When new text location is inputted, the editing operation is finished. Figure 6.58 is an example of editing text location of diameter dimension.



Pic6-81 example

(3) Edit angle dimension

Select an angle dimension, the following immediate menu will pop up:

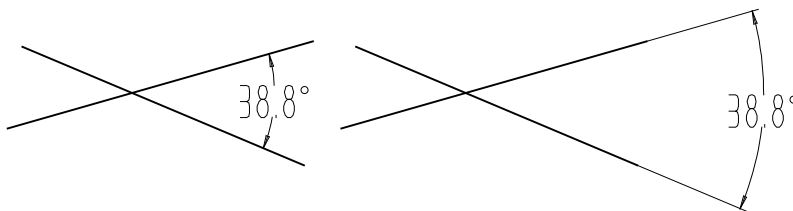


There are two options in 1: in above immediate menu, they are dimension line location and text location. The default is dimension line location.

- Edit dimension line location of angle dimension

User can modify dimension value in above immediate menu.

When new dimension line location is inputted, the editing operation is finished. The following figure is an example of editing dimension line location of angle dimension.



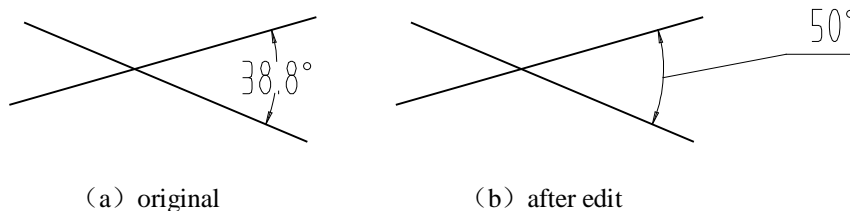
Pic6-82 example

●Edit text location of angle dimension

Switch 1: in immediate menu to text location, corresponding immediate menu will pop up:



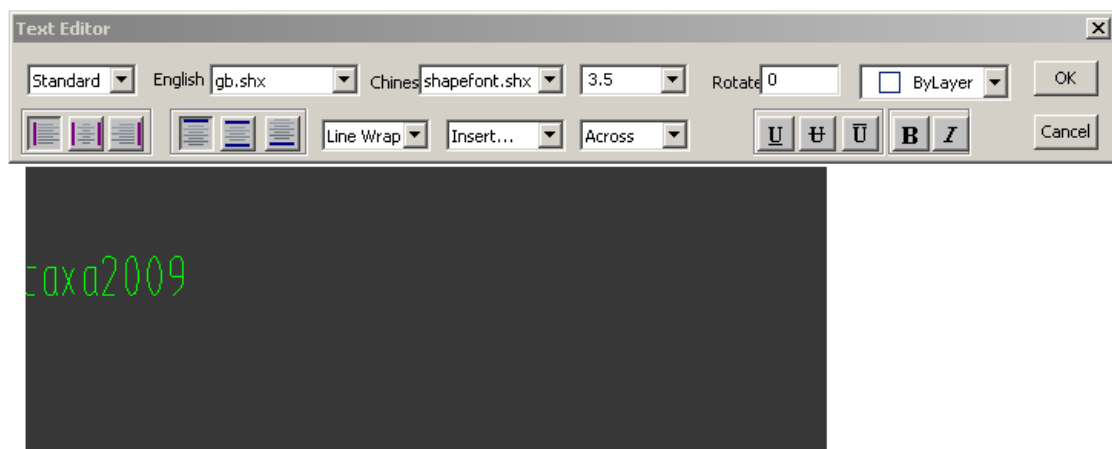
User can select whether add leader line or not in above immediate menu, and modify text dimension value. Figure 6.94 is an example of editing text location of angle dimension example.



Pic6-83 example of angel dimension

6.7.7 Text edit

Click command Dimension edit, then select text to be edited as per instruction, then corresponding dialog box will pop up, in which user can edit text and character parameter. Click button **OK** to finish editing. The system will regenerate corresponding text, as shown in following figure.



Pic6-84 dialog of text edit box

Chapter 7 Print

CAXA Draft supports all printers supported by WINDOWS, and it does not need to install printer individually in this system.

CAXA Draft supports printing drawing as per different parametric. Except for its own printing function, it supplies special printing tool for sheet-fed printing, typeset, and quantity printing. Which speeds up print efficiency.

Following is detail introduction of print function and print tool.

7.1 Print function

Print function of CAXA Draft will be introduced mainly in this section.

7.1.1 Summary

【Name】 Print



【Command】 print

【Icon】 

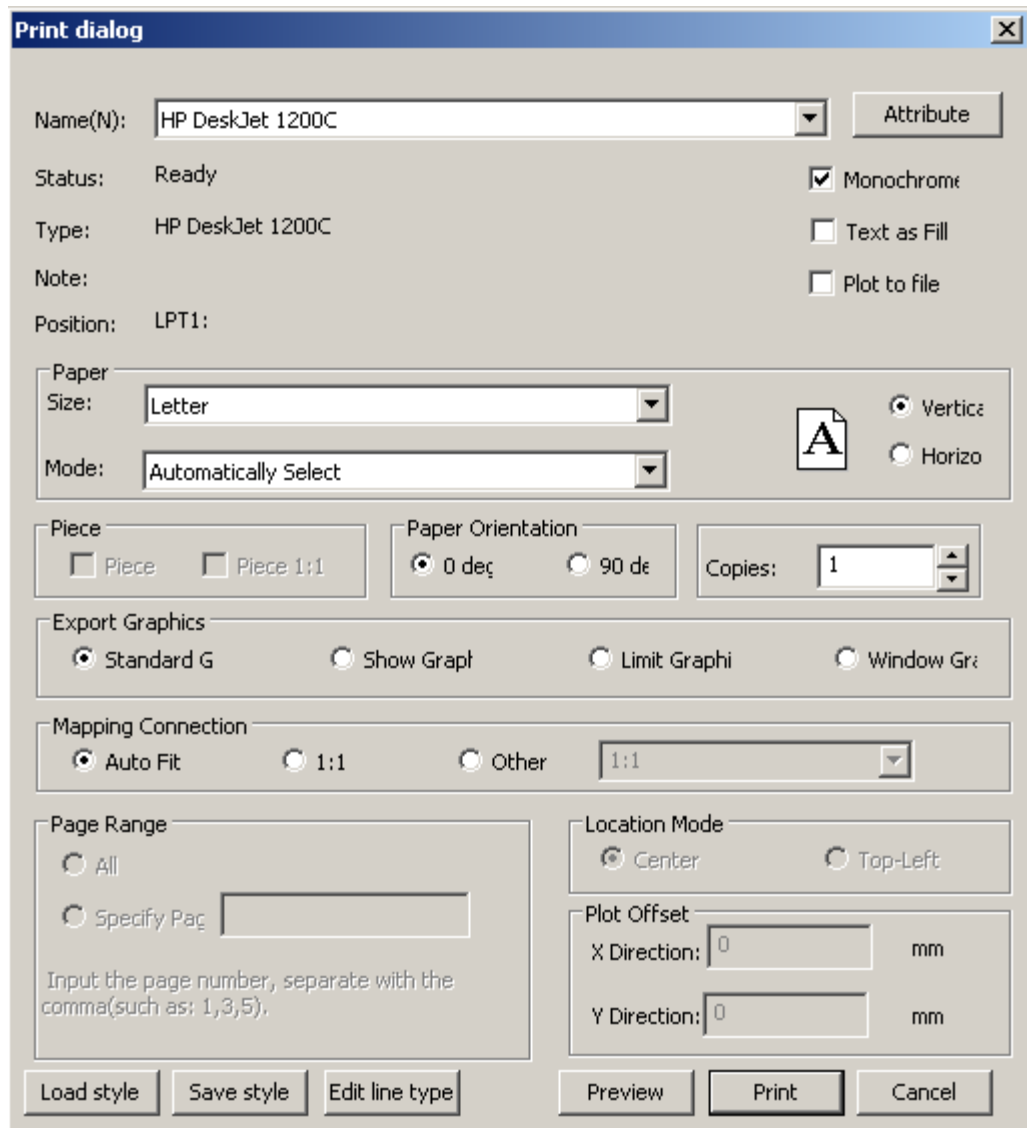
【Definition】 Print graph as per given parametric by printer.

Print function of CAXA Draft is similar to most Windows programs, user should confirm content to be print and set print parametric, then the printer will print.

【Step】

User can execute command Print in the forms of: Click button  in the quickstart tool bar, click button  in the main menu of File, or use hot key by Pressing Ctrl+P, or execute command Print directly.

Once command Print is activated, following dialog box will pop up.



Pic7-1 Print dialog

User can set pattern printout, paper size, equipment type or other parametric. Following is the detail of each option in the print dialog box.

7.1.2 Set print parametric.

Print parametric setting mainly includes “Printer setting, paper setting, paper orientation, graph orientation, output setting, jigsaw setting, location setting, print migrate setting, style save, linear setting, following is concrete explanation.

- (1) Printer setting area: User can select printer, and printer status will be shown in this

area.

(2) Paper setting area: Set paper size for the selected printer, paper source, paper direction.

(3) Paper orientation: Select Landscape or portrait.

(4) Graph orientation: Set graph rotate angle to 0 degree or 90 degree in this area.

(5) Output Graph option: it indicates scope for the to be output graph. User can only select output graph within following four scopes: standard drawing, display graph, limit graph and window graph.

- **【Standard drawing】** It indicates output graph within current system defined paper.
- **【Display graph】** it indicates output graph displayed on current screen.
- **【Limit Graph】** It indicates output all visible graphs in current system.
- **【Window graph】** it indicate output graph within specified rectangle frame.

(6) Jigsaw: Select combo box of Jigsaw, the system can make up automatically a large graph by several small drawings, number of jigsaw is decided by current paper size and all selected paper size.

【1: 1 when Jigsaw】It indicates jigsaw will be executed as per printable size, instead of paper size. Only when Jigsaw is selected, can option **【1: 1】** be selected.

If user wish to get result **【1: 1】** for jigsaw output result, and wish all graphs be in the hard cutting area of printer, please select jigsaw and **【1: 1】** simultaneously. Then needed paper number will be more than that of no **【1: 1】** selected.

(7) Mapping between graph and drawing: it indicates graph proportion between graph on screen and output graph in paper.

- **【Auto fill】** It indicates the output graph will be completely in the printable area.
- **【1: 1】** it indicates the graph is output as per **【1: 1】**

If the size of drawing paper is the same as printing paper size, the output drawing may be incomplete due to hard cutting area. If user needs to get **【1: 1】** paper, jigsaw can be adopted.

- **【Other】** It indicates output graph as per user-defined scale.

(8) Position mode: When **【1: 1】** and **【other】** are selected in the mapping, user can select two position modes, they are **【Center position】** and **【Top left position】**.

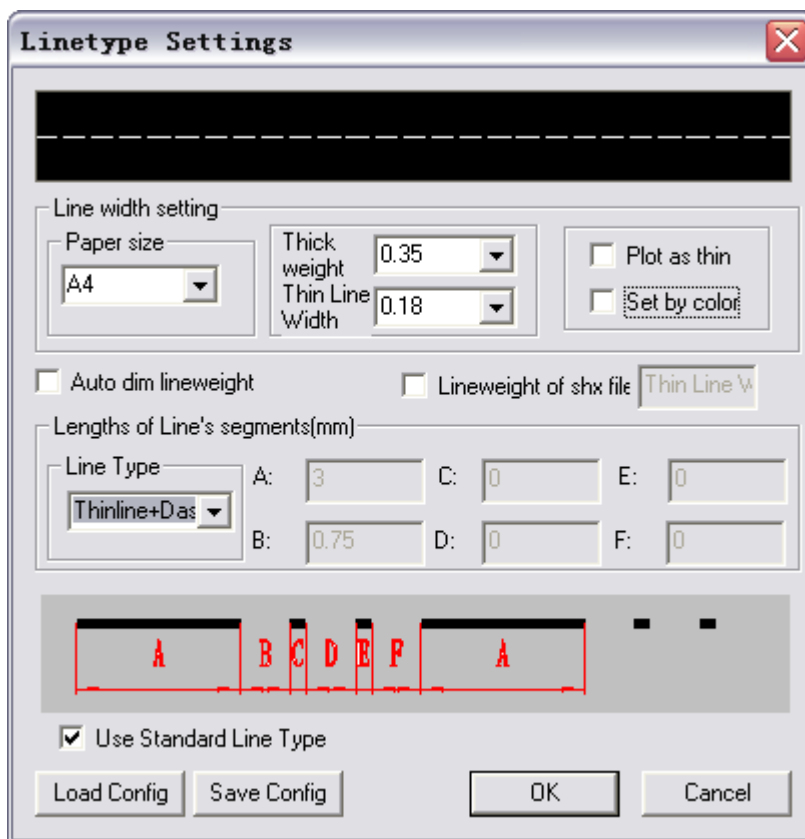
- **Center position:** Graph origin corresponds to paper center, the print result will be in the center of paper.
- **Top left position :** Top-left corner of frame corresponds to top-left corner of paper, the print result will be at the top-left corner of paper.

- (9) Preview: Click button Preview, real drawing output result will analog display on the screen, it section 8.1.4, we will introduce it in detail.
- (10) Page scope: For output multiple drawings, user can select “all” or “Specify page”.
- (11) Print Deviation : Move print position point , (X, Y) distance.
- (12) Print to file: If the file isn't sent to the printer for printing, but the result is sent to file, the check box of print to file can be selected. Then, the command of controlling drawing equipment will be output to a file with extension name .prm, instead of being sent to drawing equipment directly. Once the output is finished, this file can be used individually, and can be output into computer without electronic board.
- (13) Text as fill: When printing, user can set whether hiding text or not.
- (14) Black and white print: In order to get better black and white printing result and clear graphics color, in the black-and-white printer that supports gradation, CAXA electronic board has strengthened output equipment capability further.
- (15) Load in style and save style: save current configuration in the print dialog box, then user can load in saved style by option “load in style”.
- (16) Linear setting: Click dialog box of edit linear, as shown in figure 7-1, user can set print linear parametric, following is the detail.

7.1.3 Print linear setting

When printing, it usually needs to output different lines from that of drawing, such as adjusting line width, linear scale, adjust line width as per color, it is very convenient to set in CAXADraft.

Click button Edit linear, following dialog box will pop up. As shown in figure 7-2.

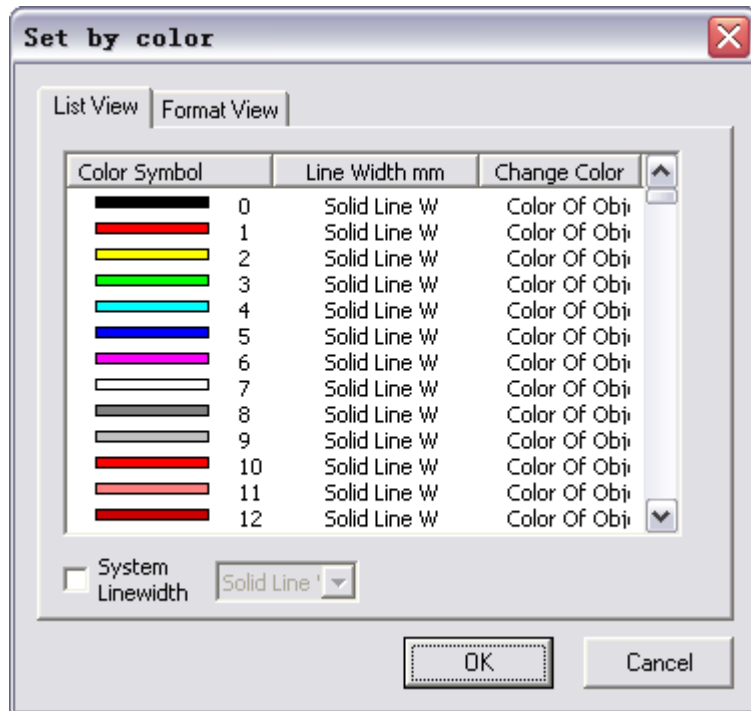


Pic7-2 linetype settings

- (1) Line width setting: input output width for standard line as per paper size. A series of line width value regulated by national standard is listed in the pull down list box. User can select any group or input value in the box, the effective scope of line width is 0.18~2.0mm.
- (2) Thin line printing : Print all line as thin line.
- (3) SHX file linear width: set linear width of SHX file.
- (4) Set linear as per color: set line width as per line color when printing drawing, and output drawing as per setting. Since the default print is thin print in the system, user should cancel this option to make sure it is not selected.

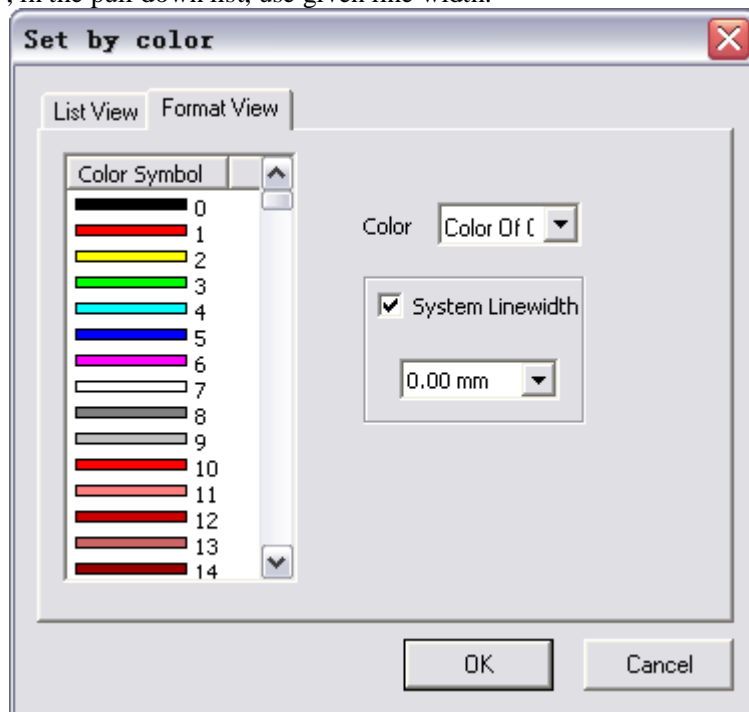
Select “set as per color”, following dialog boxes will pop up, as shown in figure 7-3 and 7-4.

The setting is divided into two parts, one is “list view”, the other is “Format view”. For list view, user can modify one to one. For format view, user can modify many to one. If multiple colors needs to be changed to one color, user Format view to modify.



Pic7-3 set by color

In the above dialog box, double click “Solid line width”, input line width, or tick and select “system line width”, in the pull down list, use given line width.



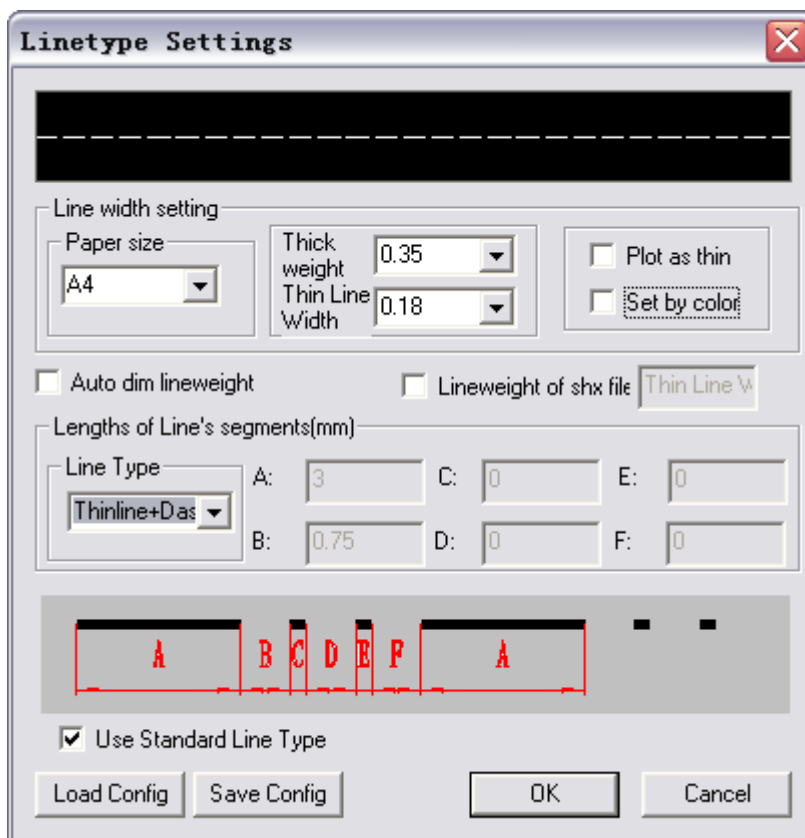
Pic7-4 set by color

In the above dialog box, Press Shift key or Ctrl key to select each color, and specify color or line width once.

The parametric in Set as per color dialog box will be saved automatically, when open it next time ,the modified one will be the default.

(5) Use standard linear: when this check box is selected, it will print as per standard linear. When this option is cancelled, it will print as per user's define. As shown in following figure.

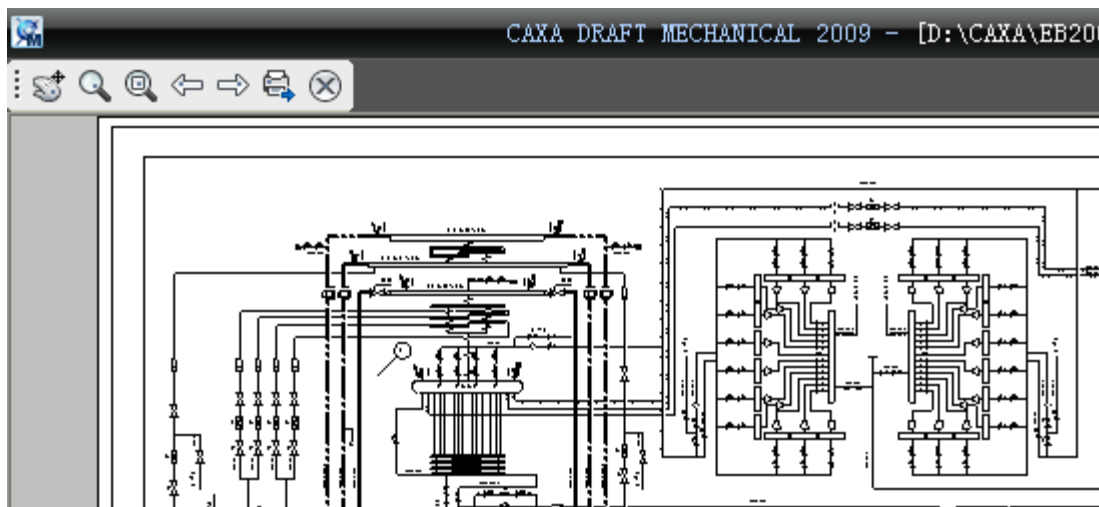
7-5



Pic7-5 customize linetype






7.1.4 Print preview

When print parametric is confirmed, user can click button “preview” dialog box shown in figure 7-1 to analog check print result, the result is shown in figure 7-6.



Pic7-6 Print preview

In the above dialog box, user can check analog result of print, the operation method is shown as follows:

- User can click command Pan, zoom, view frame or other button in the tool bar  to browse print window, or use mouse wheels to move or zoom.
- Click button  to print.
- Click button  to close print preview dialog box.
- When printing multiple drawings, user can click  or  to switch.
-

7.2 Print tool

In this section, we will mainly introduce print tool.

7.2.1 Summary

Print tool in CAXA Draft is mainly used for printing quantity drawings. It can organize drawings, including individual printing or typeset printing, user can adjust drawing setting and each print parametric conveniently.

The feature of CAXA Draft Print tool :

- It supports multiple printing items simultaneously, user can switch between different printing item at any moment.

- It supports individual printing and typeset printing, furthermore, quantity printing can be executed.
- It supports printing files in the format of EXB and DWG EXB.
- It can match print parametric automatically according to paper size.

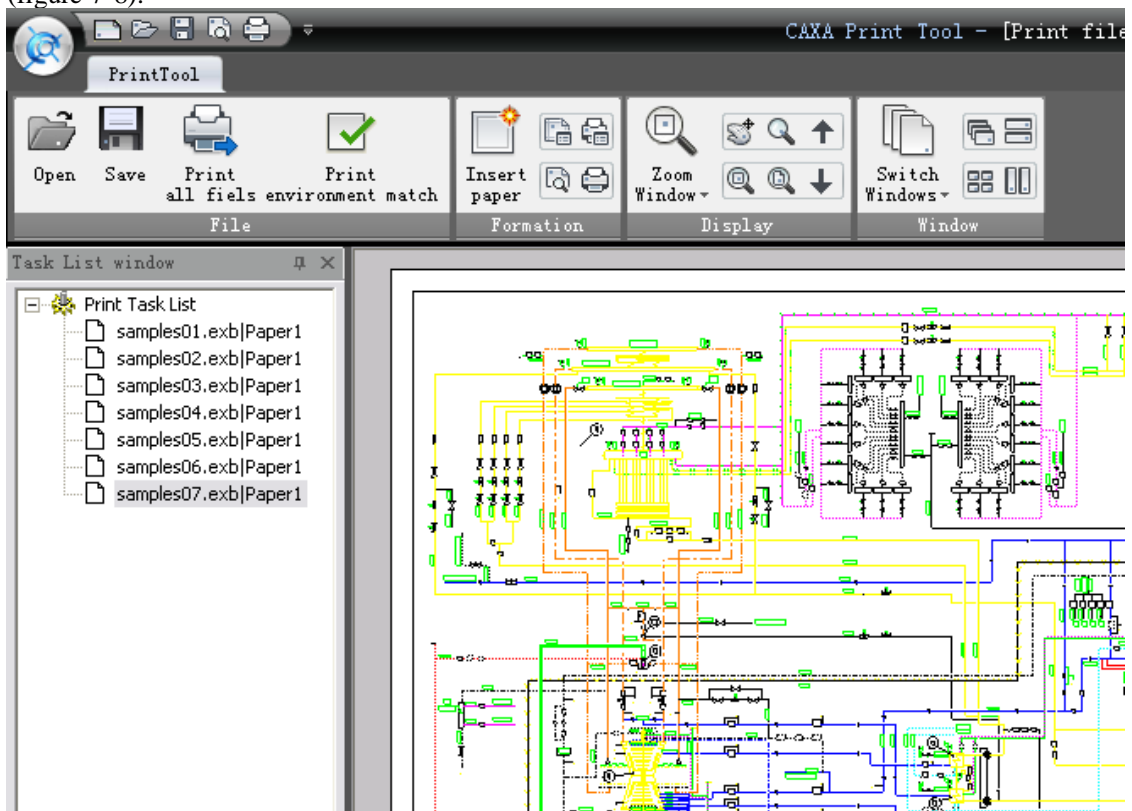
User can run Print typeset function in the following forms:

Click button outer tool in the main menu of Tool, click print tool button in the outer tool of tool function.

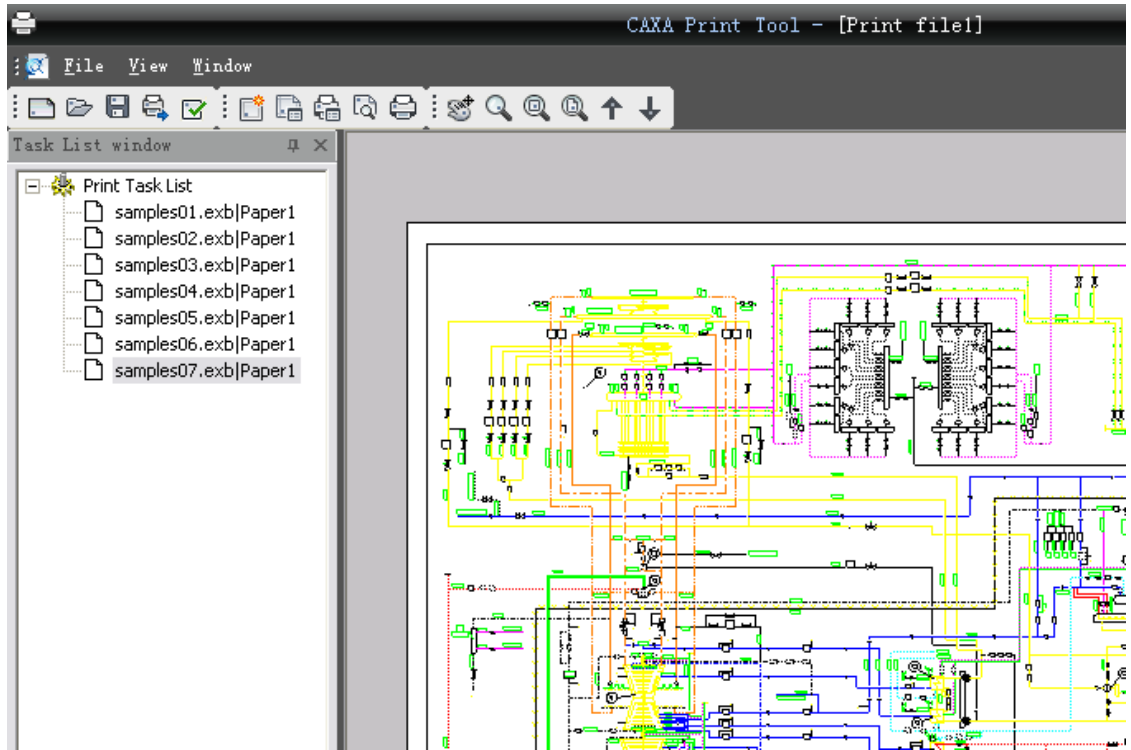
Following is the detain introduction of print tool interface, print file operation, insert drawing, browse drawing, parametric setting, print etc.

7.2.2 User interface

There are two interface styles in print tool, fluent interface (figure 7-7) and classic interface (figure 7-8).



Pic7-7 Fluent style



Pic7-8 classic style

Following is explanation to print tool interface:

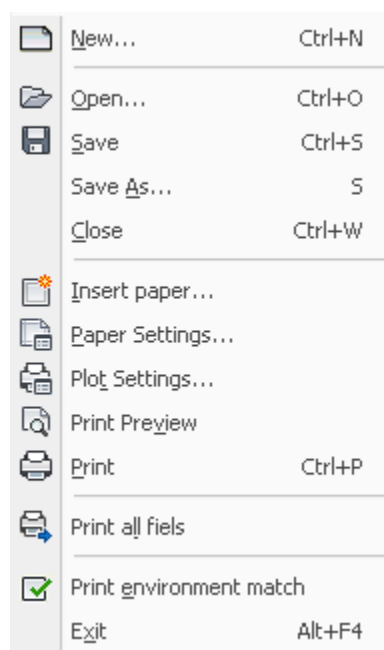
- It is Task List Window On the left of interface: display print task list, user can click each task to browse and set corresponding parametric. At the bottom of task list window, drawing attribute is displayed.
- It is Browse window on the right of interface. When one print task is selected, corresponding drawing information will be shown in the browse window.
- In fluent interface, user can use menu button, function area, quickstart toolbar to organize command, for classic interface, user can use main menu and tool bar to organize command.

7.2.3 File operation

The print tool support handling multiple printing task simultaneously, user can operate each print task , such as new, open, save, save as, close etc.

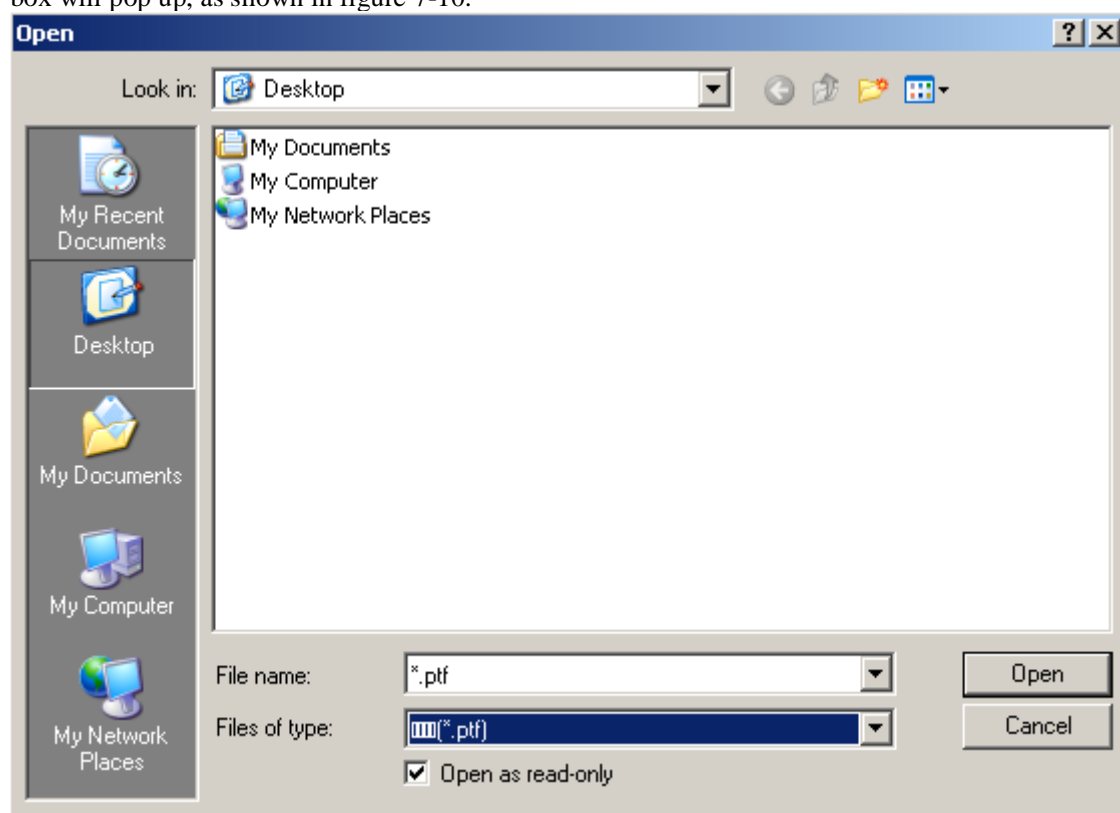
The operation method is introduced as follows:

Click menu button or click main menu File , following dialog box will pop up.



Pic7-9 file menu of print tool

Click any button to start corresponding command, for example, click Open, open file dialog box will pop up, as shown in figure 7-10.





Pic7-10 open print tool file

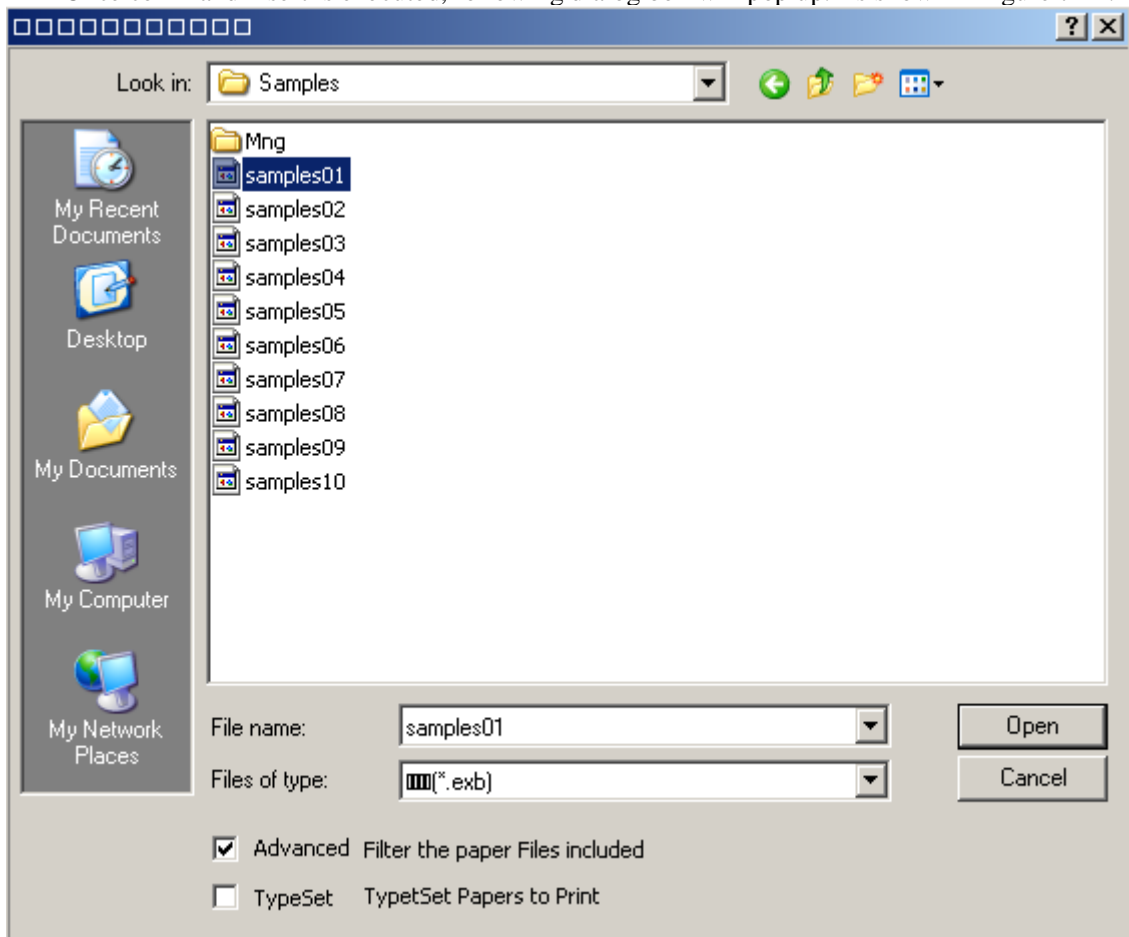
User can open PTF file in this dialog for printing.

7.2.4 Insert file

When using printing tool to print, user should insert file to be printed, then organize each printing task unit.

User can execute Command Insert in forms of : click button  in the main menu of file, or click  in the organize panel of function area.

Once command insert is executed, following dialog box will pop up. As shown in figure 7-11.

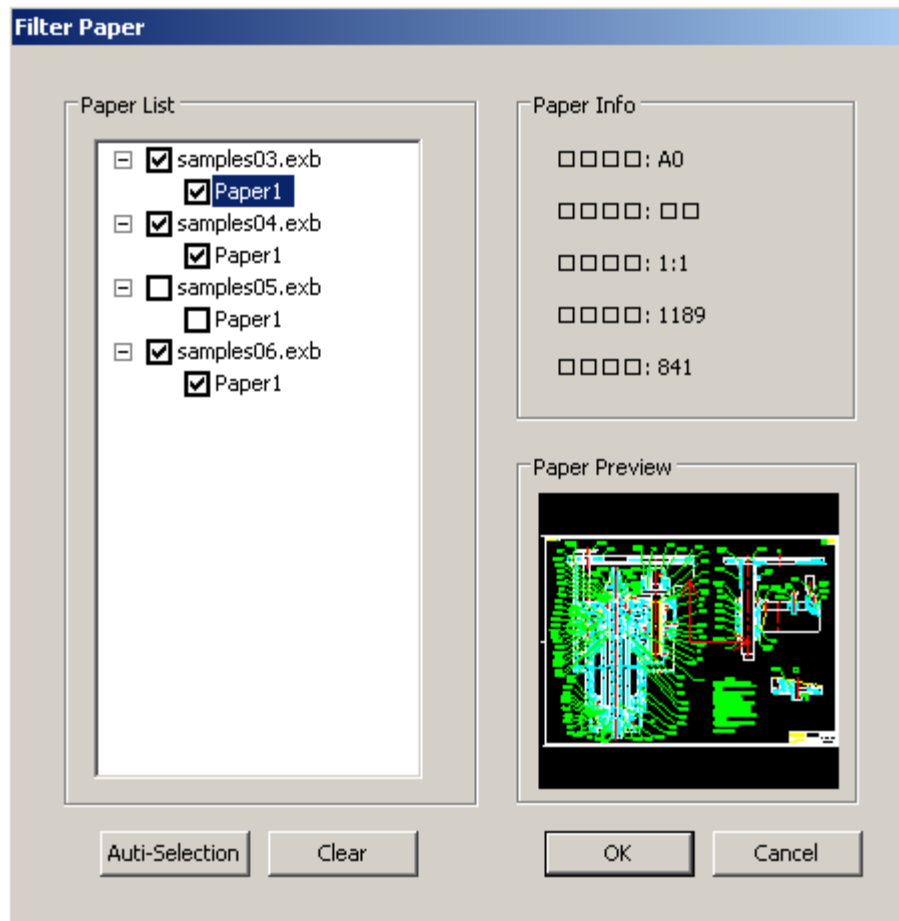


Pic7-11 insert paper file

(1) The meaning of option Advanced in the dialog box of Insert is shown as follows:

- Select this option: Select drawing then browse it, and select or cancel drawing.
- Not select this option: the selected drawing can be inserted to print task list window directly.

Select option Advanced, then select drawing, following dialog box will pop up, as shown in figure 7-12.



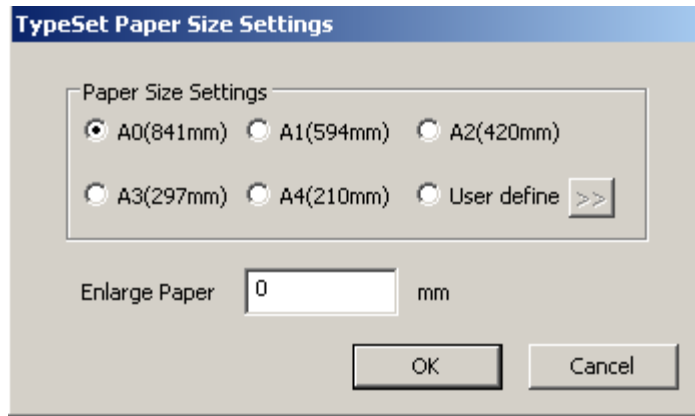
Pic7-12 select paper

In the above dialog box, user can examine selected drawing paper information and browse drawing content. Or click check box in front of the drawing name to cancel selected drawing. If there are several drawings in one file, user can select conveniently.

(2) The meaning of Option Typeset Insert in the Insert dialog box:

- Select this option: one typeset print task unit can be organized by the selected drawings.
- Not select this option: multiple print task unit can be organized by selected drawings.

Select typeset insert, then select drawing, following dialog box will pop up.



Pic7-13 typeset paper size settings

In the above dialog box, user can set typeset paper as A0,A1,A2,A3,A4 and user-defined, the meaning of Paper frame zoom-in is , space between typeset drawing frame is kept as per input value.

(3) When drawing is inserted, the inserted printing task unit is displayed in the window of “print task list”, user can continue to execute “insert” command to insert other new drawing to the print task unit.

7.2.5 Set parametric

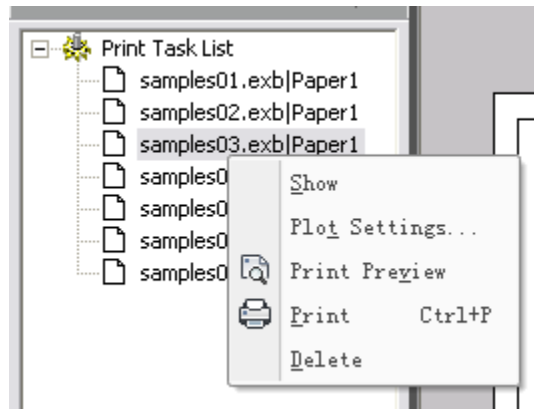
In this section, we will mainly introduce individual sheet print setting, typeset print setting, and print tool environment configuration etc.

7.2.5.1 Individual print setting

In print task list, select individual print task unit, the drawing information will be displayed directly in the preview area.

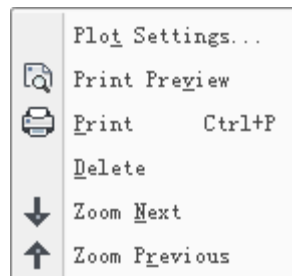
Individual drawing operation includes: print setting, print preview, print, delete etc. User can execute this command in forms of:

- Click right key on Print task unit, following dialog box will pop up, as shown in figure 7-14.



Pic7-14 select right menu

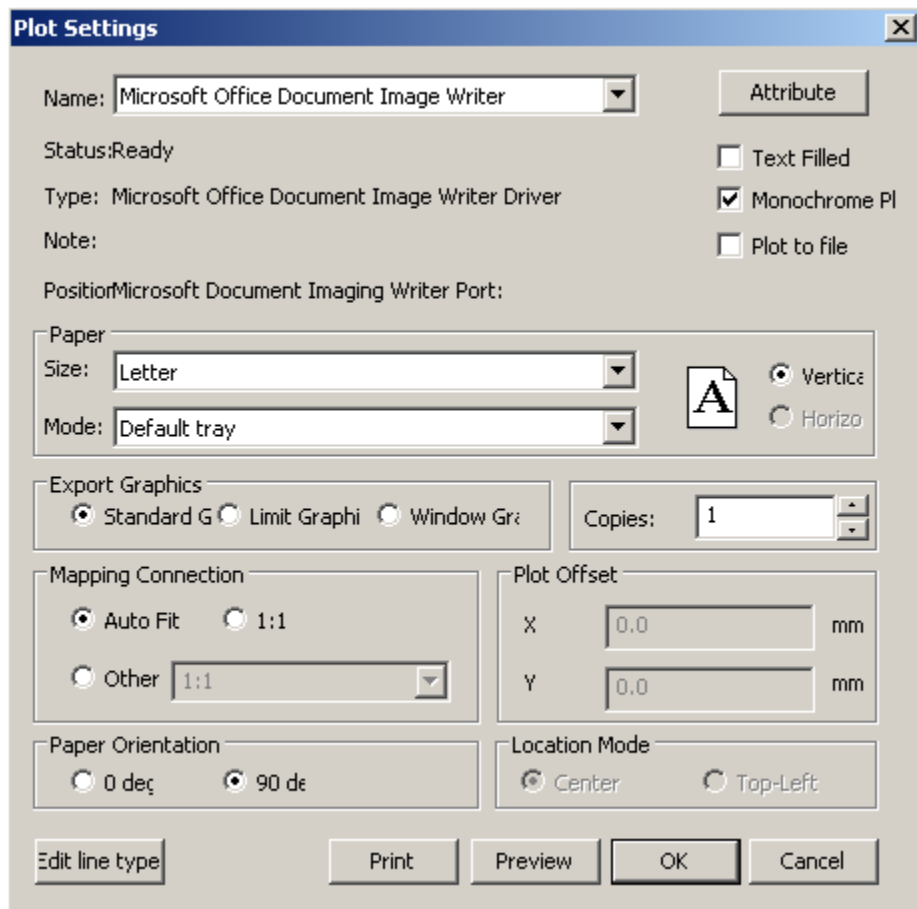
- Click corresponding button on the Organize function area.
- In the drawing preview window, click right key, following dialog box will pop up, as shown in figure 7-15.



Pic7-15 view right menu

Following is concrete method for operating individual sheet printing.

- (1) Print setting: set printer, paper, output graph, linear etc, the setting method for print is the same as that of introduction in section 8.1.



Pic7-16 plot settings

- (2) Print preview: preview actual graph as per set parametric. It is similar to introduction in section 8.1.
- (3) Print: Print select print task unit as per set parametric directly.
- (4) Delete: If print task unit is deleted, the delete operation can't be renewed.

7.2.5.2 Type set print setting

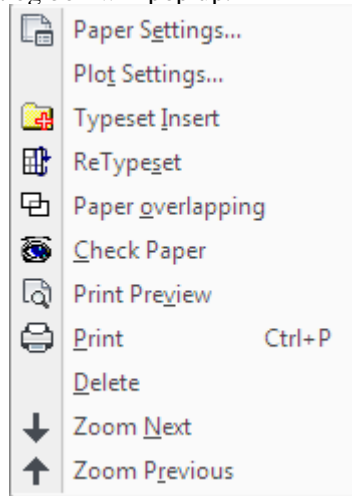
In the print task list window, select one typeset paper print unit, the selected typeset drawing information will be displayed in the preview area, as shown in figure 7-8.

Select one typeset print task unit, then set print, print preview, delete etc. the operation method is the same as that of individual sheet print.

In addition, typeset printing set and operation includes: typeset insert, rearrange, pan, rotate, delete, hide, graph overlap, paper examine, real display etc. user can execute these commands in

forms of :

- Select typeset print task unit: once this unit is selected, click right key in blank area the preview window, following dialog box will pop up.




Pic7-17 right menu of typeset print

- Click corresponding button in the Typeset panel of function area.
- Click corresponding button in the main menu of Typeset.

Following is the concrete method for typeset print operation.

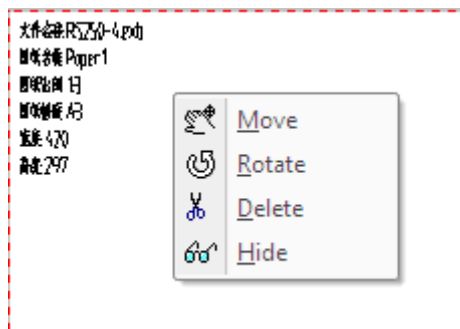
- (1) Typeset insert: select drawing, then insert it to current typeset print task unit.

Click button  , and select file to be inserted in the pop up dialog box, then click button Open, the open file will be inserted to newly created print typeset environment. When inserting file, it support multiple file selection.


Insert drawing means making the drawing be new print task unit.

- (2) Delete file: it includes delete print task unit and delete file from typeset print task unit.

- Delete print task unit. Select print task unit, then click right key on the unit or at blank area of preview, in the pop up menu, click button Delete to deleted selected print task unit.
- Delete drawing in typeset print: Select file to be deleted in the preview area or in the sub-node of typeset print task unit. There will be prompt for the selected drawing frame. Then click right key , in the right-key menu, select button “delete”,as shown in following figure.



Pic7-18 delete paper of typeset print

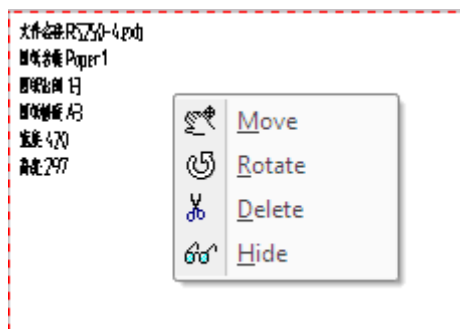
Or click button  in the menu of “Typeset” to delete relative file directly.

(3) Manual adjustment

By default, the drawing inserted to typeset print unit will be typeset automatically as per energy saving rule. User can adjust drawing by the command of Pan or Rotate.

User can execute Pan or rotate in forms of:


- Click corresponding button in the main menu of Print typeset.
- Click corresponding button in the panel of Typeset function area.
- Select typeset print task unit, in the preview area select drawing to be edited, then click right key, corresponding dialog will pop up, then user click related button, as shown in figure 7-19.



Pic7-19 right menu


(4) Graph overlap

By the command of Graph overlap, user can overlap file temporarily when pan or rotate file, in order to adjust file position conveniently.

Click Graph overlap in the main menu of Typeset or panel of Typeset function area, or select typeset print task unit then click right key to select button , then user can adjust file position directly.

(5) Re-typeset


User Re-typeset command to typeset again ignoring modification of manual typeset.

Click Graph overlap in the main menu of Typeset or panel of Typeset function area, or select typeset print task unit then click right key to select button , or user can click right key in the blank area of preview area to select Re-typeset option.

Execute command of Re-typeset, in the pop up dialog box, user can re-select paper size, space between drawings, then click OK, the system will re-typeset multiple open files.

(6) Hide


When drawing is inserted to typeset print unit, if a certain drawing won't be typeset temporarily, user can hide that drawing by command Hide.

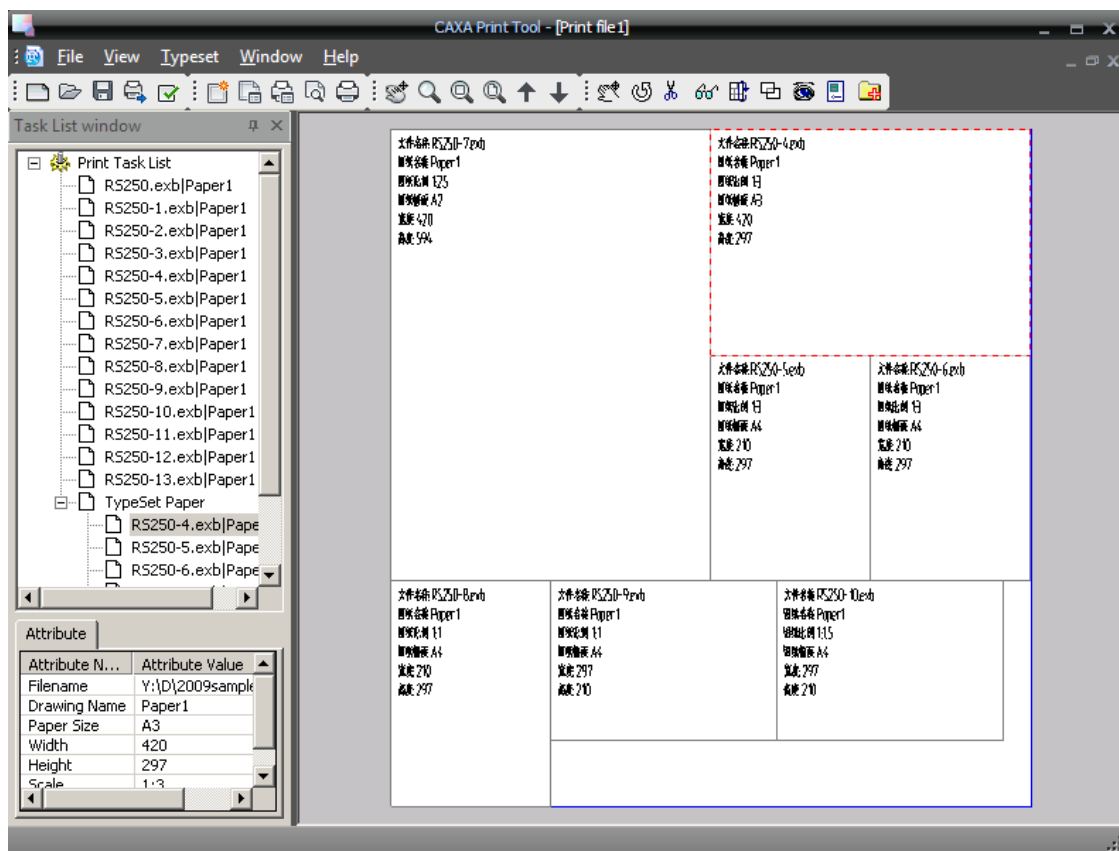
Select drawing in the print typeset task unit, click button main menu Typeset or button  in the panel of Typeset function area to hide drawing. Or select drawing , then click right key, in the pop up menu , select Hide.

(7) Real display

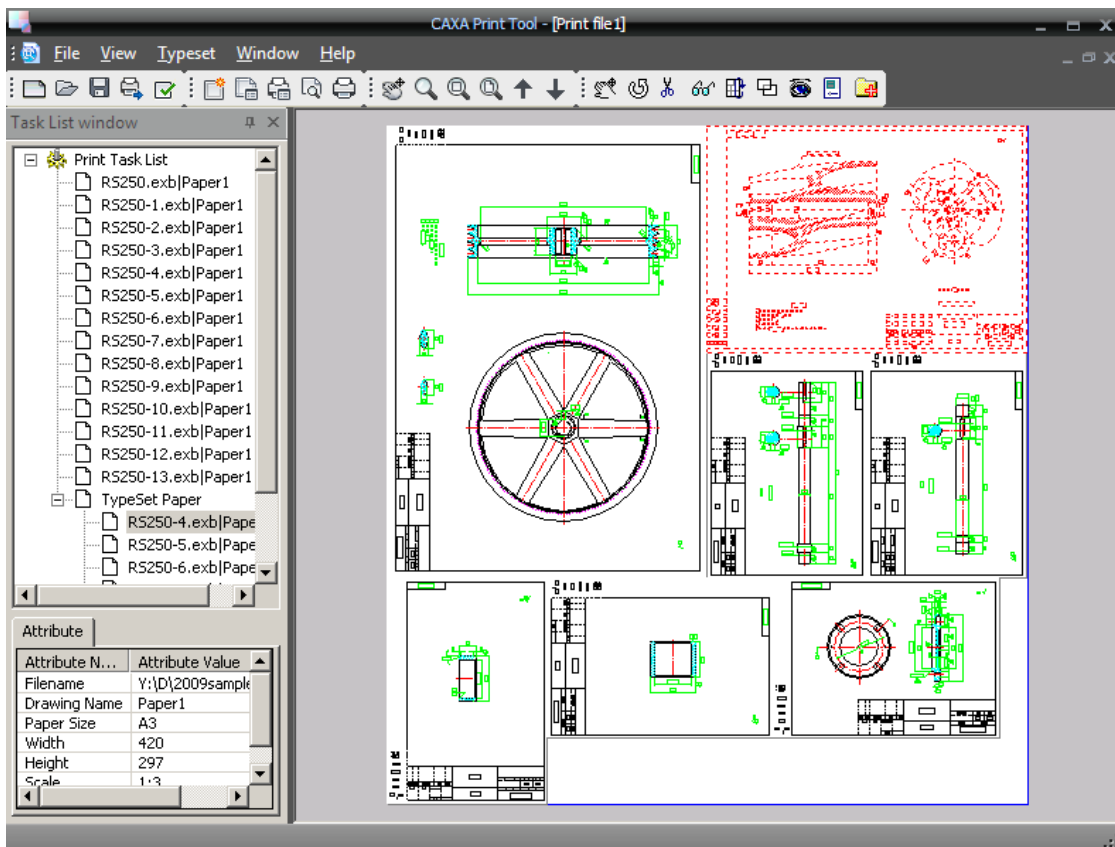
There are two preview modes in the preview area for typeset print task unit.

- Only preview drawing information, including file name, drawing name, drawing scale, drawing size, drawing width, drawing height etc.
- Preview real graph

Use Real display to switch preview between two preview modes. Click button Real display in the main menu of Typeset or button  in the panel of typeset function area, following is the comparison of two preview modes.




Pic7-20 (a) preview of typeset print





Pic7-20 (b) true view of typeset print

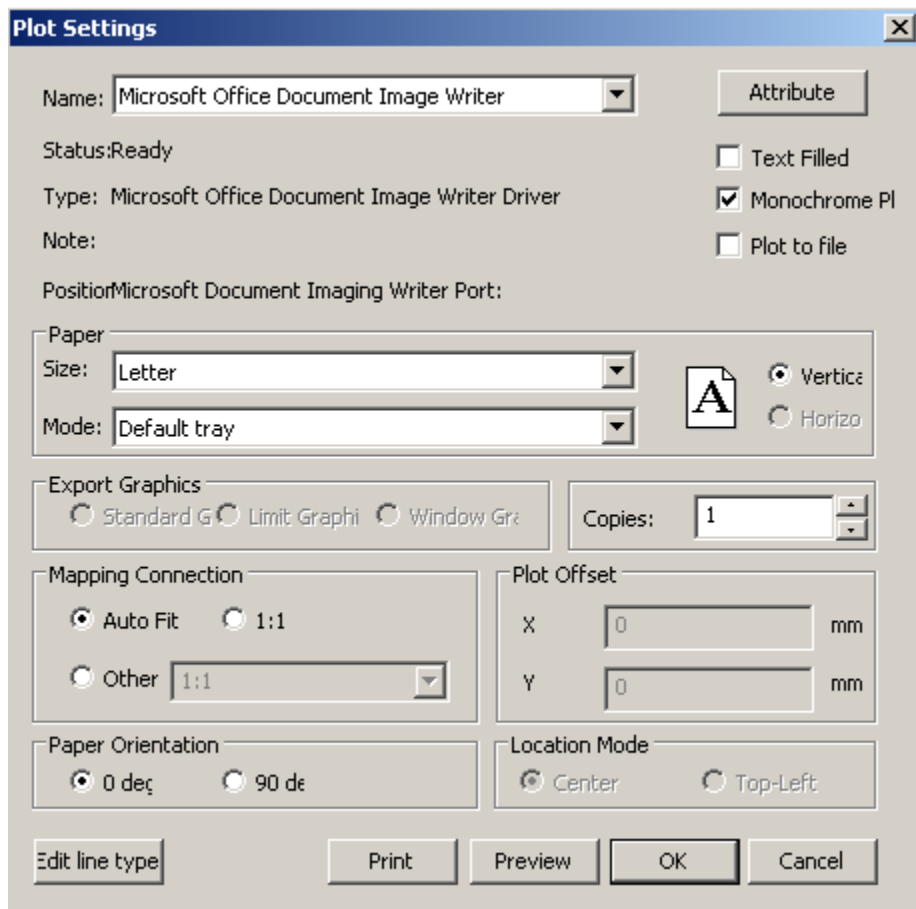
(8) Paper check

Check whether the drawing exceeds paper setting or not, in order to avoid displacement.

Click button Paper check in the main menu of typeset or panel of Typeset function, or select typeset print task unit, then click right key, and select “Paper check” , user can also click right key in the blank area of preview area to select button Paper check.

(9) Typeset paper unit print setting

Click Print setting in the main menu of file or button  in Organization panel. Or select typeset print task unit, then click right key, or click right key in the preview area, in the pop up menu, select option , as shown in following figure.





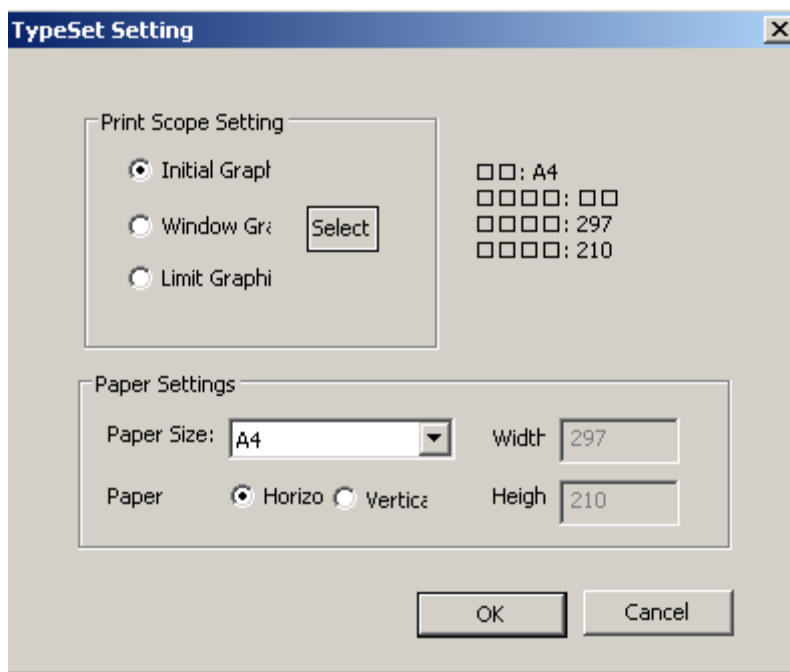
Pic 7-21 print settings of typeset print

In the above dialog box, user can set print parametric, such as printer setting, paper setting, mapping, graph direction, print deviation, linear setting etc. the concrete operation is the save as that in section 8.1.

(10) Drawing setting

By default, the drawing to be inserted to typeset print task unit will be inserted as per paper size itself. If it needs to adjust print area, user can use Paper setting .

Click paper setting in the main menu of file or button  in the Organization function area, or select typeset print task unit, then click right key, or click right key in the preview area. In the pop up menu, select button , following dialog box will pop up.



Pic 7-22 paper settings of typeset print

In the above dialog box, user can set Paper scope and set paper, the detail is shown as follows.

【Paper scope】 it indicates setting scope for output graph. Paper scope includes following three modes:

- Initial graph:

By default, the inserted drawing is inserted as per standard paper size. In the dialog box of 7-22, the default option is Initial graph, paper information is shown on the right side.

- Window graph:

Specify output graph scope in the form of rectangle window. Click Selection then specify two angle point of rectangle. Then click OK to confirm.

- Limit graph


Limit scope as per graph in the file.

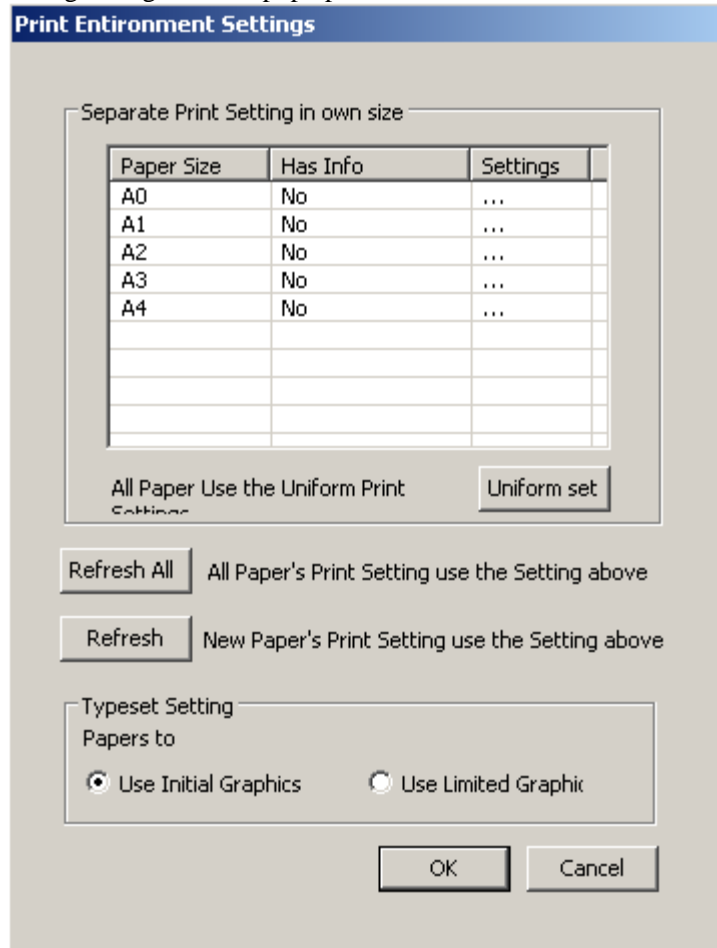
【Paper setting】 it indicates occupied space for file in typeset print paper .Paper setting includes parametric of drawing paper and paper direction, eg. Initial graph is a drawing of A2 paper, if A3 is selected in the drawing paper, the graph will be printed as A3 when typeset print.

7.2.5.3 Print environment configuration

When organizing graph printing by CAXA Draft Printing tool, the system can match print

setting automatically as per paper information itself.

Click Superior setting in the main menu of file or click button  in File panel in the function area, following dialog box will pop up.



Pic 7-23 print environment settings

Following is the explanation to each parametric.

- Match print setting as per paper size

User can set drawing to be inserted to print task table as per paper information itself, to match print setting. The default paper is : A0、 A1、 A2、 A3、 A4, double click one paper, in the pop up dialog box of Print setting, user can set parametric and confirm. Then, the corresponding answer“no”will be changed to “yes” to the question “whether there is information”.

If the default option “no”is not selected, then A4, and auto-fill will be adopted by default.

Click Unify Setting to select unified print setting for all drawings.

- Force refresh and partial refresh

If user adjusted task unit to be inserted to print task table, then configure print environment, user can choose tool setting modes, they are force refresh and partial refresh.

Force refresh: Use parametric in print environment configuration to refresh all print task unit in the list.

Partial refresh: keep the previous adjusted print setting, one refresh un-adjusted print task unit.

- Print typeset setting

For typeset print task, the inserted drawing, by default , can be typeset as per initial graph typeset and as per limit graph typeset.

7.2.6 Browse drawing

User can browser the inserted print unit in print task table in the preview area.

(1) In the print task list, click left key to select print unit, user can check print unit preview inform in the preview area. For individual print unit, it will be shown in preview area directly. For typeset print unit, user can preview paper information or preview real graph information by real display.


(2) Check graph in the preview area, use mouse-wheel to zoom in or zoom out preview graph, double click mouse mid key to display whole graph. Or user can use corresponding button in Display panel to check graph, including: display window, display whole, dynamic pan, dynamic zoom, display previous, display next.

(3) Click right key on the graph in the preview area, in the pop up menu, select “display previous”, and “display next”.

7.2.7 Print

Insert drawing and set corresponding parametric, print graph by printing equipment. Following is commands for printing.

(1) Print


Select one unit in the print typeset task unit, click Print in the main menu of file or in the panel of organize function area, or click button  in the quickstart tool bar, following dialog box will pop up.



Pic7-24 print dialog box


Click OK to print as per set parametric, click Cancel to cancel print.

(2) Print preview

Select one unit in the print typeset task list, click Print preview in the main menu of file or in the Organization panel of function area, or click button  in the quickstart tool bar. In the pop up preview window, user can check graph and print it directly.

Select print unit, then in the dialog box of print setting, user can click print directly or click Preview.

(3) Print all

Click Print all in the main menu of file, or click button  in the file panel of function area to start up Print all, the system will print all tasks in the print task list directly.

Chapter 8 System setting

In this chapter, we will mainly introduce system setting, such as layer, linear, color, point and select setting, style control, system configuration, interface configuration etc.

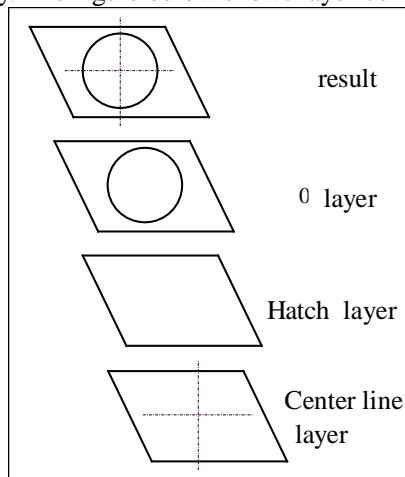
User can adjust these settings to meet all kinds of drawing request.

8.1 Layer

8.1.1 Summary

Layer is an indispensable software environment to develop structurization design. It's known that a mechanical engineer drawing contains various information, such as geometry information of entity shape, non-geometry information of properties for line type and color, various dimension and symbol. If related information can be collected, or collect a certain part and module together to draw or edit individually, extract them individually or combinatorially if necessary, which makes design more simple and easier. The layer introduced in this chapter has such function.

Imagine the layer as a translucent slice without thickness, entity and information are saved in it. In CAXA Draft, each layer must have its unique name. Different line type and color or other information can be designed on different layers. It is located by world coordinates between layers, all layers can be folded orderly. The figure below shows layer concept vividly.



Pic8-1 layer

The coordinates for each layer are united, What's more, zoom modulus are united, so all layer can match completely. A point on one layer will correspond at the same position of other layers precisely.

Layer has its own status, including layer name, layer description, line type, color, opening and

closing etc. The status can be changed. Each layer corresponds to a color and line type set by system. It is regulated the initial layer is 0 layer when being activated, the line type is thick real line. Modify line type and color of entity on layer through edit in the main menu, or as per latter introduced method in this chapter or via permanent menu.

Layer can be set up , be deleted, be opened or be closed. Entity on open layer can be seen on screen, the closed one won't be shown on screen.

The system has pre-defined 7 layers, they are 0 layer, centre line layer, dash line layer ,thin real line layer, dimension line layer, hatching layer and hide layer, each layer has its corresponding line type and color as per its name.

8.1.2 Layer setting




【Command】 layer

【Icon】 

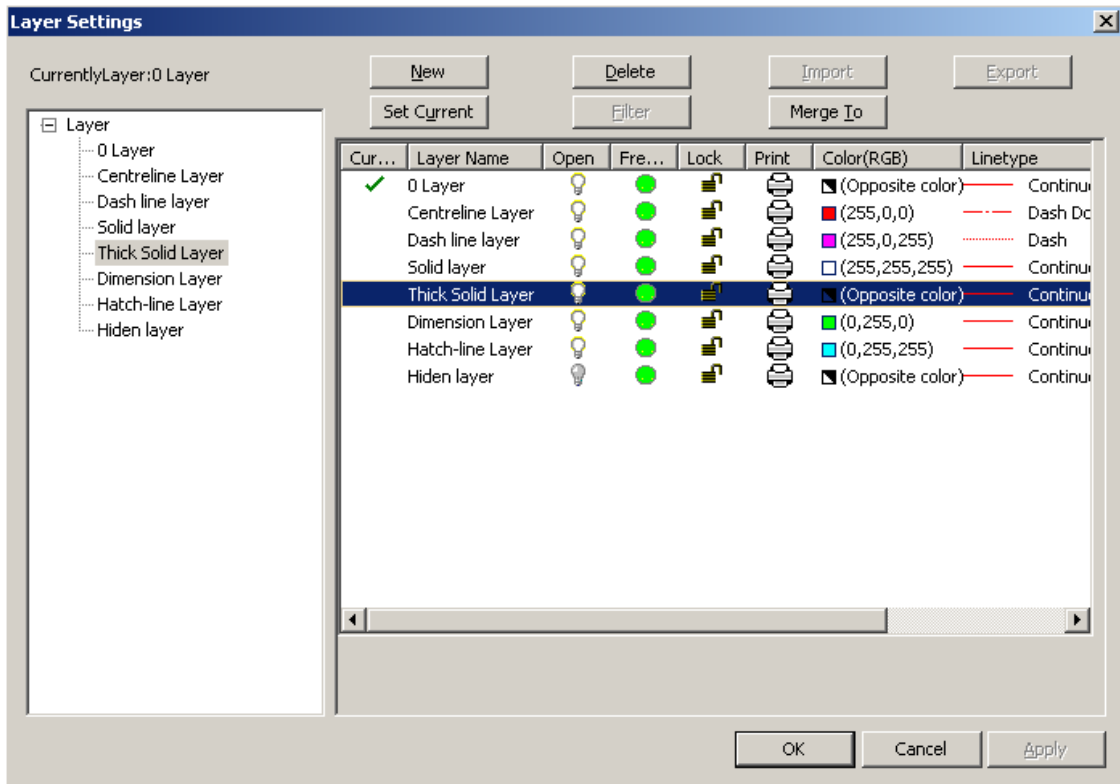
User can set layer by the command of Layer setting, the operation includes: set current layer, rename , new, delete, open/close, freeze/thaw, set color, set linear, set line width, lock layer, print layer.

When layer status is modified, all object in the layer will be updated.

【Step】

User can execute command Layer setting in forms of :click button  in the main menu of Format, click button  in the color layer, click button  in the attribute panel of Common option card, use Layer command.

Execute command Layer setting, following dialog box will pop up, as shown in figure 8-2.



Pic8-2 Layer settings

Detail introduction to each layer operation is shown as follows:

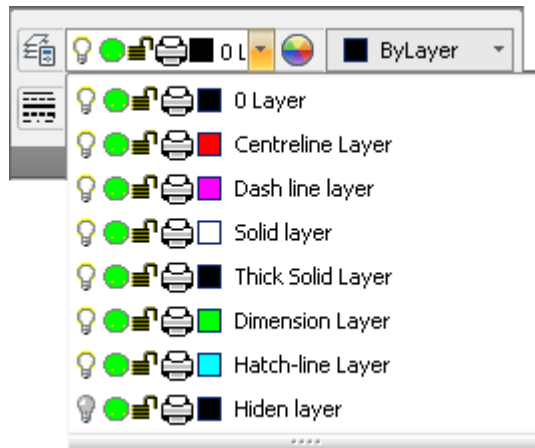
8.1.2.1 Set current layer

【Definition】: Set one layer as current layer, then the drawn element will be put on it.

There is only a unique current layer, all other layers are non-current layers.

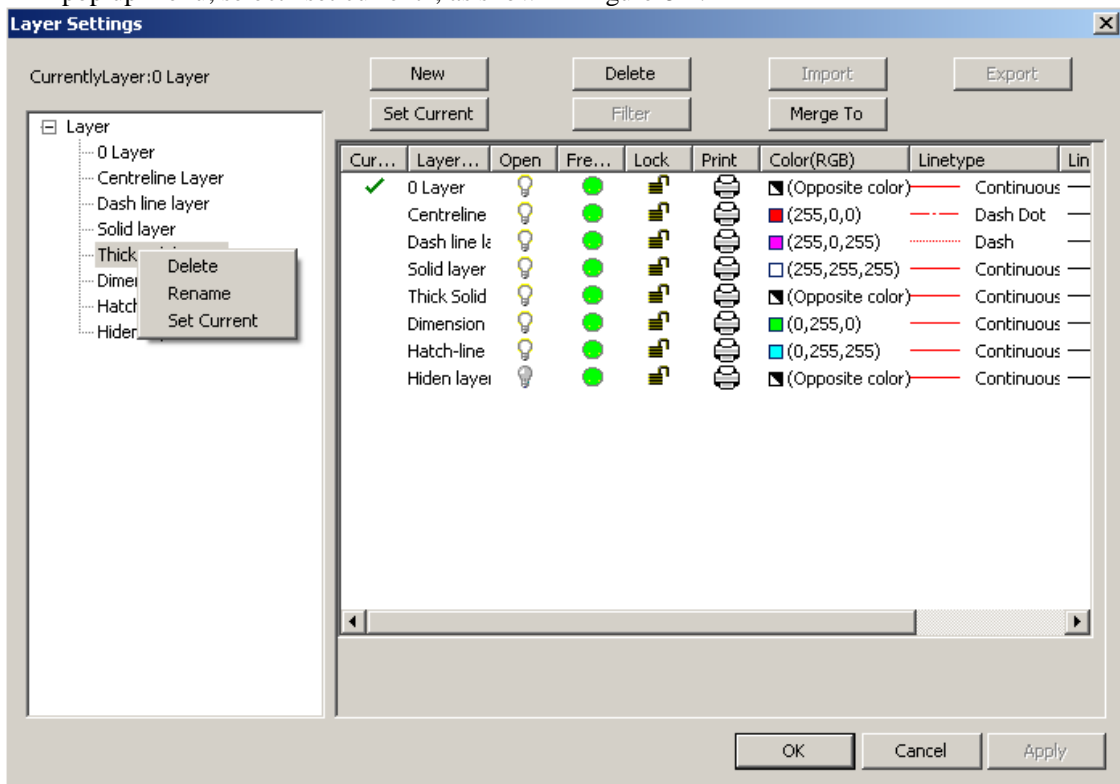
Current layer means the layer is being operated currently, it is also called motion layer. In order to operate existing graphics in a certain layer, this layer must be current layer. There are three methods to set current layer:

- (1) Click tool bar of Color layer, or click pull down list of Attribute panel in Command option card, Layer list will pop up, click left key in the desired layer to select current layer and operate. As shown in figure 8-3.



Pic8-3 Set current layer

- (2) In figure 8-2, select layer needs to set, then click button “set current”.
- (3) Click a Layer in the layer list on the left side of dialog box, and click right key, in the pop up menu, select “set current”, as shown in figure 8-4.



Pic8-4 Set current layer with menu

8.1.2.2 Rename layer

【Definition】: Change an existing name of a layer.

Layer title has two parts, they are layer name and layer description. Layer name is symbol of layer, by which the layers can be distinguished, so each layers should have different name. Layer description means describing layer character, different layer can have the same description.

The step for changing layer name:

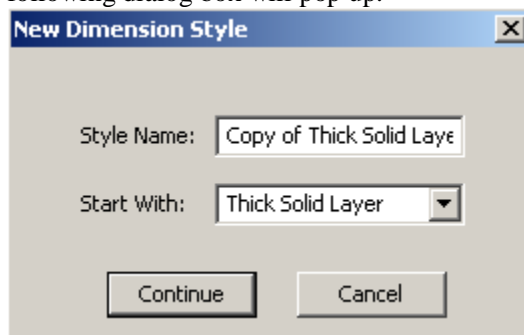
- (1) Execute command Layer setting, Layer setting dialog box will pop up, as shown in 8-2.
- (2) Select layer that needs to change name on the layer list in the dialog box, then click right key, in the pop up menu, select Rename, as shown in figure 8-4.
- (3) Then the layer name can be edited, input character and click blank area in the dialog box to confirm.

8.1.2.3 Create layer

【Definition】: Create a new layer.

【Step】:

- (1) Execute Command Layer setting, dialog box of layer setting will pop up, as shown in figure 8-2.
- (2) Click New, following dialog box will pop up.



Pic8-5 Create layer

Input a layer name, and select one reference layer, then click Next, the newly created layer will be shown under the lowest line in the layer list. By default, the newly created layer will adopt setting of selected reference layer.

8.1.2.4 Delete layer

【Definition】: Delete a layer created by user.

【Step】:

- (1) Execute Command Layer setting, dialog box of layer setting will pop up, as shown in figure 8-2.
- (2) Select layer to be deleted, click button delete layer, a warning dialogue box will pop up, click Yes .
- (3) Select layer to be deleted on the left side of the layer list, click right key, select Delete in the pop up menu to confirm.



It should be noted when deleting layer.

- Only newly created layer can be deleted, the original layer in the system can't be deleted.
- The current layer can't be deleted.
- If there are some graph are being used in the layer, then this layer can't be deleted.

8.1.2.5 Open and close layer

【Definition】: Open and close one layer

【Step】:

- (1) Execute Command Layer setting, dialog box of layer setting will pop up, as shown in figure 8-2.
- (2) In the layer status  of layer to be opened or closed, click  to switch open to close for the layer.

It should be noted when Opening or closing layer.

- Current layer can't be closed.
- When dialogue box layer control pops up, move the mouse to the location of layer status (open/close) to be changed, switch open and close by clicking left key. But the current layer can't be closed. When the layer is open, entity on this layer can be shown in the drawing area. If it is closed, the existing entity is invisible.
- Function of open and close layer is very useful for complicated drawing. Hide some irrespective entity when drawing complicated multiple view, which will make the drawing area more legible and tidy, and speed up drawing. When the drawing is finished, open it again and display all content.



● For example, put the dimension and hatching on dimension line layer and hatching layer respectively, close it when modifying view, then the view will be more legible. Also, put some auxiliary line on hide layer, when drawing is finished, close it and hide auxiliary line, it doesn't need to delete them one by one. There are many using skills, try to learn and accumulate in practice to improve work.

8.1.2.6 Freeze or Thaw layer

【Definition】: Freeze or thaw a certain layer.

Object in the frozen layer is invisible, what's more, it won't cover other object. For large graph, freeze unneeded layer, it will speed up operation of display and re-new. When Freezing one or more layers, the graph maybe created again. It needs more time to freeze or thaw layer than operation of open and close layer.

【Step】:

- (1) Execute Command Layer setting, dialog box of layer setting will pop up, as shown in figure 8-2.
- (2) In the layer status  of layer to be frozen or thawed, click  to switch freeze to thaw for the layer.

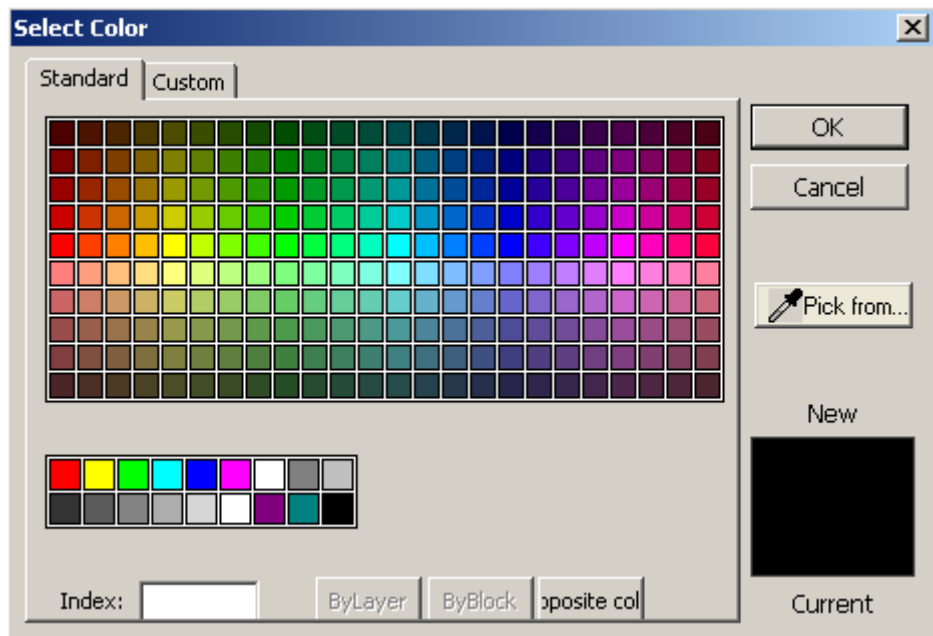
8.1.2.7 Layer color

【Definition】: Set layer color. User can set one color for each layer, and the color can be changed.

Different color can be set for commonly used layer. Following is introduction of changing layer color.

【Step】:

- (1) Execute Command Layer setting, dialog box of layer setting will pop up, as shown in figure 8-2.
- (2) In the layer status that needs to change color, click button Color, following dialog box will pop up, as shown in figure 8-6.



Pic8-6 select color

User can select color as per request, then click OK to return to Layer control dialog box, at that time, the corresponding layer color will be the selected color.

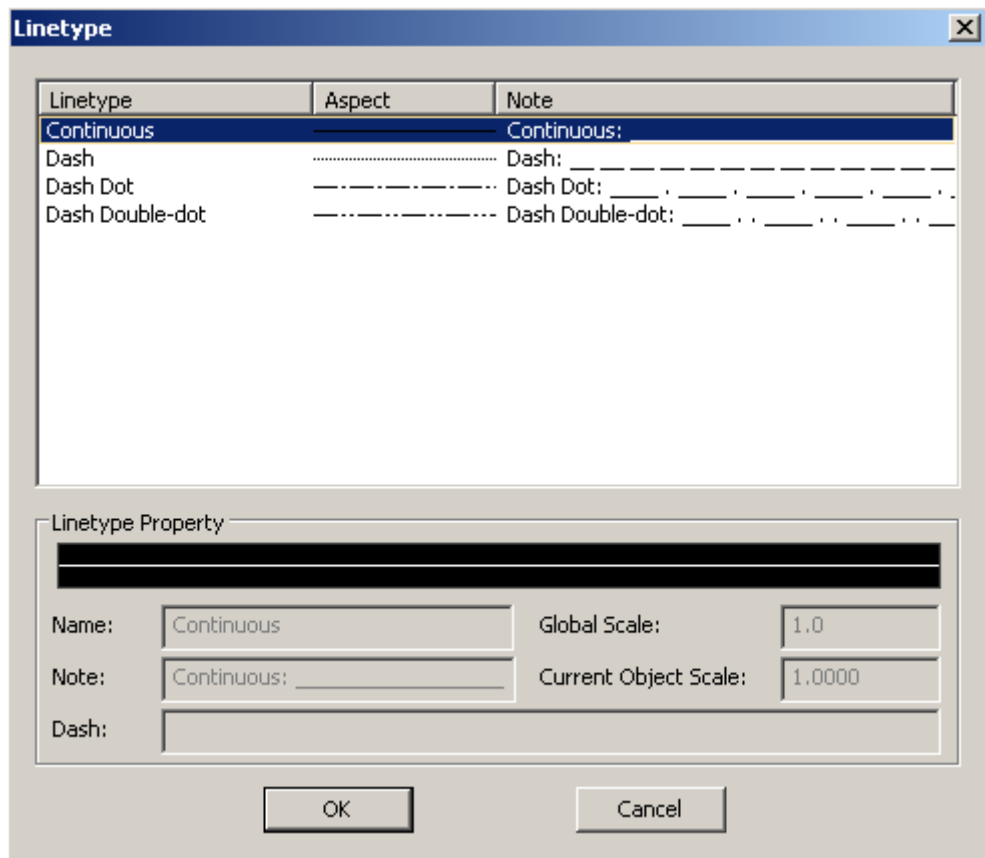
8.1.2.8 Lay linear

【Definition】: Set linear for the selected layer.

User can set different linear for existing layer. All these linears can be set again by this function.

【Step】:

- (1) Execute Command Layer setting, dialog box of layer setting will pop up, as shown in figure 8-2.
- (2) In the layer status that needs to change linear, click button Linear, following dialog box will pop up, as shown in figure 8-7.






Pic8-7 select linetype

User can select linear as per request, then click OK to return to Layer control dialog box, at that time, the corresponding layer linear will be the selected linear.

8.1.2.9 Lock or unlock layer

【Definition】: Lock selected layer




【Step】:

- (1) Execute Command Layer setting, dialog box of layer setting will pop up, as shown in figure 8-2.
- (2) In the layer status  of layer to be locked or unlocked, click  to switch lock to unlock for the layer. When the layer is locked, the icon will be changed to , user can only add element to the locked layer, select element ,user can copy, paste, array, query attribute etc in the locked layer. But it is unable to delete, pan, stretch, zoom in or zoom out, attribute modification, create block etc. It doesn't limit element in title bar, specification and frame.。

8.1.2.10 Layer print setting

【Definition】: Select whether print content in the selected layer or not.

【Step】:

- (1) Execute Command Layer setting, dialog box of layer setting will pop up, as shown in figure 8-2.
- (2) In the layer status  of layer to be printed or unprinted, click  to switch print to unprint for the layer. The icon for unprint status is , then when print, content in this layer won't be plot. Auxiliary line in the drawing won't be printed.

8.2 Linear

【Name】 Linear setting




【Command】 ltype

【Icon】 

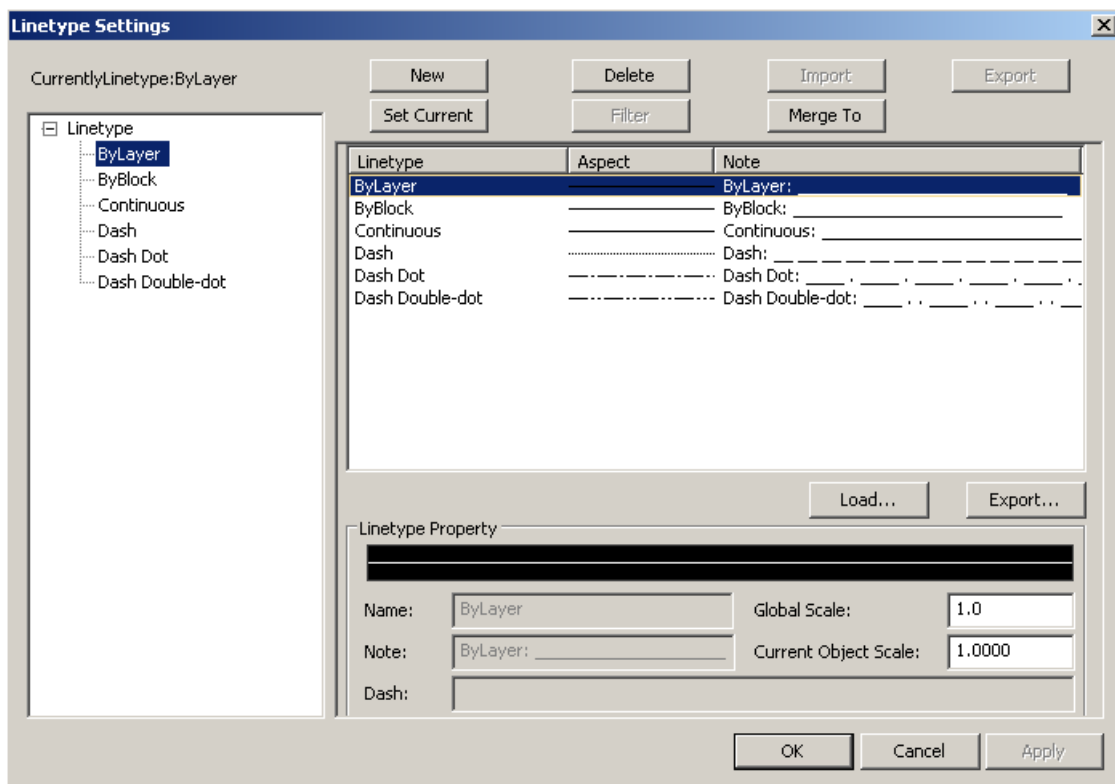
【Definition】 set and management system linear.

User can manage and set linear by the command Line setting, such as set current linear, modify linear, new, delete etc.

【Step】

User can execute command Linear setting in forms of :click button  in the main menu of Format, click button  in the color layer, click button  in the attribute panel of Common option card, use Ltype command.

Execute command Linear setting, following dialog box will pop up, as shown in figure 8-8.



Pic8-8 linetype settings

Following is detail introduction for linear setting.

8.2.1 Set current linear

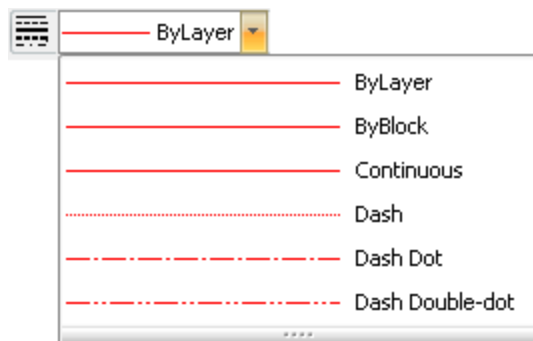
【Definition】: Set a linear as current one, then all element to drawn will use such linear.

It should be noted when setting current linear.

- Set current linear as bylayer: user current layer linear to draw.
- Set current linear as byblock: when the drawn element is defined as block, it will use linear selected by the block.
- Remove bylayer or byblock as current linear: selected linear will be used to the element to be drawn.

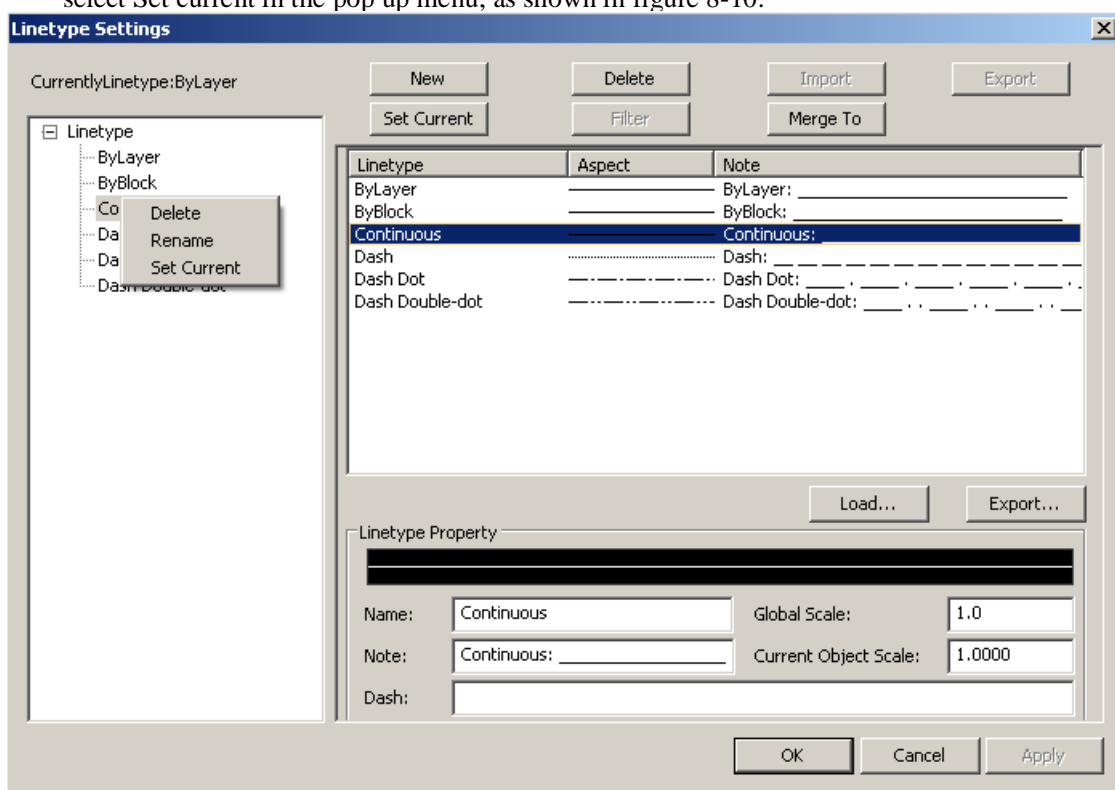
Method for setting current linear.

- (1) Click tool bar of Color Layer or click pull down list of Attribute panel in Common option card, linear list will pop up, click left key to select desired linear in the list, the selected linear will be current one, as shown in figure 8-9.



Pic8-9 set current linetype

- (2) In the dialog box of figure 8-8, select linear to be set , then click Set current.
- (3) In figure 8-8, select linear in the list on the left side of dialog box, click right key, select Set current in the pop up menu, as shown in figure 8-10.



Pic8-10 set current linetype with menu

8.2.2 Modify linear

【Definition】: Modify parametric of existing linear.

Linear parametric includes name, note, overall scale gene, current object scaling, space etc. Bylayer and byblock in the linear dialog box can't be modified.

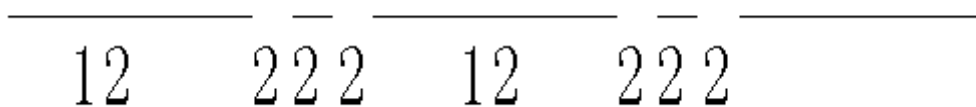
【Step】

- (1) Execute command Linear setting, dialog box of figure 8-8 will pop up.
- (2) Select one linear in the dialog box, all parametric under linear information can be edited. The meaning and modification method of linear information is shown as follows:

- **Name:** Set a name for selected linear. User can input it directly, or select one linear in the linear list on the left side dialog box, then click right key, in the pop up menu, select Rename to input.
- **Note:** Input note for the selected linear, user can input it directly.
- **Overall scale gene:** Modify all linear scale gene in the graph.
- **Current linear scaling:** set linear scale gene being edited. Scale gene for all linear is the product of overall scale gene and the linear scaling.
- **Space:** Input current linear code, the code can be organized at most by 16 digits. Each number represent pixel value of stroke or space distance.

Odd number digit represent stroke length, even number digit represent space length. Number 0 represents one pixel. There is a comma between stroke and space, linear code digits must be even number.

For example, space number is 12,2,2 for line drawn by dot, linear result is shown as in figure 8-11.



Pic8-11 example

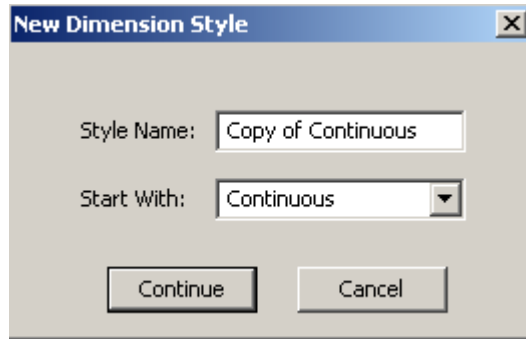
- (3) When parametric is modified, click OK to confirm.

8.2.3 Create linear

【Definition】: Create a new linear

【Step】:

- (1) Execute command Linear setting, dialog box of figure 8-8 will pop up.
- (2) Click button New, following dialog box will pop up.



Pic8-12 Create linear

Insert a linear name, and select a reference linear, click Next, user can see newly created linear at the lowest line in the linear list. By default, the newly created linear will use the setting of reference linear.

8.2.4 Delete linear

【Definition】: Delete one linear

【Step】:

- (1) Execute command Linear setting, dialog box of figure 8-8 will pop up.
- (2) Select linear to be deleted, click button Delete, in the pop up dialog box, select Yes to delete linear.
- (3) Or Select linear to be deleted in the linear list, then click right key, select button Delete in the pop up menu to delete.

Note:

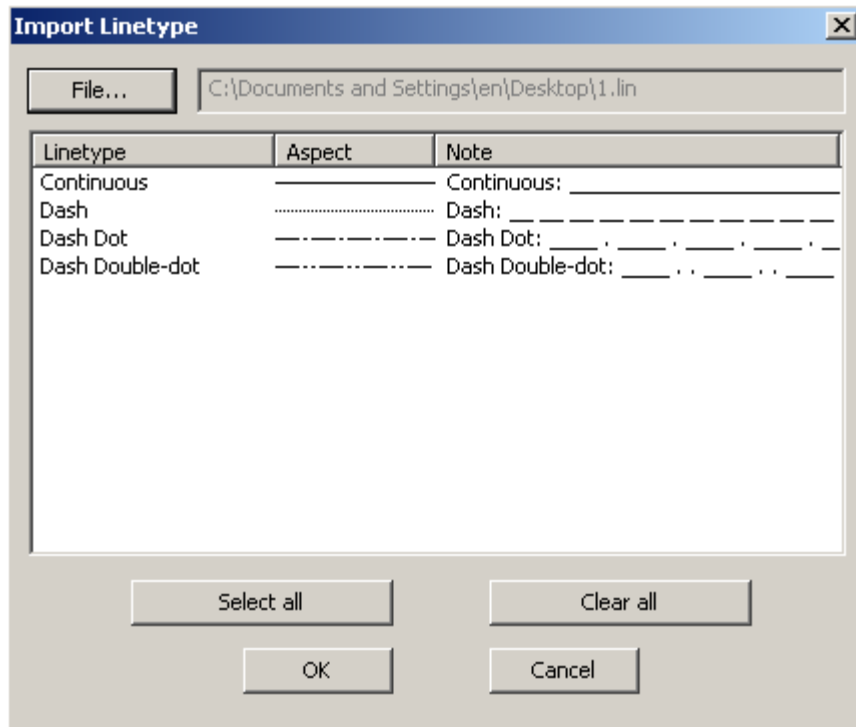
- User can only delete newly created linear, the original linear in the system can't be deleted.
- When the linear is set as current one, it can't be deleted.

8.2.5 Add-on linear

【Definition】: Load in linear from existing file.

【Step】:

- (1) Execute command Linear setting, dialog box of figure 8-8 will pop up.
- (2) Click button Add-on, following dialog box will pop up.



Pic8-13 import linetype

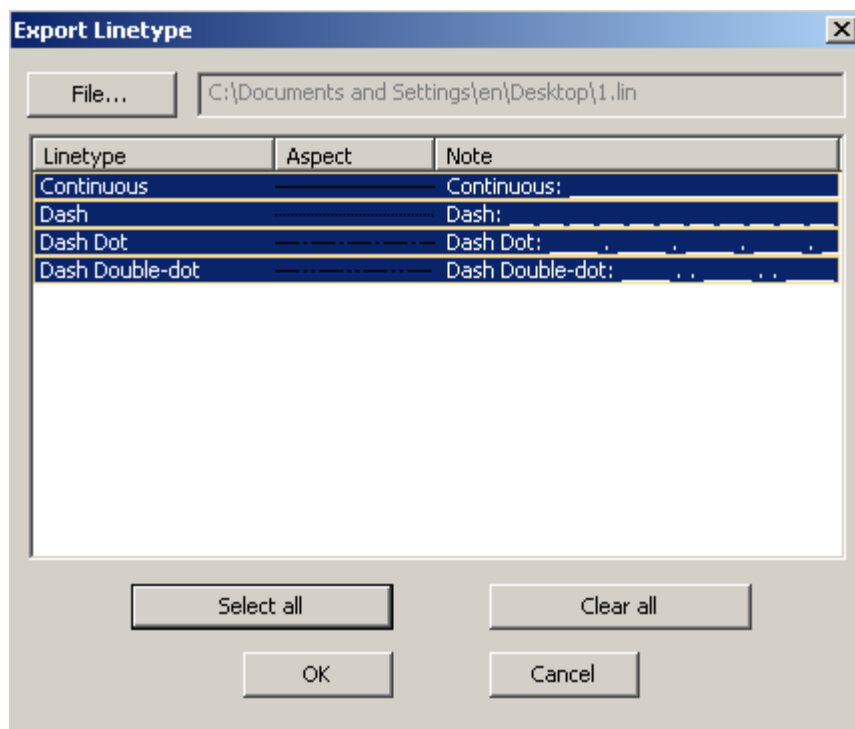
Click File and select a linear file, then select linear file to be add-on, click OK.

8.2.6 Output linear

【Definition】: Output existing linear to one linear file to save it.

【 Step 】:

- (1) Execute command **Linear** setting, dialog box of figure 8-8 will pop up.
- (2) Click button **Output**, following dialog box will pop up.



Pic8-14 export linetype

Click File and select a linear file, then select linear file to be output, click OK.

8.3 Color setting




【Command】 color

【Icon】 

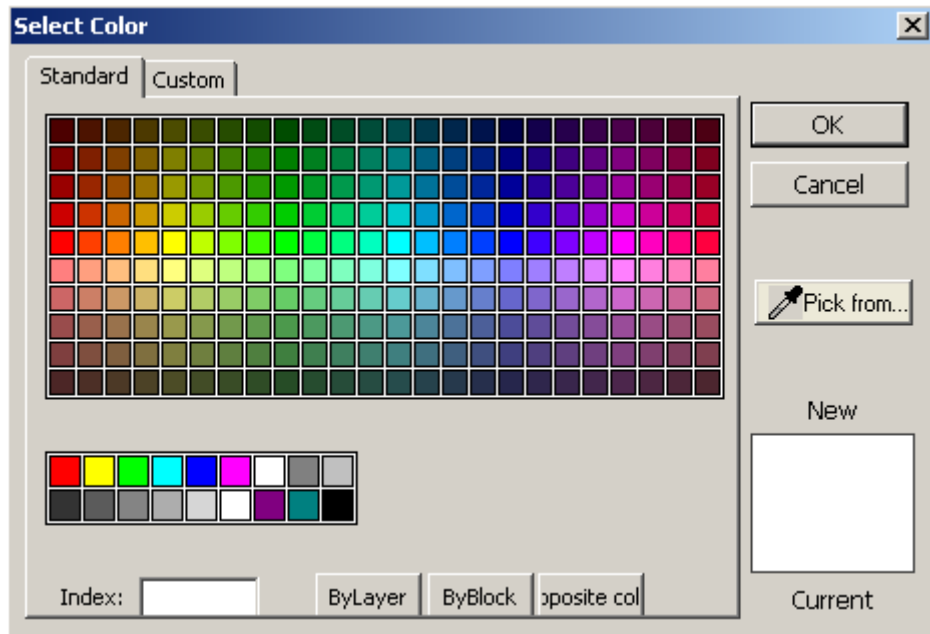
【Definition】 set and manage system color.

In CAXA Draft, user can manage and set color by command Color, such as use standard color, user customized color etc.

【Step】

User can execute command Color setting in forms of :click button  in the main menu of Format, click button  in the color layer, click button  in the attribute panel of Common option card, use Color command.

Execute command Color, following dialog box will pop up.




Pic8-15 select color

Following is detain introduction of color setting.

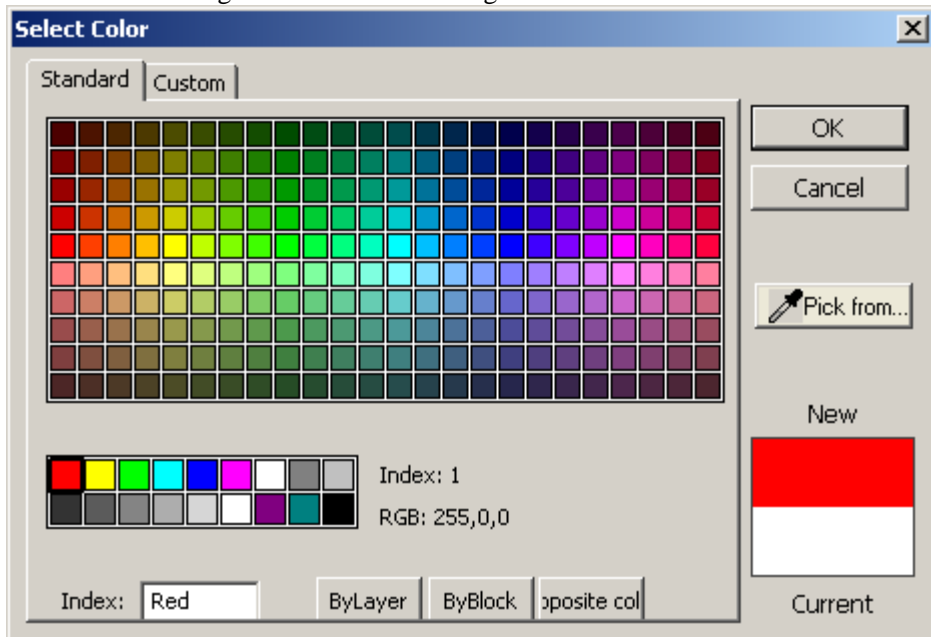
8.3.1 Use standard color

【Definition】: Use standard color and set as current color

【Step】:

- (1) Execute command Color , dialog box of figure 8-15 will pop up, by default, it use standard color.
- (2) Select one color in the dialog box, the selectable color includes:
 - Index color: Click color cell to use color in index option card.
 - Bylayer: Click button **【bylayer】** to use color specified color for current layer.
 - Byblock: Click button **【byblock】** to use byblock color, create object and set is as block, the object color will be the same as that of block.
 - Black and white: Click button Black and white , when the system background is white, the object drawn will be black, whereas, if the system background is black, the object drawn will be white.
 - From Screen: Click button From screen, the cursor will be changed to , then click one point on the screen to select one color.
- (3) Select one color, it will prompt Index name, the selected color and current color will

be shown at the low right side. As shown in figure 8-16.



Pic8-16 show selected color

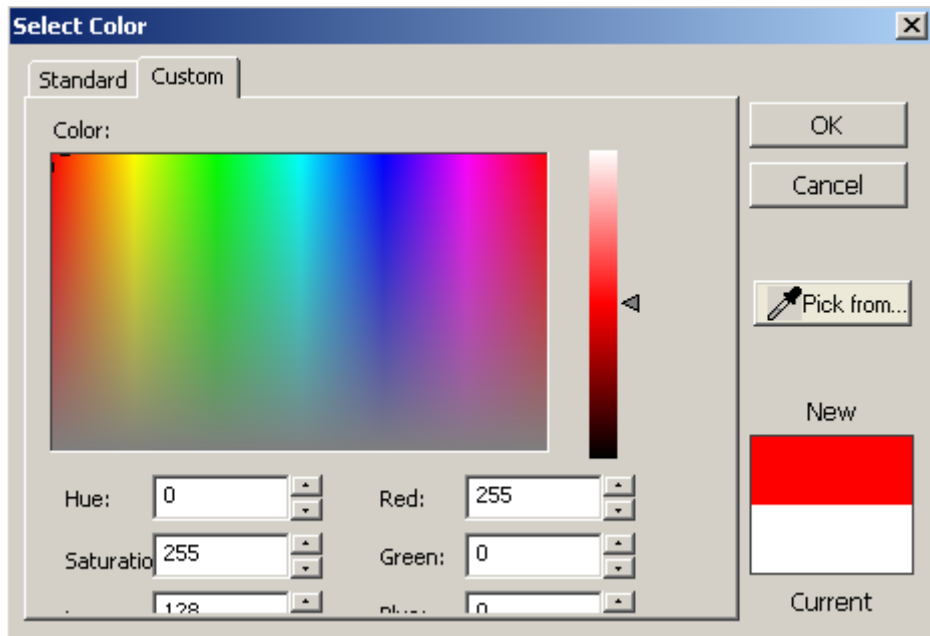
- (4) Click button Ok, the current color will be set as selected one.

8.3.2 Use customized color

【Definition】: Use standard color and set it as current color.


【Step】:


- (1) Execute command Color , dialog box of figure 8-15 will pop up, by default, it use standard color.
- (2) Click option Customize, following dialog box will pop up.



Pic8-17 custom color

Following are mode of customize color:

- Click left key under the command Color.
- Use HSL mode, specify numeric value for Hue, Saturation, Brightness.
- User RGB mode, specify numeric value for Red, Blue, Blue.
- Click button From screen, the cursor will be , then click on the screen to select one color.

When customizing color, user can drag button  on the right to match color customizing.

(3) Select one color, it will prompt Index name, the selected color and current color will be shown at the low right side. As shown in figure8-16.

(4) Click button Ok, the current color will be set as selected one.

8.4 Line weight setting

Line weight setting includes Setting current line weight and setting line weight scale.

8.4.1 Set current line width

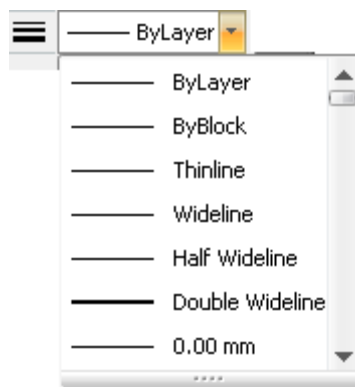
【Definition】: Set a certain line weight as current one, and this line weight will be used to the element to be drawn.

Note for setting current line width.

- Set current line weight as Bylayer, then line weight in current layer will be used to element to be drawn.
- Set current line weight as byblock: when element is defined as block, the block line weight will be used.
- Set current line weight as Remove bylayer or Byblock, then the selected line weight will be used when drawing.
- Thin line and thick line: user can set display scale and print parametric individually.

Method for setting current line width.

Click tool bar of Color layer, or click pull down menu of Attribute panel in Common option card, Line weight list will pop up, click left key in the list to select desired line width. As shown in figure 8-18.



Pic8-18 set current lineweight

8.4.2 Set line weight scale

【Name】 Line weight setting

【Command】 wide

【Icon】

【Definition】 set line weight Zoom.

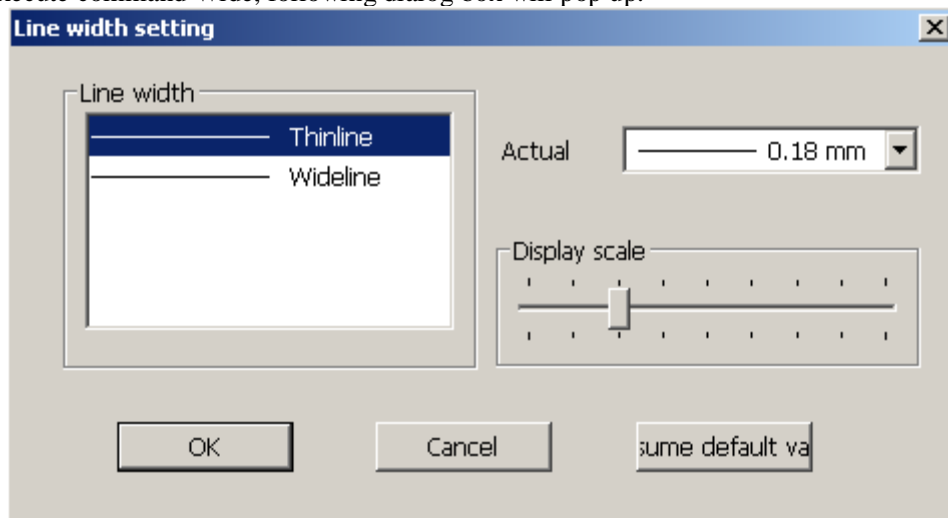
In CAXA Draft, user can set line weight by command Wide, and line weight zoom can also be set.

【Step】

User can execute command Wide in forms of :click button in the main menu of Format, click button in the color layer, click button in the attribute panel of Common option card,

use Wide command.

Execute command Wide, following dialog box will pop up.



Pic8-19 set lineweighgt scale

Method for setting and using Lineweight.

- Select Thin line or Thick line, then specify line weight for thin line or thick line in the Actual value on the right.
- Drag handle in Zoom to adjust line weight scale, increase zoom when dragging rightward, decrease zoom when dragging leftward.

8.5 Point and select setting

In CAXA Draft, multiple select and capture tools are supplied to increase efficiency. Following are details introduction of capture point setting, 3D view navigating, screen point setting, select and filtrate.

8.5.1 Capture setting




【Command】 potset

【Icon】

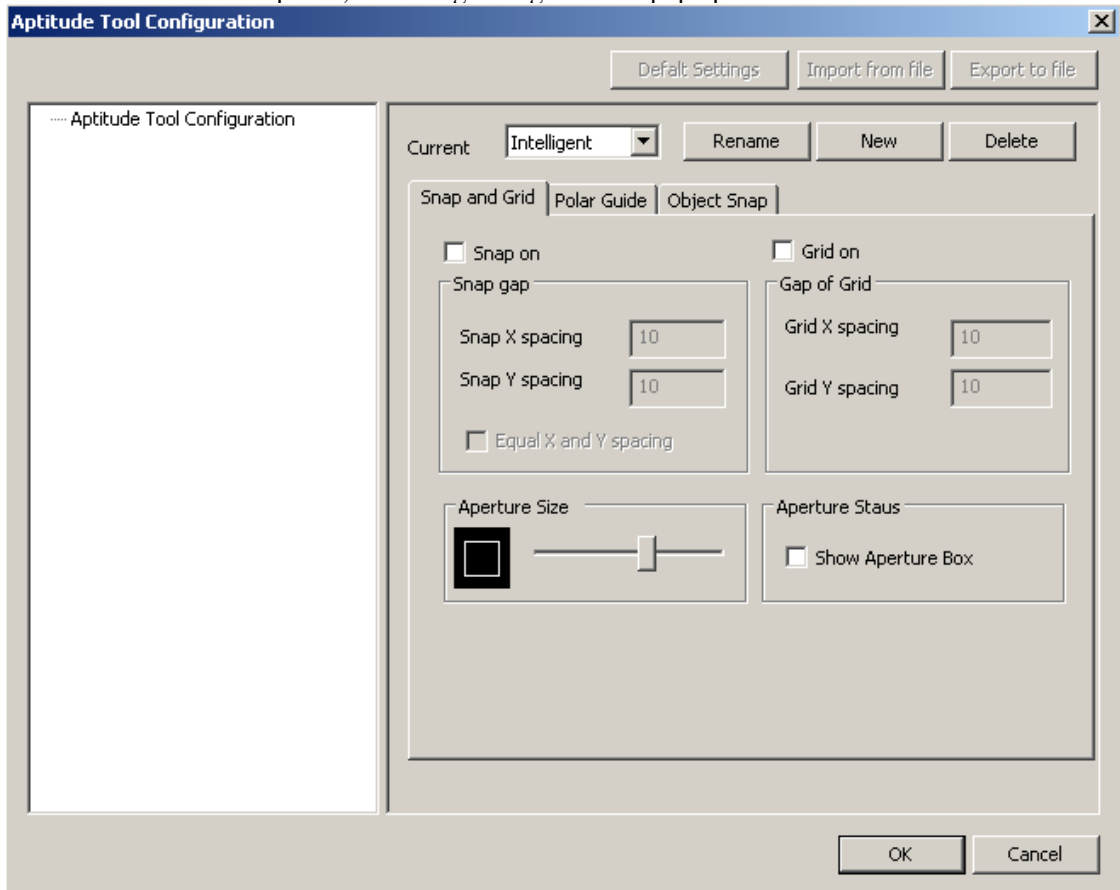
【Definition】 Set capture mode when the cursor is in the screen .

Capture mode includes: space grid, polar axis navigating, object capture. These modes can be combined as free capture, intellectual capture, grid capture, navigating capture etc.

【Step】

User can execute command potset in forms of :click button  in the main menu of Tool, click button  in the tool bar of setting tool, click button  in the attribute panel of Tool option card, right click Capture setting in the status bar, then select Set, use potset command.

Execute command potset, following dialog box will pop up



Pic8-20 snap and grid

Method for setting and using Capture.

(1) Capture and grid:

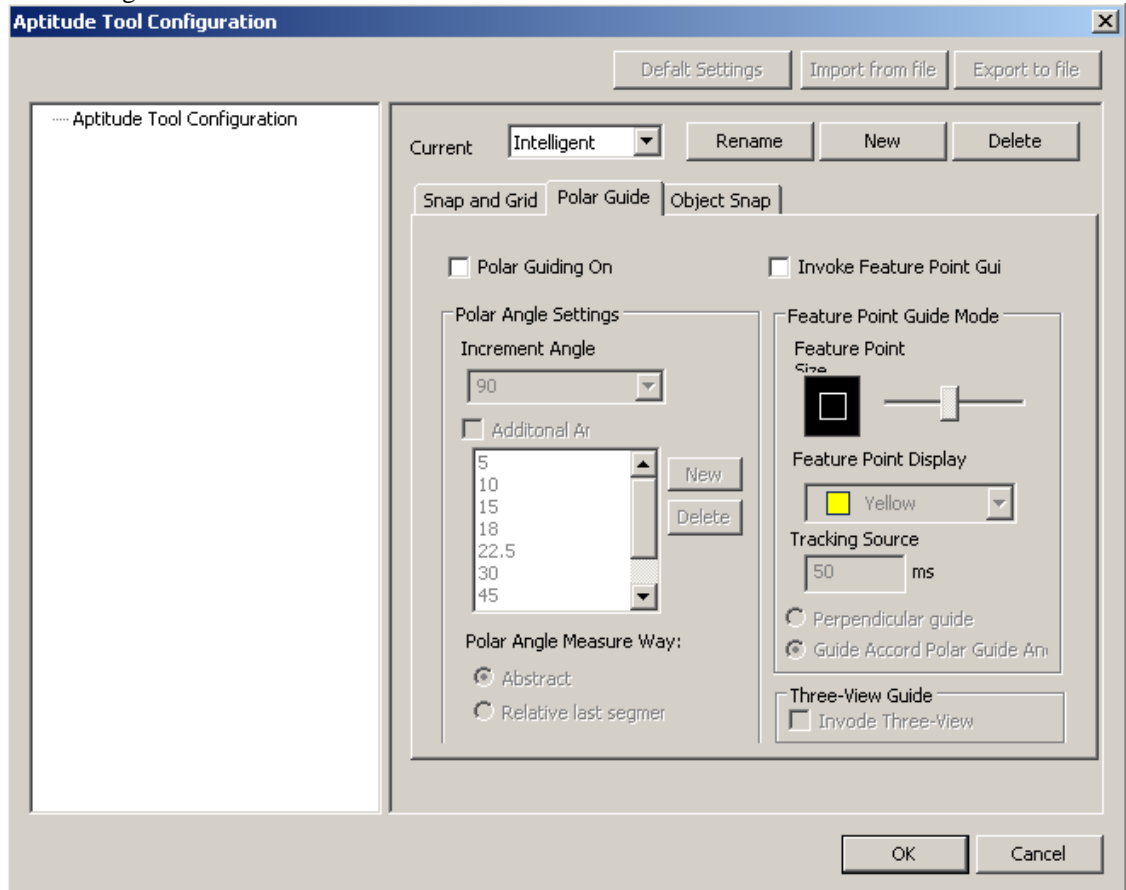
In the above dialog box, user can set Space capture and grid display by option Capture and grid. Following is detail method.

- Select Start capture to open space capture mode, then set capture space as per X axis and Y axis.
- Select Start Grid to open grid display, then set grid space as per X axis and Y axis.
- Drag handle under the Target size to set select frame size.
- Select option Display auto capture target frame in Target status to set showing target frame

when auto capture.

(2) Polar axis navigating:

Click Polar navigating in the dialog box of Capture setting to set polar navigating parametric, as shown in figure 8-21.



Pic8-21 polar guide

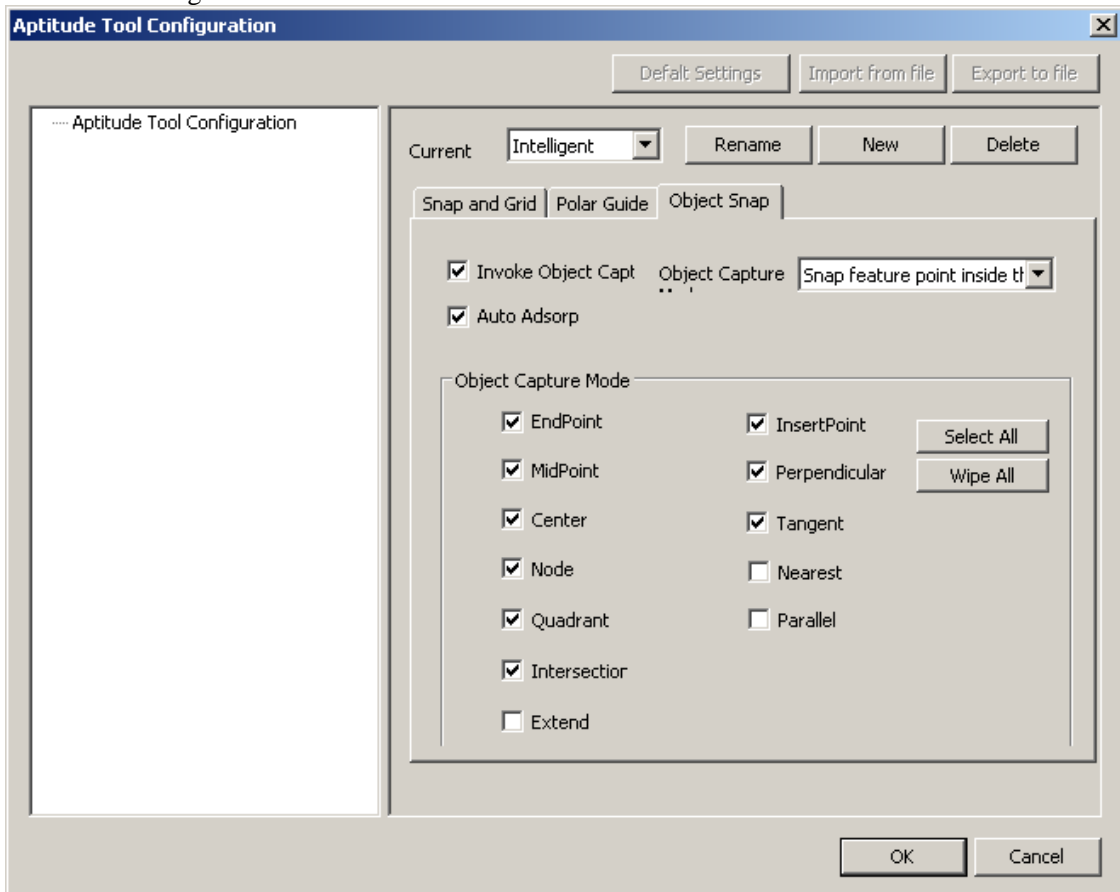
Method for setting Polar axis navigating:

- Click Start Polar axis navigating to open or close start polar navigating.
- Open Polar axis navigating, set polar axis angle parametric to specify alignment angle of polar axis navigating.【increment angle】 means setting polar angle increment which is used to display polar navigating alignment path, user can input any angle value, or select commonly used angle. 【additional angle】 means adding or deleting any additional angle in polar navigating list. 【Polar angle measurement mode】 includes absolute previous section and relative previous section.
- Select Start feature point navigating to set open feature point navigating mode. And user can

then set feature point size, color, navigating source activate time , and use 3D view navigating.

(3) Object capture.

Click Object capture in the dialog box of Capture setting to set Object capture parametric, as shown in figure 8-22.

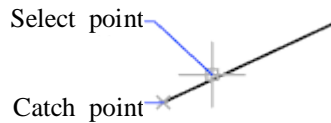


Pic8-22 Object snap

Method for setting Object capture:

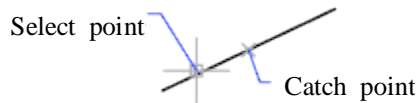
- Click option Start object capture to open or close object capture. When object capture mode is open, user can select “Capture feature point within cursor target” and “capture nearest feature point.
- Select Auto Adsorb to set cursor auto absorb when object capture.
- Feature for object capture.

【Start point】: capture the nearest end point to arc, line, polyline, spline. As shown in figure 8-23.



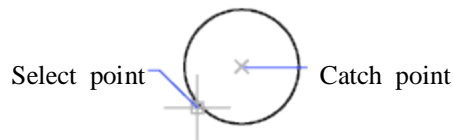
Pic8-23 Catch Start point

【Mid point】: capture the mid point to arc, line, polyline, spline. As shown in figure 8-24.



Pic8-24 Catch Mid point

【Center】: capture the center to arc, circle, ellipse, or ellipse arc . As shown in figure 8-25.



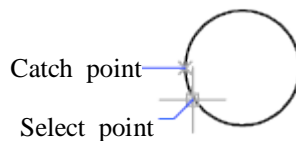
Pic8-25 Catch center point

【Node】: capture point object, dimension definition point, or dimension text origin. As shown in figure 8-26.



Pic8-26 Catch Node point

【Quadrant point】: Capture quadrant point to arc, circle, ellipse or ellipse arc. As shown in figure 8-27.



Pic8-27 Catch Quadrant point

【Intersection point】: Capture intersection point to arc, circle, ellipse, line, polyline, spline.

【Stretch】: When cursor passing object end point, extending line or arc will be temporally displayed, which will help user to specify point on extending line or arc, as shown in figure 8-28.



Pic8-28 Catch Intersection point

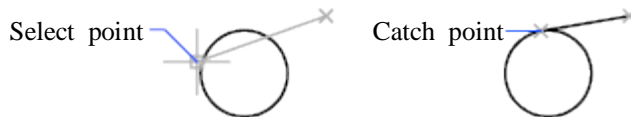
【Insert point】: capture insert point to attribute, block, text.

【perpendicular foot】: Capture perpendicular foot to arc, circle, ellipse, line ,polyline, spline. as shown in figure 8-29.



Pic8-29 Catch perpendicular foot point

【tangency point】: Capture tangency point to arc, circle, ellipse, spline. as shown in figure 8-30.



Pic8-30 Catch tangency point

【Nearest point】: Capture nearest point to arc, circle, ellipse, ellipse arc, line, point, polyline, radial, spline, reference line.

【Parallel】: Make line segment , polyline segment be parallel to other linear. Specify first point of linear object, then specify parallel object capture. It is different from other object capture. User can move cursor to other linear object , then one angle can be got. After that , move cursor to the object being created, if the object path is parallel to previous linear object, alignment part will be shown, which can be used to create parallel object.

(4) Capture mode:

User can set capture mode and its parametric flexibly, and combine them into multiple capture mode, as shown in figure 8-20, the system supplies several default capture mode for cursor.

- Free: Close all capture modes, such as capture and grid, polar axis navigation, object capture.

For dot input, it is completely decided by actual position of current cursor.

- Smart: only open object capture, the cursor will capture some feature points automatically, such as center, tangency point, perpendicular point, mid-point, end point etc.

- Grid: Only open capture and grid, cursor will capture grid point, set it visible or invisible.

- Navigation: Open polar navigation and object capture simultaneously. User can navigate by cursor for several feature point, such as isolated point, end point of line segment, middle point of line segment, circle center or arc quadrant point etc. Smart capture can be used when using navigating capture to enhance capture accuracy.

By default, the capture mode is Smart point capture. User can use hot key F6 to switch capture mode, or switch in list box of status bar.

In dialog box of 8-20, user can use command New, Delete, Rename to operate capture mode.

8.5.2 3D view navigation

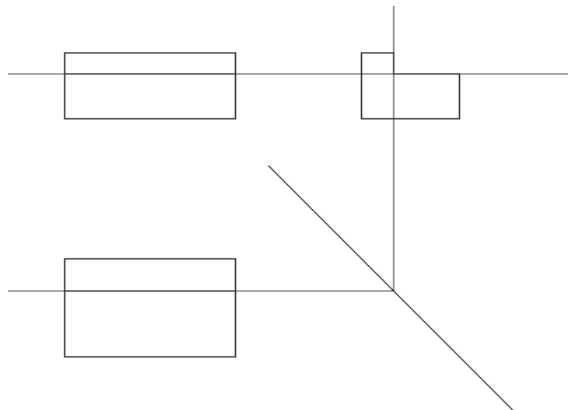
【Command】 guide

【Definition】 it is expansion of navigation, user can define projection relation conveniently. It can help to draw 3D view and multiple view.

【Step】

User can execute command Guide in forms of: click button Guide in the main menu of Tool.;Use F7; Use guide command.

【Example】:



Pic8-31 3D view navigation

8.5.3 Dot style

【Command】 ddptype

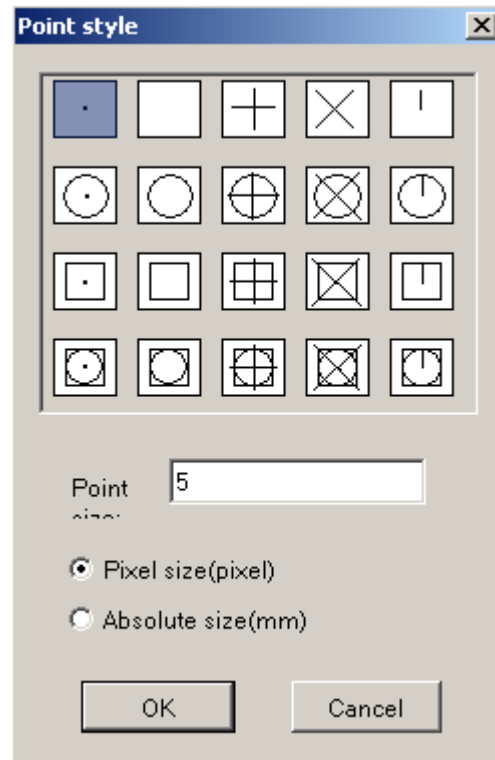
【Icon】

【Definition】 Set style and size for screen middle dot.

【Step】

User can execute command Ddptype in forms of: click button in the main menu of Format, click button in the tool bar of Setting tool, click button in the option panel of Tool option card. Use command ddptype.

Execute command Ddptype, following dialog box will pop up, as shown in figure 8-32.



Pic8-32 Point style

There are two parts in the above box:

1) Point type

20 types of points meet user's requirement.

2) Point size

Point size: pixel size and absolute size

Pixel size: pixel value is related to screen size

Absolute: actual size , with unit of millimeter




8.5.4 Pick filter

【Command】 objectset

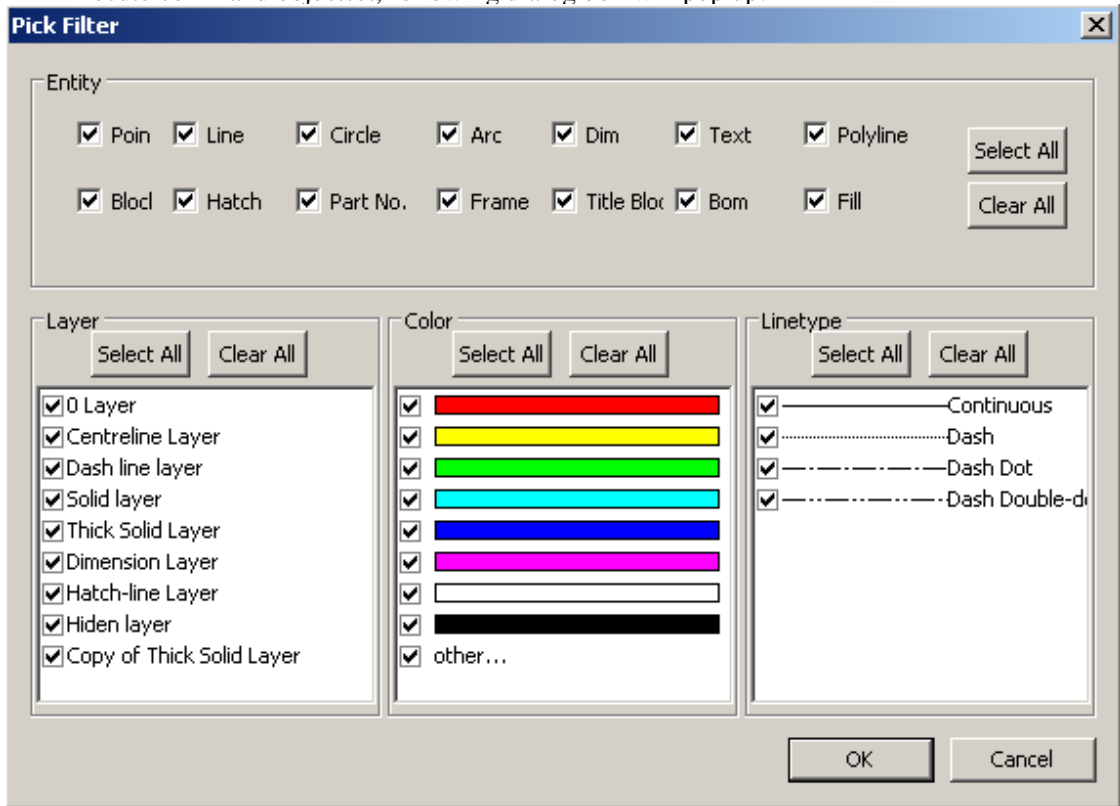
【Icon】 

【Definition】 Set filter condition for selecting element.

【Step】

User can execute command objectset in forms of: click button  in the main menu of Tool, click button  in the tool bar of Setting tool, click button  in the option panel of Tool option card. Use command objectset.

Execute command objectset, following dialog box will pop up:



Pic8-33 pick filter

There are 4 kinds of filter conditions: Entity, Layer, Line, color

The intersection of above four filter conditions is valid picking. User can pick the desired element correctly and promptly by the combined condition.

The default condition for pick filter is shown in the upper figure.

Select or cancel check box in front of each condition to add or filter select condition.




8.6 Style control

【Command】 type

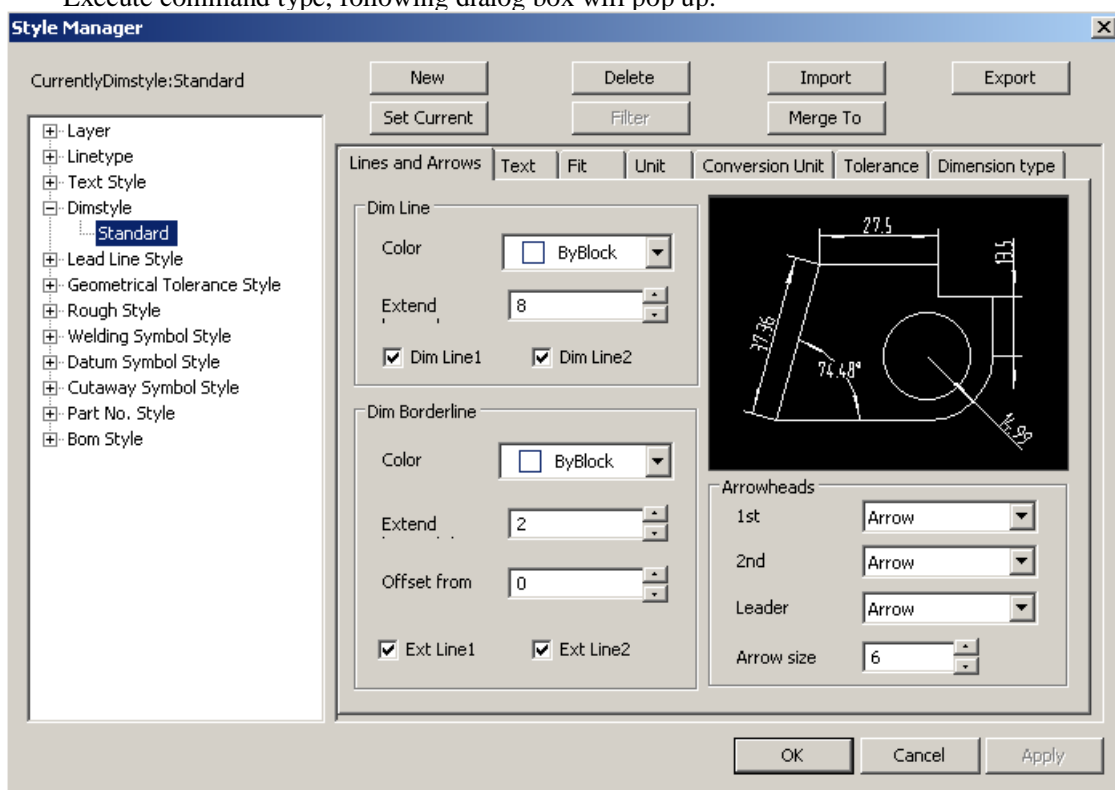
【Icon】 

【Definition】 Dimension style, text style and layer can be set centralized, and management function of merge, filtration and export etc. are provided.

【Step】

User can execute command type in forms of: click button  in the main menu of Format, click button  in the tool bar of Setting tool, click button  in the option panel of Common option card. Use command type, press Ctrl+T key .

Execute command type, following dialog box will pop up:

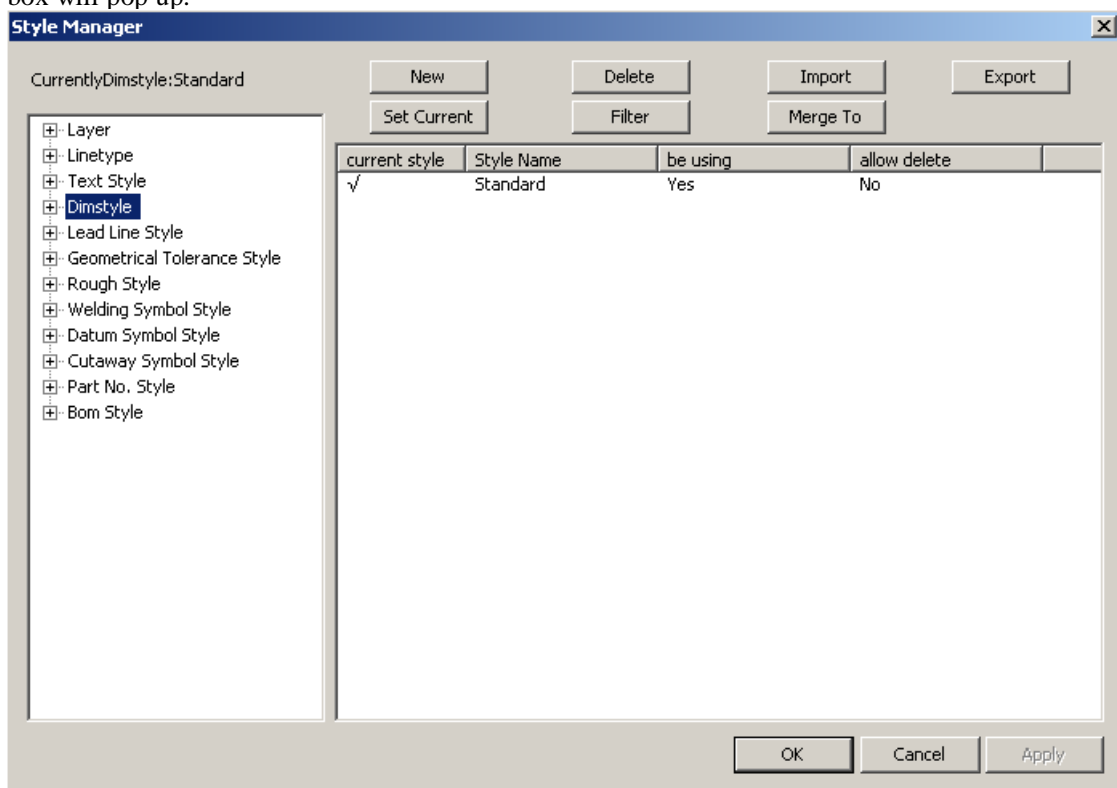


Pic8-34 style manager

In the dialog box of Type, user can set each parametric, lead in style or lead out style. Following is detail introduction.

8.6.1 Parametric setting

Click command Type, on the left side of the dialog box, it is style list, select one style, the style status will be shown on the right side, for example, select Dimension style, following dialog box will pop up.



Pic8-35 current style

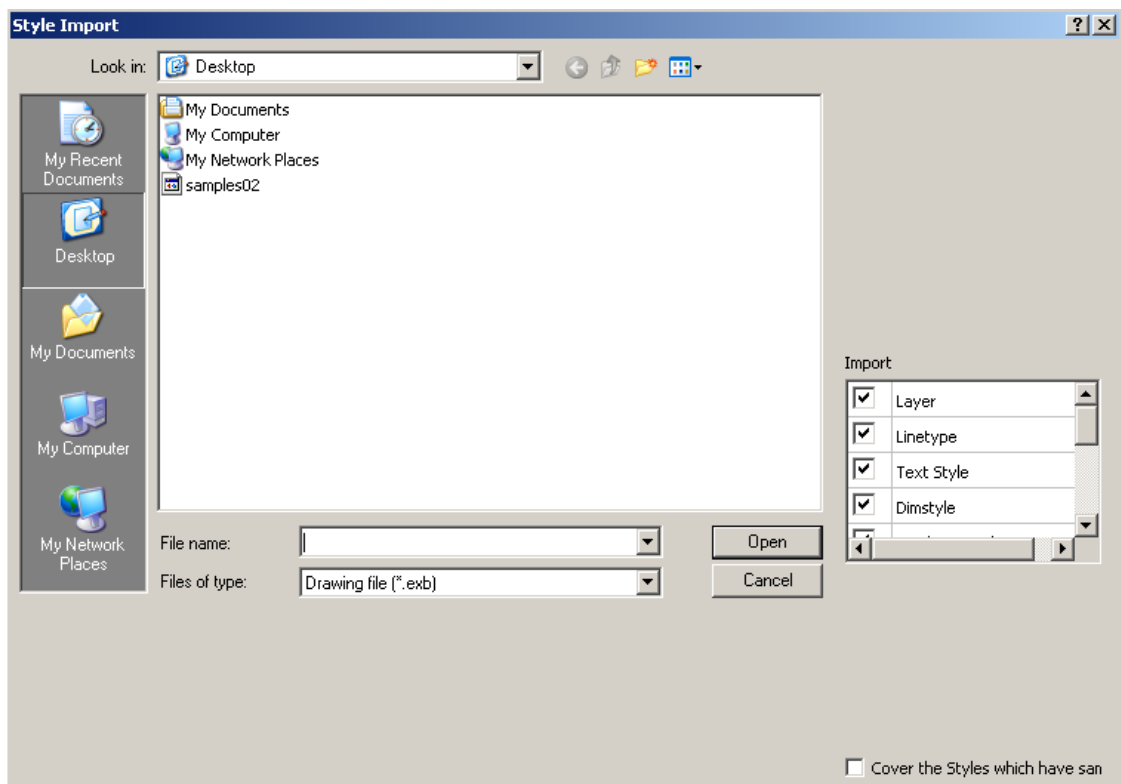
User can execute option New, Delete, set current in the above dialog box.

Click symbol + on the left of dimension style to select Standard, or double click Standard directly, the dimension style interface will pop up, in which user can modify directly. The operation method is the same as that of section 6.6.2.

8.6.2 Style management

Execute command Type, user can operate merge, filtration and export in dialog box 8-34.

(1) **Import:** the saved template or style can be imported to current drawing by this command. Click button **Import**, the window of Import style will pop up, as shown in figure 8-36.



Pic8-36 style import

Click option File type , select graph file or template file, then select drawing or template, from which the style will be imported.

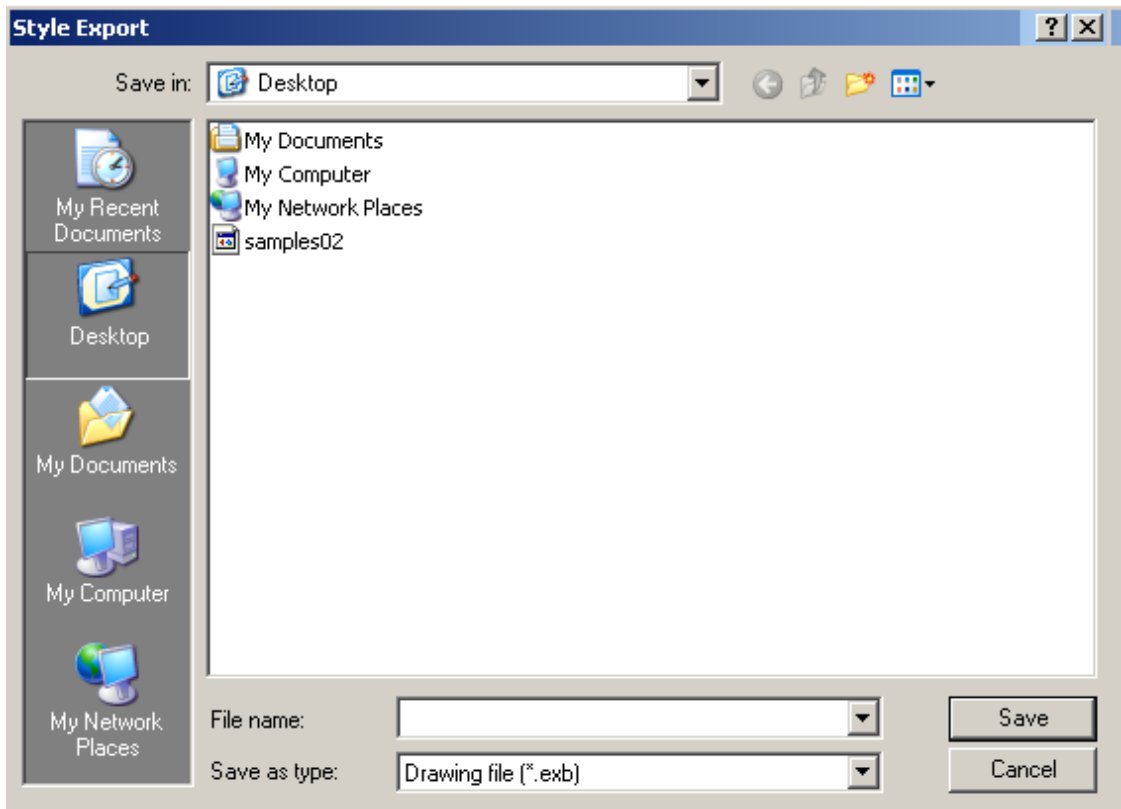
Click check box of each style to determine style type to be imported, and determine whether covering style with the same name after import. Then click button **open** to finish style import.

(2) **Export:** Export the current style as template file or drawing .

Saved as graph file, a blank file with current style will be saved for later use.

Saved as template file, it will be copied to corresponding support folder under installation catalog for future use.

Click button Export, following dialog box will pop up, as shown in figure 8-37.

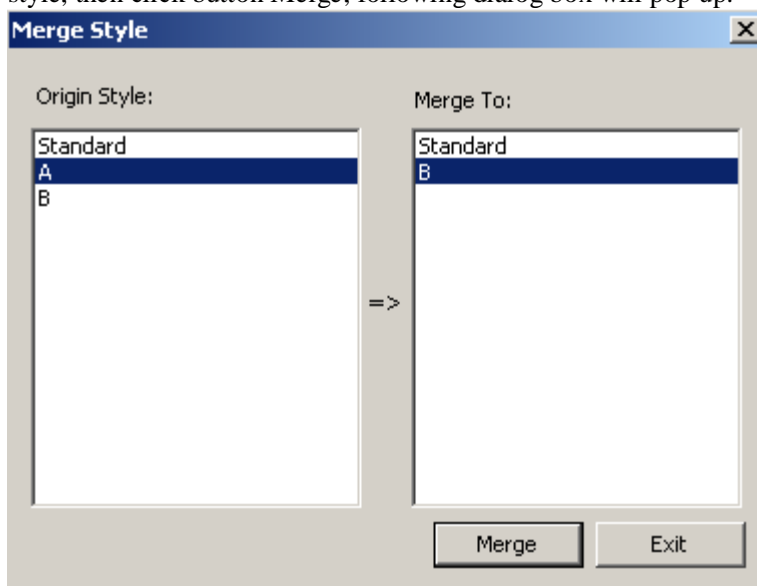


Pic8-37 style export

Select save type, then input file name and specify save path, at last , click Save.

(3) Merge: change the objected style to another one.

Select one style, then click button Merge, following dialog box will pop up.

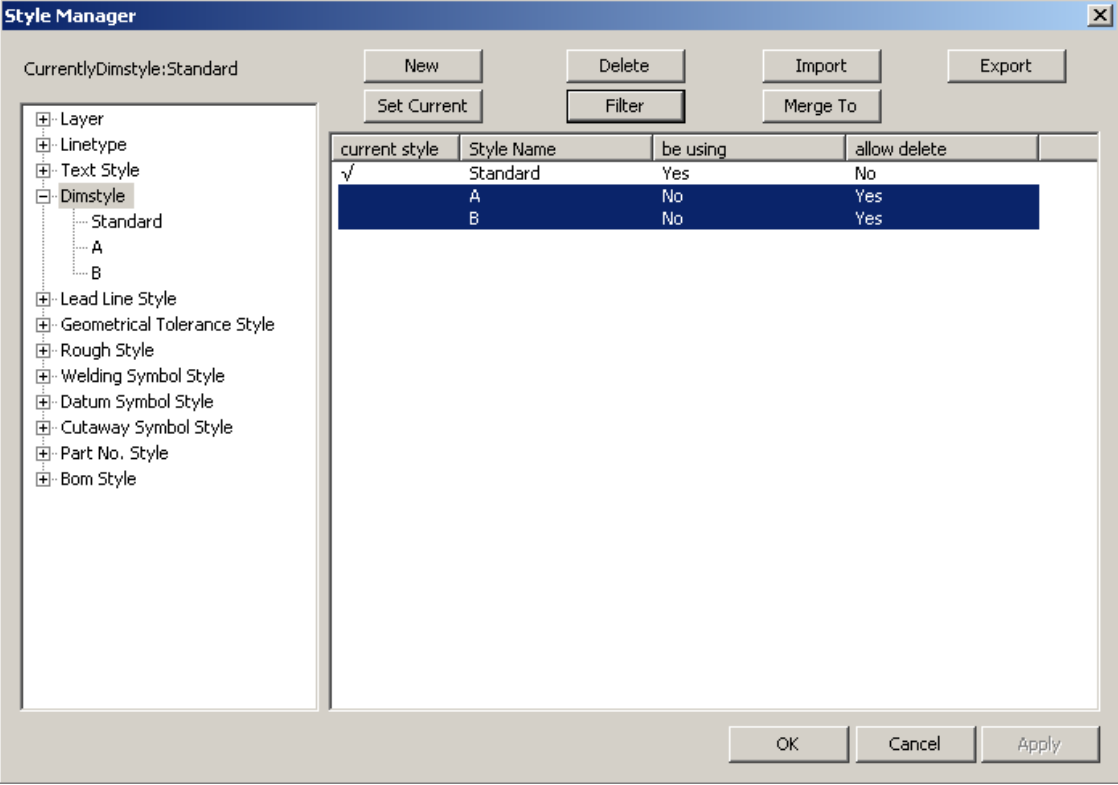


Pic8-38 style merge

In the above dialog box, Select **A** in original style box, then click Merge to **B**, at last click button **merge** to confirm style merge and operate. Then A style will be changed to B style.

(4) **Filtration:** Filtrate untapped style.

For example, click **dimension style** on the left, then click button **filtration**, as shown in following figure.



Pic8-39 Style filter

The untapped dimension style A and B will be filtrated. Click button **delete** to delete the useless style.

8.7 User's coordinates system

Coordinate system in CAXA Draft includes World Coordinate system and user coordinate system, in world coordinate system, X axis is horizontal, Y axis is vertical. The origin is intersection point (0,0) of X axis and Y axis. User can create user coordinate system to input

coordinate, display grid, capture ,or edit object conveniently.

The UCS operation includes New, Manage and Switch. Following is detail introduction.


8.7.1 Create UCS

【Command】 newucs

【Icon】 

【Definition】 Create a user coordinate system.

【Step】

User can execute command newucs in forms of :click button New in the sub menu of NEWUCS from the main menu of Format, click button  in the UCS panel of View option card, use Newucs command.

Click "Newucs", it will hint "Specify UCS origin".

If new origin is inputted via keyboard, all inputted coordinates for new origin are the value in the original coordinates. Then it will hint "Input rotate angel <-360,360>".

New UCS setting is finished When the rotate angel is inputted, and it will be current coordinates.


8.7.2 Manage UCS

【Command】 switch

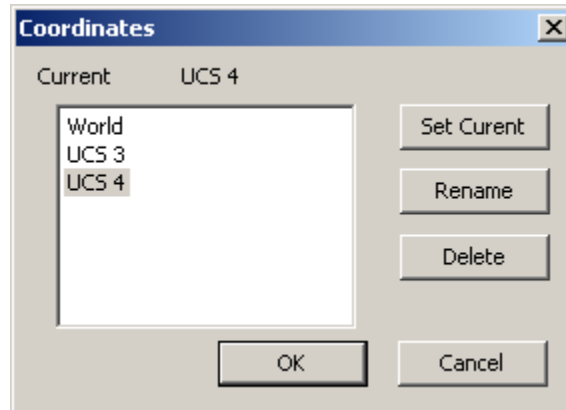
【Icon】 

【Definition】 Manage all current UCS.

【Step】

User can execute command switch in forms of :click button Switch in the sub menu of NEWUCS from the main menu of Format, click button  in the UCS panel of View option card, use Switch command.

Execute command Switch, following dialog box will pop up, as shown in figure 8-41.



Pic8-41 Manage UCS

The meaning and method for Switch operation:

- (1) Set current: Select one coordinate system, click button Set current , then that coordinate system will be the current one, and the current coordinate will be displayed in bright purple, all other coordinates will be displayed in red.
- (2) Rename: Select one coordinate system, click button Rename, then input a new name to confirm.
- (3) Delete: Select one coordinate system, click button Delete to delete coordinate system directly.

8.7.3 Switch coordinate system

【Definition】 Switch current coordinate system, user can switch between World Coordinate system and User Coordinate system.

【Step】

User can execute Switch coordinate system in forms of: execute Manage UCS, then use function of Set current, Click coordinate system display list from UCS panel of View option card, Press F5 key to switch between different coordinate systems.

Specify current coordinate system by command of Switch coordinate system, by default, current coordinates system color is purple. UCS color can be set in the color setting section of system setting dialog box.

8.8 System configuration


【Command】 syscfg

【Icon】 

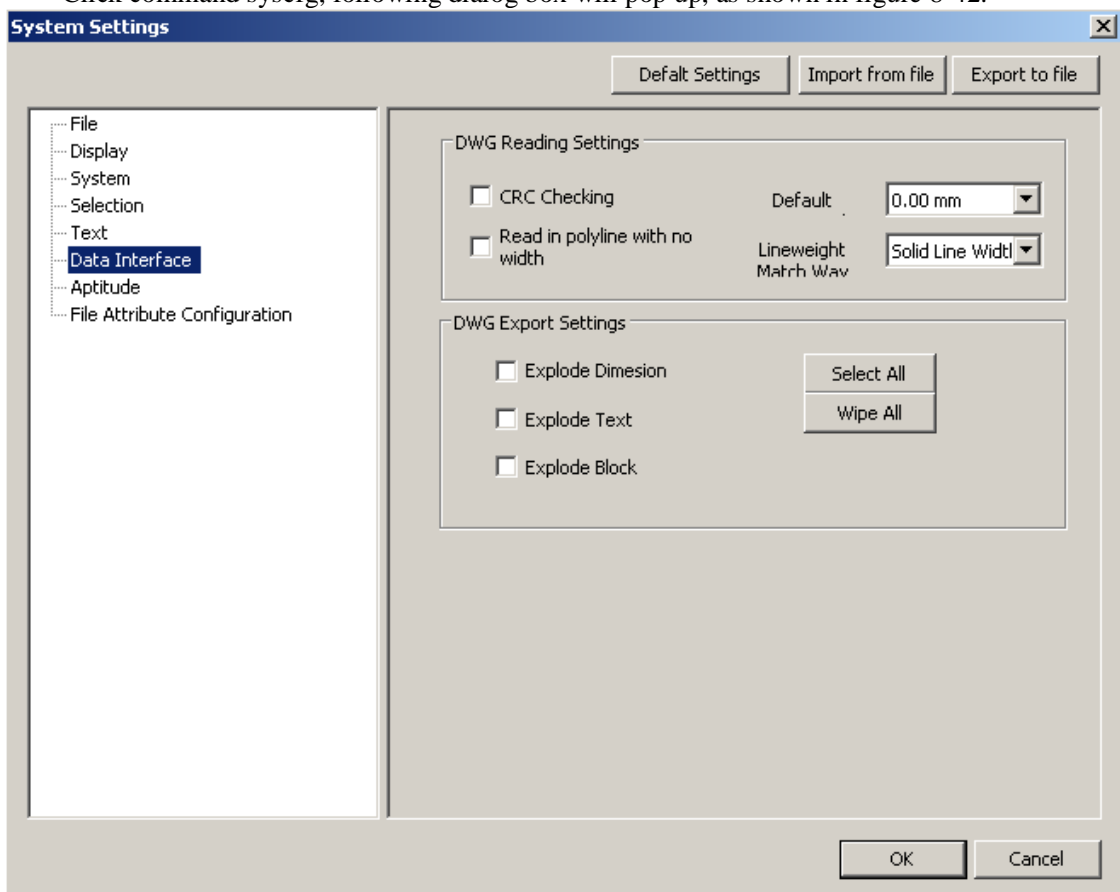
【Definition】 Set commonly used parametric.

Commonly used system parametric includes: DWG interface setting, system parametric setting, text setting, file path setting, display setting, selection tool setting, smart point tool setting, file attribute setting etc. the set parametric in the system can be saved or add-on.

【Step】

User can execute command syscfg in forms of :click button Option in the main menu of Tool, click button Option in the menu button. Click button  in the Option panel of Tool option card. use syscfg command

Click command syscfg, following dialog box will pop up, as shown in figure 8-42.



Pic8-42 System configuration

Method for using system configuration dialog box.

- On the left side of dialog box, it is parametric list, click to select the parametric, then set it on right side of the dialog box.

- Click Default to undo modified parametric.
- Click “Import From File” to add on the already saved parametric configuration file, load saved parametric setting.
- Click “Export to file” to saved current system parametric in one parametric file.

Following is detail introduction of commonly used parametric setting, among which, smart point tool setting has been introduced in section 9.5.1.

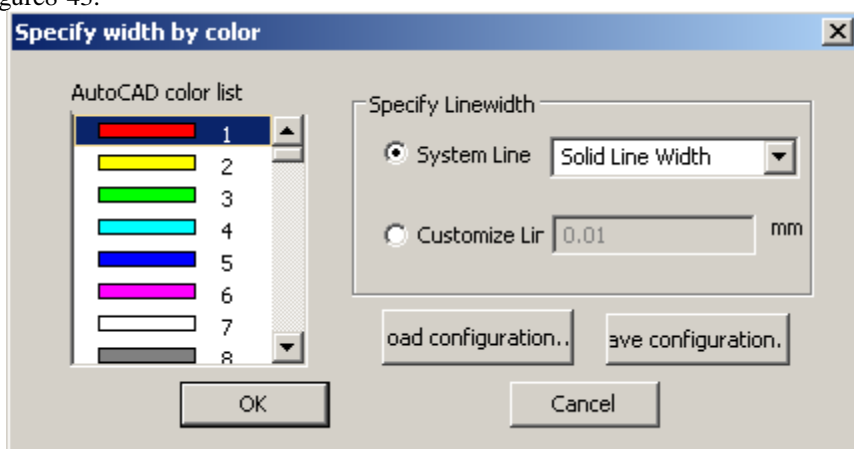
8.8.1 DWG interface setting

【Definition】: Set parametric of DWG file when import and export.

Click command Syscfg, on the left side of pop up dialog box, select DWG interface setting, as shown in figure 8-42, each parametric meaning is explained as follows:

- (1) CRC check: User can select whether proceed CRC check when loads DWG file . Select this option, when open wrong DWG file, it will prompt “Error”, and stop file readin. Cancel this option, it will omit such error and continue to readin DWG file.
- (2) Default line weight: Readin as per default line weight in DWG file.
- (3) Non-width readin polyline: Select this option, all DWG polyline will be readin as per 0 line weight. Cancel this option, it will readin as per polyline default line weight in DWG file.
- (4) Lineweight match mode: Read in DWG file as per original object line weight and color matching line weight. By default, it will read in as per line weight.

Select color in the pull down list of Lineweight match mode, following dialog box will pop up, as shown in figure8-43.



Pic8-43 Specify width by color

In the above dialog box, user can specify lineweight as per linetype color in AutoCad. Lineweights

in the pulldown list of system lineweight can be selected, also the lineweight can be defined in option of user-defined lineweight.

The set parametric can be saved, use can directly load in configuration file when printing next time.

Click button Save configuration or button add on configuration. User can readin or export parametric setting in the dialog box, as shown in figure 8-43.

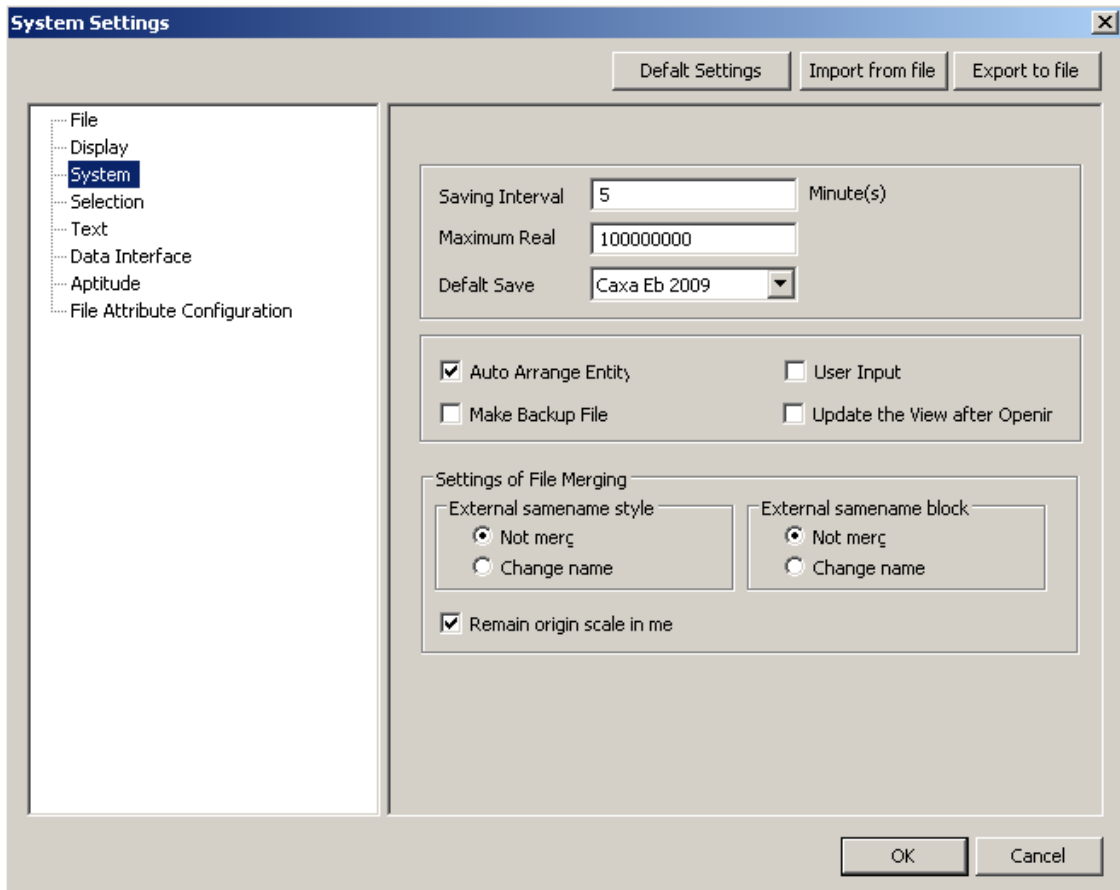
(5) DWG export setting: whether break object or not when export DWG , object which can be broken includes Dimension, text and block.

When saving CAXA Draft file as DWG/DXF file, the default mode is to save text and dimension as block. User can break corresponding part by selecting check box of dimension, text and block in the dialog box of DWG output setting.

8.8.2 System Parametric setting

【Definition】: Set system commonly used parametric.

Execute command Syscfg, on the left side of pop up dialog box, select system parametric setting, as shown in figure 8-44.

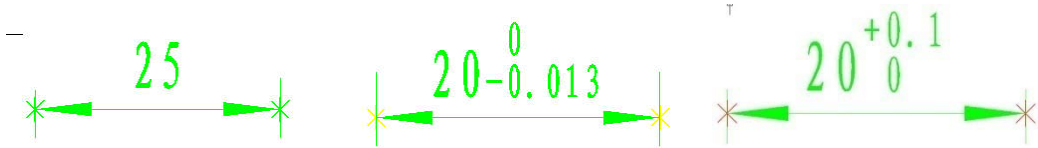


Pic8-44 System settings

Each parametric meaning and using method is shown below:

- (1) Save after :the unit is add/erase, when record of adding /erasing reaches the set value, the current drawing will be saved in tmp0000.exb of temp directory in system automatically , avoiding losing information on all drawing at abnormal exit. Effective range is 0~900000000.
- (2) Maximum real number: It means the maximum real number to be inputted in immediate menu.
- (3) Default save format: user can set default save format when saving file in CAXA Draft.
- (4) Auto arrange layer: center line, section line, measure dimension can be set in due layer automatically.
- (5) Make backup: .bak file is produced automatically after each amendment.
- (6) Update file when opening view: If this function is picked, each view will be updated along with changed 3D file automatically when view file is opened.

- (7) Inputted dimension display: If the dimension is inputted by user, instead of actually measured one, this option will be marked, the mark method is explained as follows:



a)only dimensions

b)only tolerance

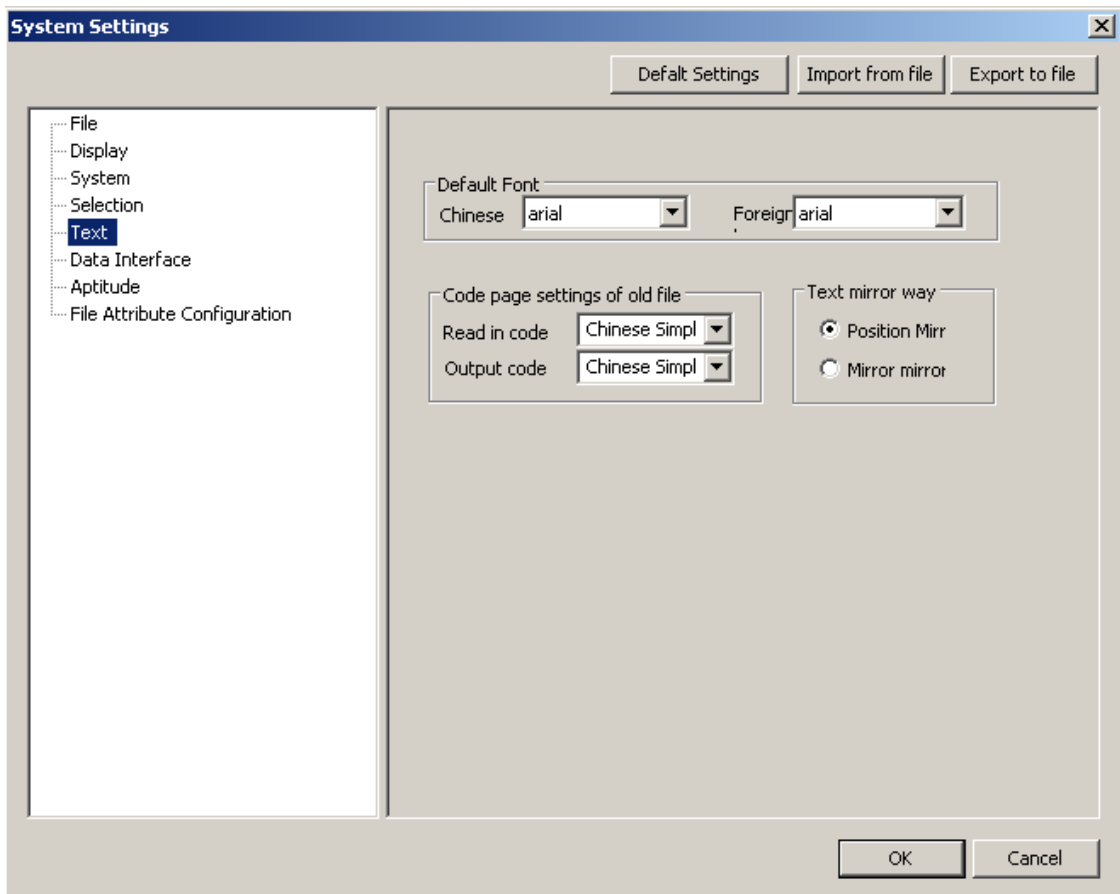
c)dimensions and tolerance

- (8) File merge setting: When merging file or paste object to current drawing, use can set whether merging style or block with the same name. whether keep original scale when being merged.

8.8.3 Text setting

【Definition】: Set system text parametric.

Execute command Syscfg, on the left side of pop up dialog box, select Text setting, as shown in figure 8-45.



Pic8-45 text settings

Click Pull down list of Chinese font or western font in Default font, specify default Chinese font or western font. Default font illustration: When font in opened file is not the installed one, the system will use default font.

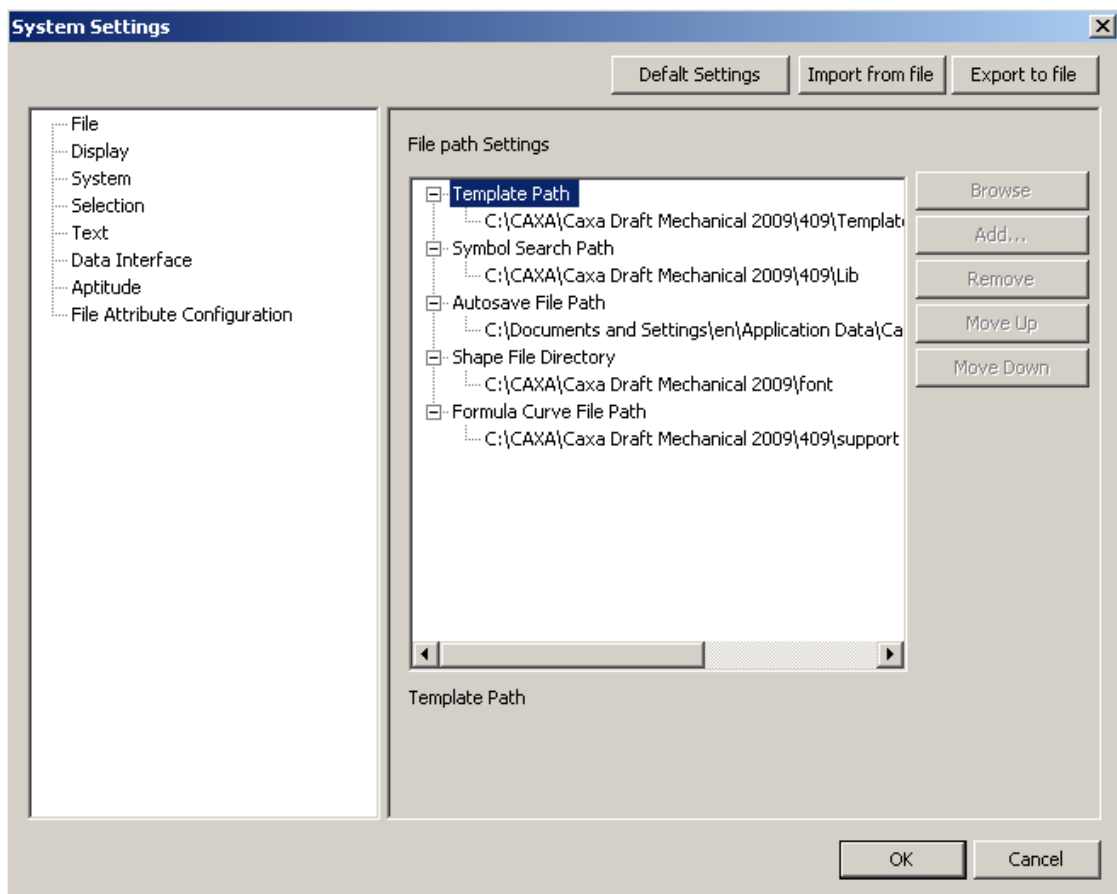
Click “Old file code page”,user can specify open or export code page of old file.

Click “Text mirror” to select Place mirror or Mirror for the text.

8.8.4 File path setting

【Definition】: Set file path in the system.

Execute command Syscfg, on the left side of pop up dialog box, select File path setting, as shown in figure 8-46.



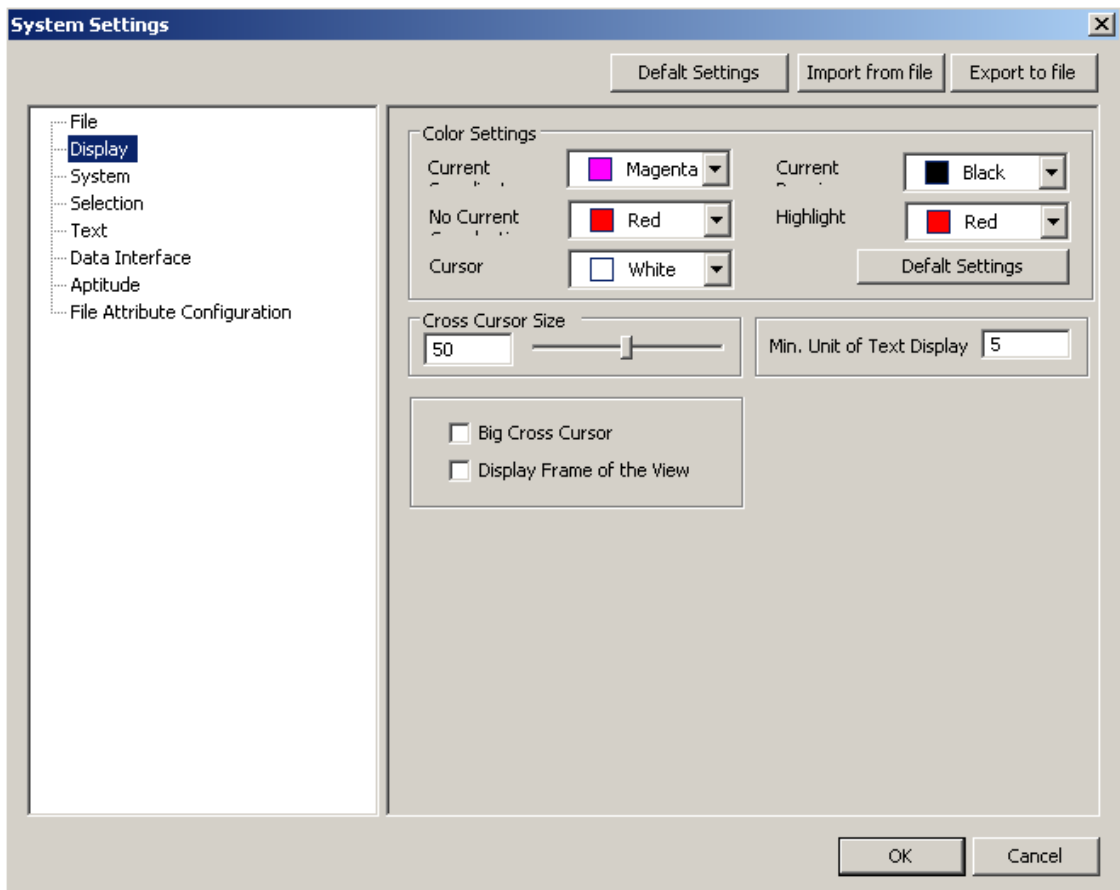
Pic8-46 File path settings

In the above dialog box, user can set file path, such as template path, library searching path, Auto save file path, SHX file path, formula curve path. Select one path, then user can browse, add, delete, move upward, move downward etc.

8.8.5 Display setting

【Definition】: Set system display parametric

Execute command Syscfg, on the left side of pop up dialog box, select Display setting, as shown in figure 8-47.



Pic8-47 display settings

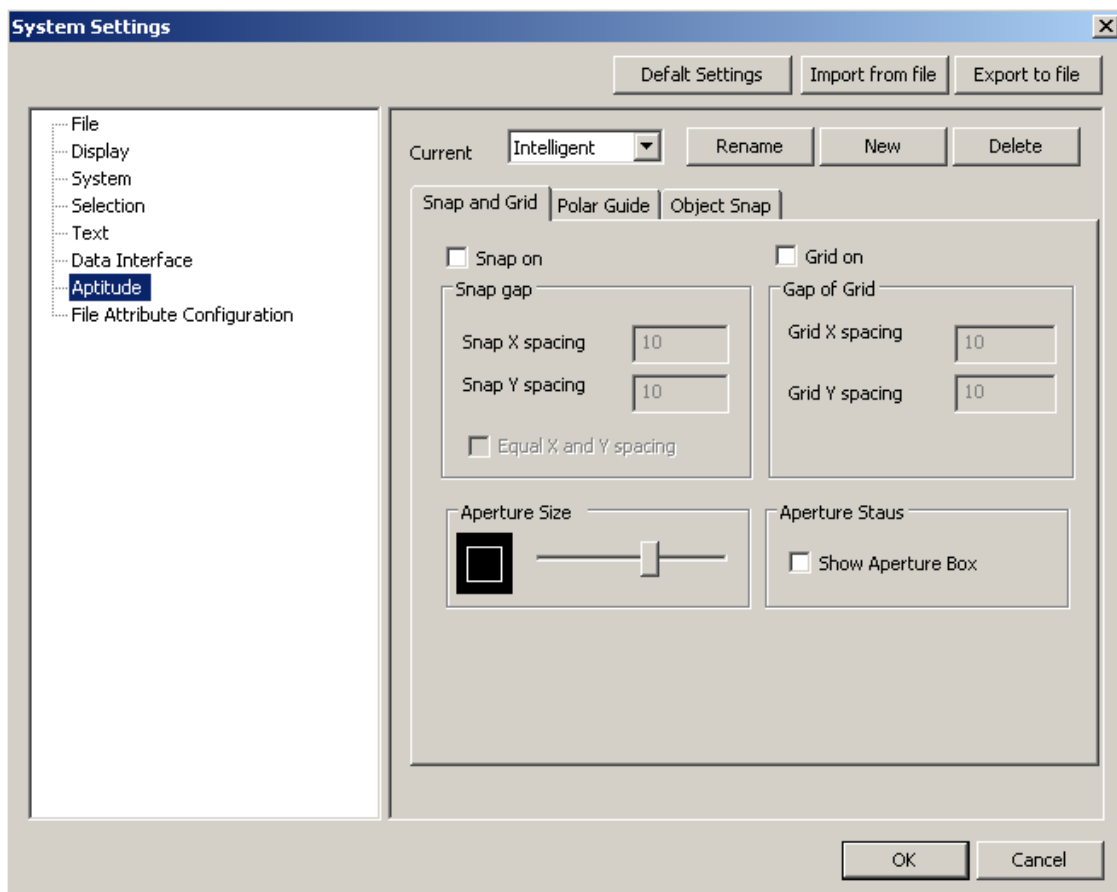
Each parametric meaning and operation is shown below:

- (1) Color setting: in this dialog box, it will display current coordinate system, non-current coordinate system, current drawing area, highlighted selection, cursor color. Click each parametric list to modify each color setting. Click "default", the system will return to the default color setting.
- (2) Cross cursor size: Input or drag handle to specify cross cursor size.
- (3) Text display minimum unit: Specify value for text minimum display unit.
- (4) Big cross cursor: Select this option to set cursor as big cross cursor.
- (5) Display view frame: Select this option to set Display 3D view frame.

8.8.6 Select tool setting

【Definition】: Set system select tool parametric.

Execute command Syscfg, on the left side of pop up dialog box, Click “Select tool setting”, as shown in figure 8-48.

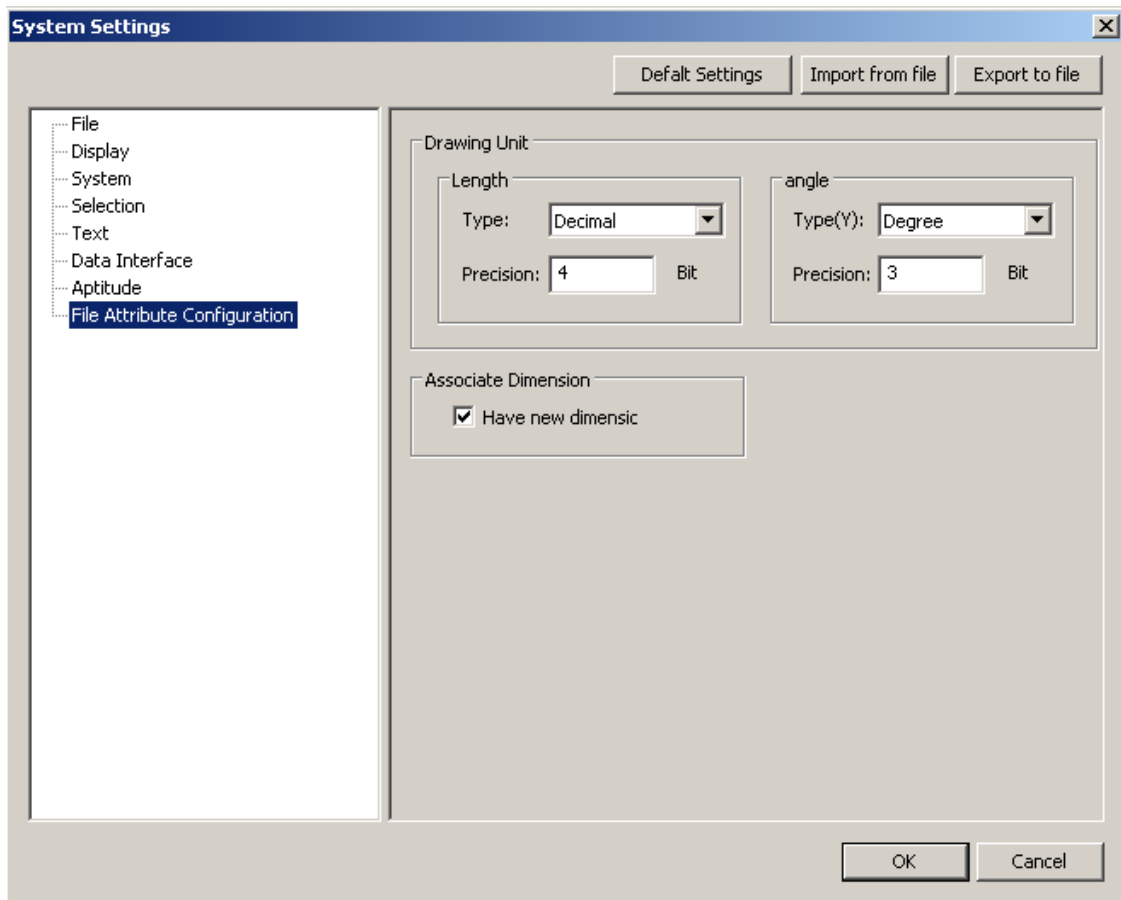


Pic8-48 aptitude settings

8.8.7 File attribute setting

【Definition】: Set system file attribute parametric.

Execute command Syscfg, on the left side of pop up dialog box, Click “File attribute setting”, as shown in figure 8-49.



Pic8-49 File attribute settings

In the above dialog box, user can set graph unit, such as length type and accuracy, angle type and accuracy.

8.9 Interface configuration

Interface configuration includes interface switch, reset, save, add-on etc, as well as interface element customize. Following is detail introduction.

8.9.1 Interface operation

【Interface operation】includes: interface switch, interface reset, save interface configuration, add-on interface configuration.

8.9.1.1 Interface switch


【Command】 interface

【Icon】 

【Definition】 Switch between Typical interface and Fluent Interface.

CAXA Draft software includes Typical interface and Fluent Interface, use command interface to switch between them.

【Step】

User can execute command interface in forms of :click button interface in the main menu of Tool, click button  in the interface operation panel of View option card, use interface command, Press F9 key.

Execute command interface to activate.


8.9.1.2 Interface reset

【Command】 interfacereset

【Icon】 

【Definition】 resume system interface to default status.

【Step】

User can execute command interfacereset in forms of :click button interfacereset in the main menu of Tool, click button  in the interface operation panel of View option card, use interfacereset command.

Execute command interfacereset to activate.


8.9.1.3 Save interface configuration

【Command】 Interfacesave

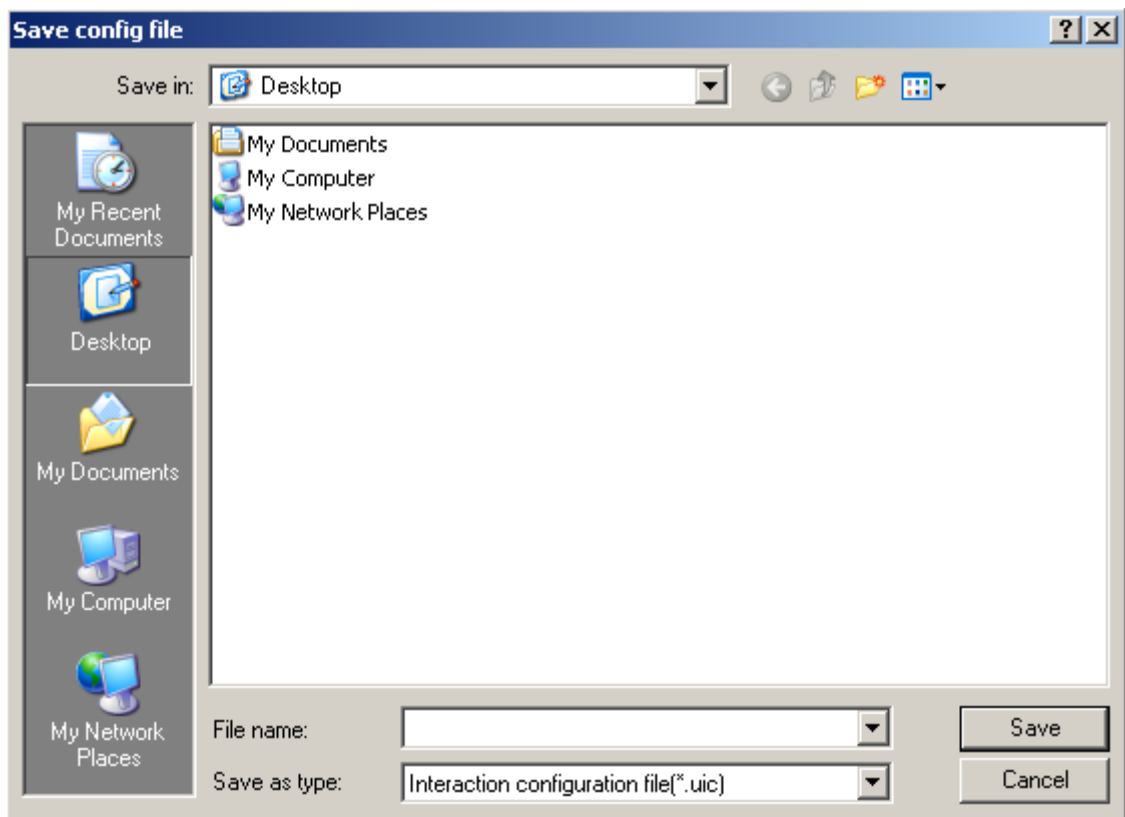
【Icon】 

【Definition】 Save current interface status to interface configuration file.

【Step】

User can execute command Interfacesave in forms of :click button Interfacesave in the main menu of Tool, click button  in the interface operation panel of View option card, use Interfacesave command.

Execute command Interfacesave , following dialog box will pop up.



Pic8-50 Save interface config file

Specify save path and file name, then click button Save.

8.9.1.4 Add on interface configuration

【Command】 Interfaceload

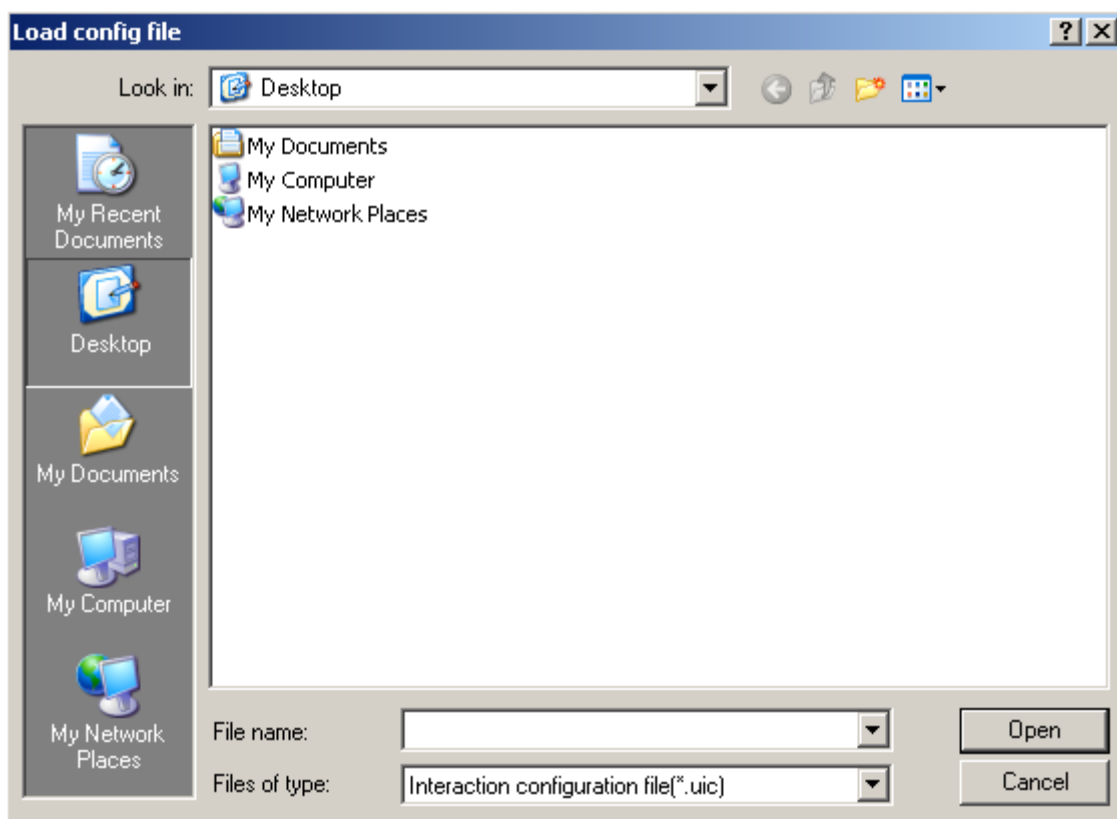
【Icon】

【Definition】 Add on already saved interface configuration, and resume system interface status.

【Step】

User can execute command Interfaceload in forms of :click button Interfaceload in the main menu of Tool, click button in the interface operation panel of View option card, use Interfaceload command.

Execute command Interfaceload , following dialog box will pop up.



Pic8-51 load interface config file

Select one interface configuration file, click Open.

8.9.2 Interface customize

【Command】 customize

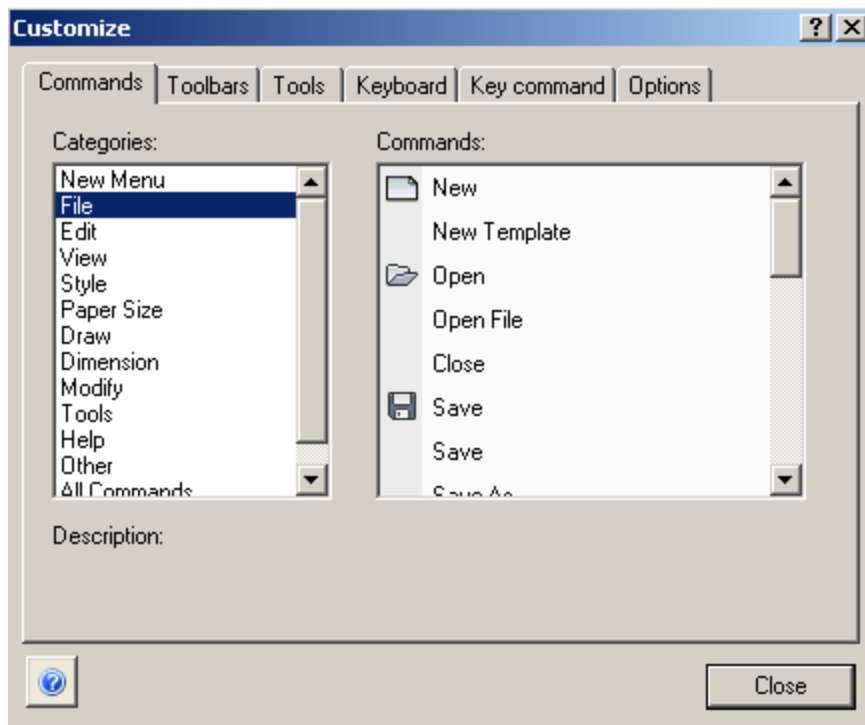
【Definition】 Customize interface element and interface status.

Element to be customized includes main menu, tool bar, outer tool , short-cut key, keyboard command etc.

【Step】

User can execute command customize in forms of : click right key on tool bar or function area, select customize in the pop up menu, use command Customize directly.

Execute command Customize, following dialog box will pop up.



Pic8-52 interface customize

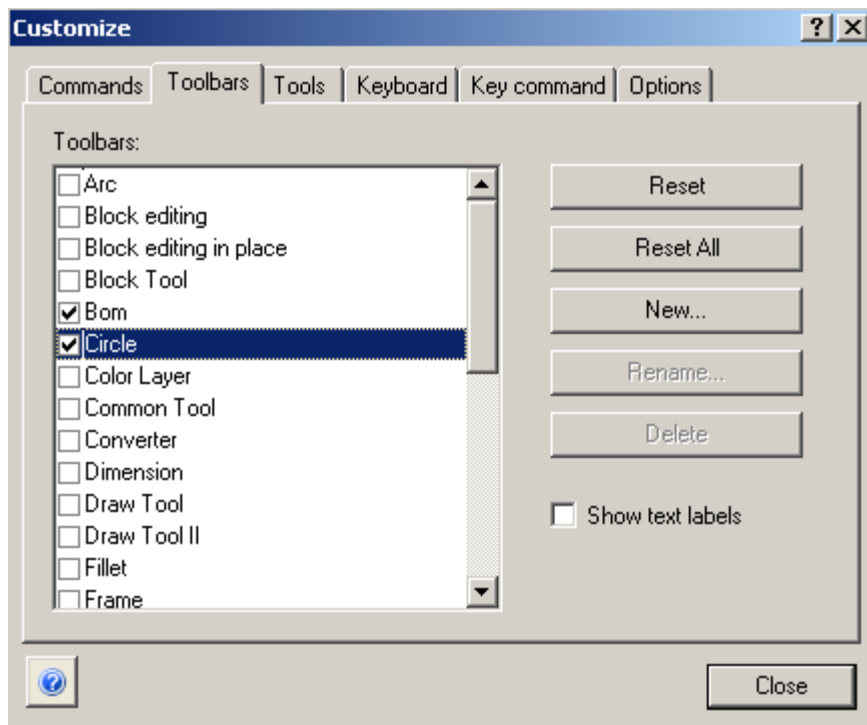
Then user can operate in the dialog box:

(1) Customize menu

Click command Customize, then select one main menu button, corresponding menu will popup, in dialog box of 8-52, select option Command, click left key to select option Command and drag it to the pop up main menu, then this command is added. User can also drag one command from main menu to interface customize dialog box, then this command is removed.

(2) Customize tool bar

Click command Customize, select option Tool bar, following dialog box will pop up.as shown in figure 8-53.



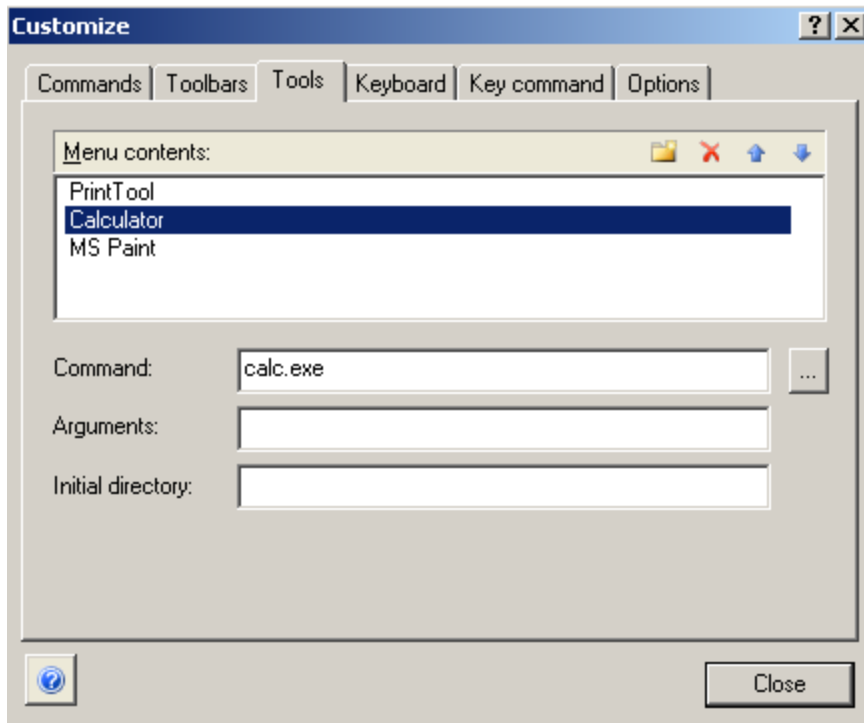
Pic8-53 toolbars customize

Operation method for customize tool bar:

- On the left side of dialog box, toolbar list is displayed, click check box to open or close tool bar.
- Click button New to create one new tool bar.
- Click button Command, drag this command to tool bar by left key, or drag command from tool bar to dialog box.






(3) Customize outer tool: integrate some commonly used tool to CAXA Draft.

Click command Customize, and select option Tool, then following dialog box will pop up, as shown in figure 8-54.



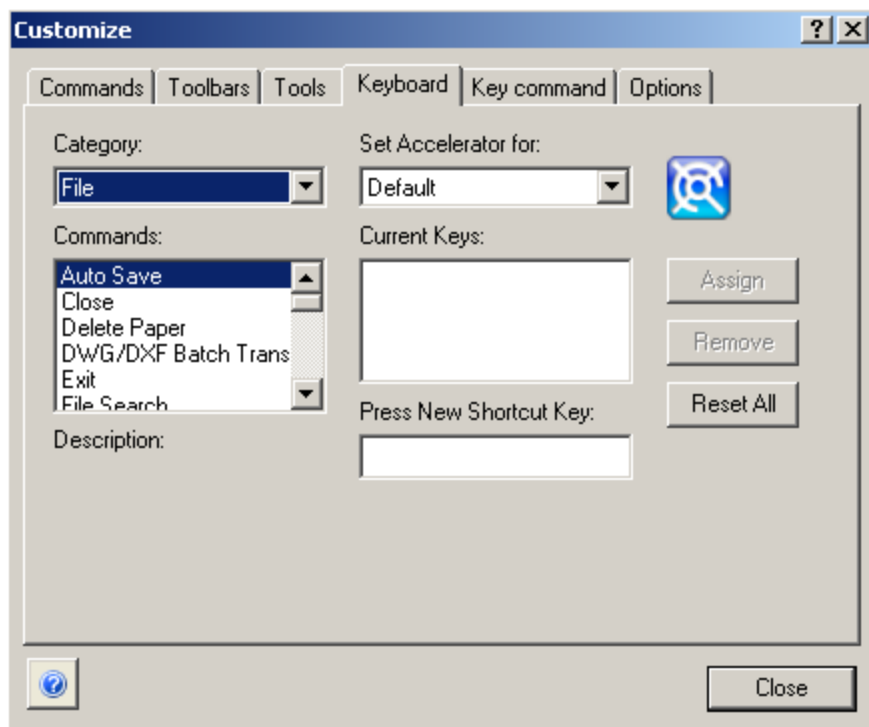
Pic8-54 tools customize

Operation method for customize outer tool:

- Click button     on the right of Menu catalog to Create outer tool, delete outer tool, move outer tool upward, move outer tool downward.
- Select one outer tool, click button  to specify its path.

(4) Customize short cut key

Click command Customize, and select option Tool , then following dialog box will pop up, as shown in figure 8-55.



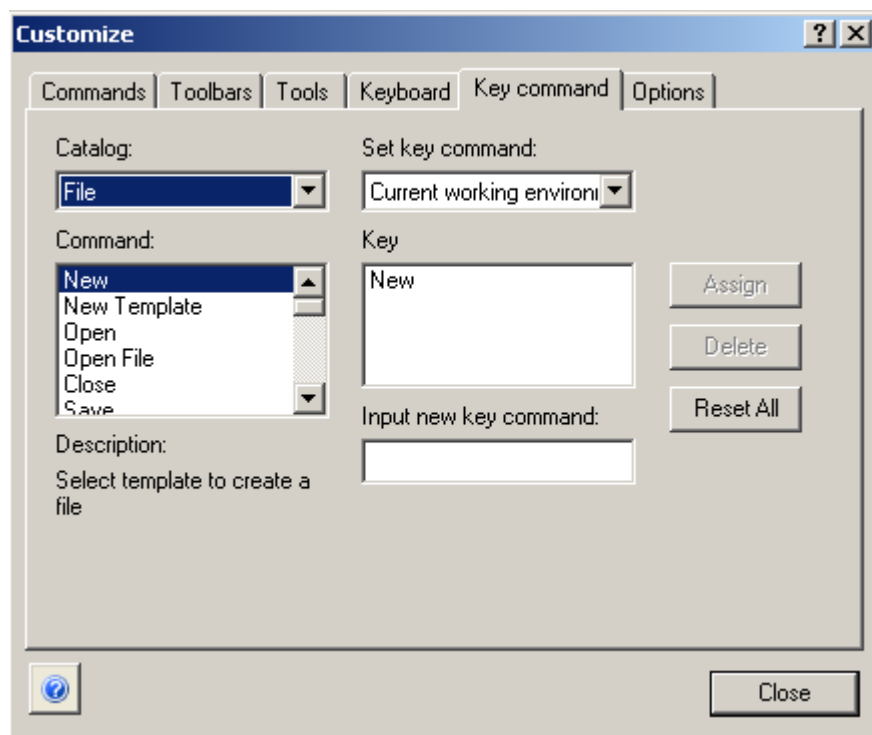
Pic8-55 keyboard customize

Operation method for customize short cut key.

- Click pull down menu of Type to select command type, each command will be shown under Command.
- Select one command, click input box under option “Press new short cut key”, then press keyboard combination keys, at last click button Specify.
- Select one specified short cut key, Click button Delete to delete.

(5) Customize keyboard command

Click command Customize, and select option keyboard command , then following dialog box will pop up, as shown in figure 8-56:



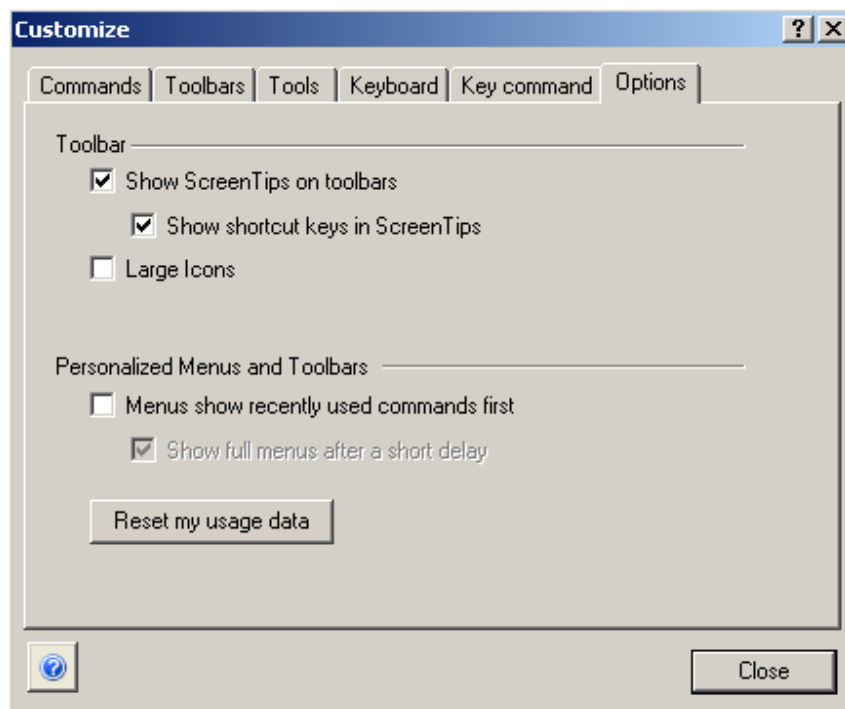
Pic8-56 customize key command

Operation method for customize keyboard command:

- Click pull down menu of Type to select command type, each command will be shown under Command.
- Select one command, click input box under option “Press new short cut key”, then press keyboard combination keys, at last click button Specify.
- Select one specified short cut key, Click button Delete to delete.

(6) Interface option

Click command Customize, and select Option, then following dialog box will pop up, as shown in figure 8-57:

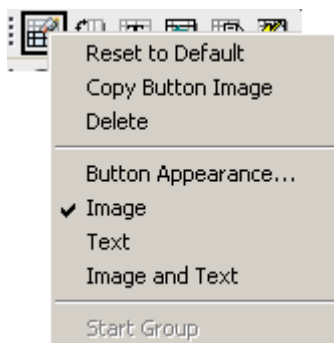


Pic8-57 customize options

Click check box in front of each option to set “Display prompt about tool bar”, Display short cut mode in screen prompt, large icon, display the latest used command in menu etc.

(7) Chang appearance of Icon button

Execute command Customize, click right key on the icon button, following menu will pop up.as shown in figure 8-58.



Pic8-58 Chang appearance of Icon button

In the above menu, set display mode for selected button as Icon, text, icon text etc, user can also delete button, copy button icon etc.

Chapter 9 Tool

As a universal CAD drawing software, CAXA Draft has perfect DWG data interface, it is compatible with all data of DWG in each version, meanwhile, it supports multiple handling modes of DWG file, such as open and save DWG file, transfer DWG file by lot, transfer drawing tool to handle DWG file etc.

In CAXA Draft, multiple aided tool is supplied, such as EXB file browser, system query, file retrieve, component library, template manager etc, by which user can browse, retrieve, query, modify drawing conveniently. What's more, second develop module can be added on. Following is the detail.

9.1 DWG interface

The content of DWG data interface includes: open and save DWG file, transfer DWG file by lot, transfer drawing tool to handle DWG file.

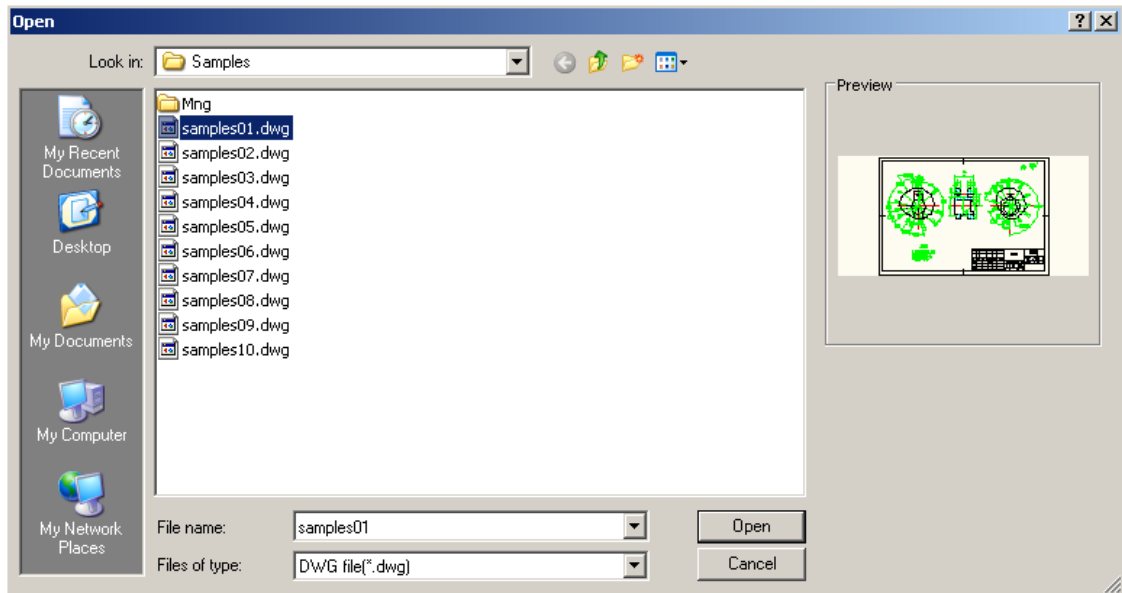
Refer to section 9.8.1 for DWG interface setting. The operation method for DWG is similar to that of DXF file.

9.1.1 Open and save DWG file

DWG file can be open and saved directly by command Open and Save in CAXA Draft.

9.1.1.1 Open DWG file

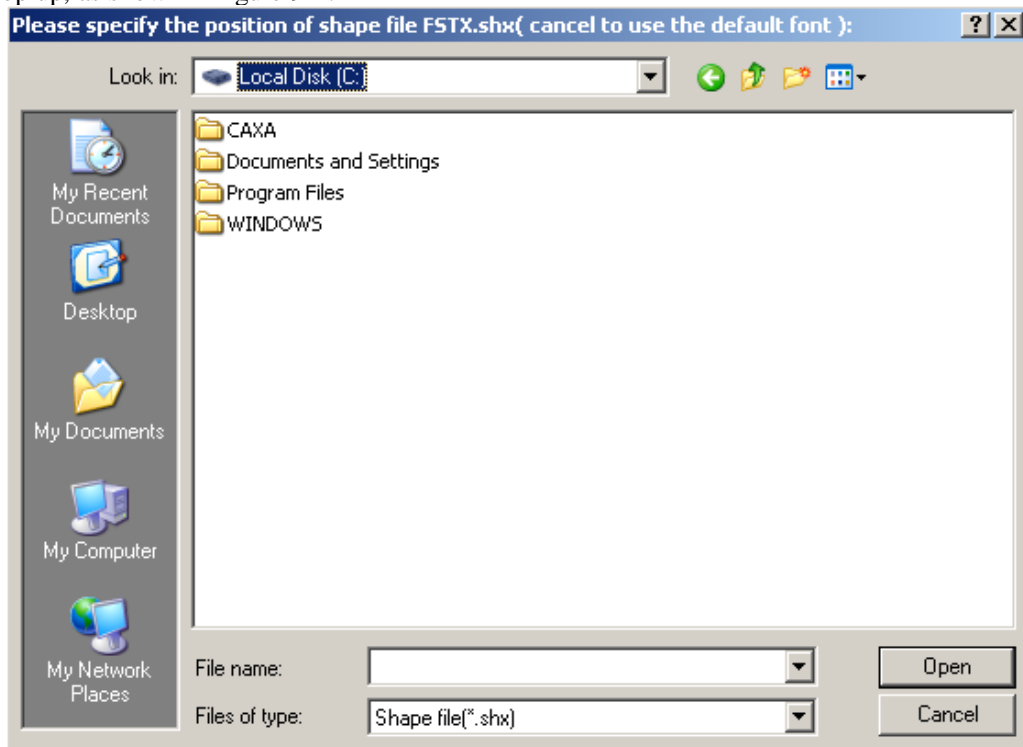
Click command open, following dialog box will pop up.



Pic9-1 Open DWG file

Select DWG file in the file type in the above dialog box, then select one DWG to open it.

When opening DWG file, if mongline font is used in the open file, following dialog box may pop up, as shown in figure 9-2.



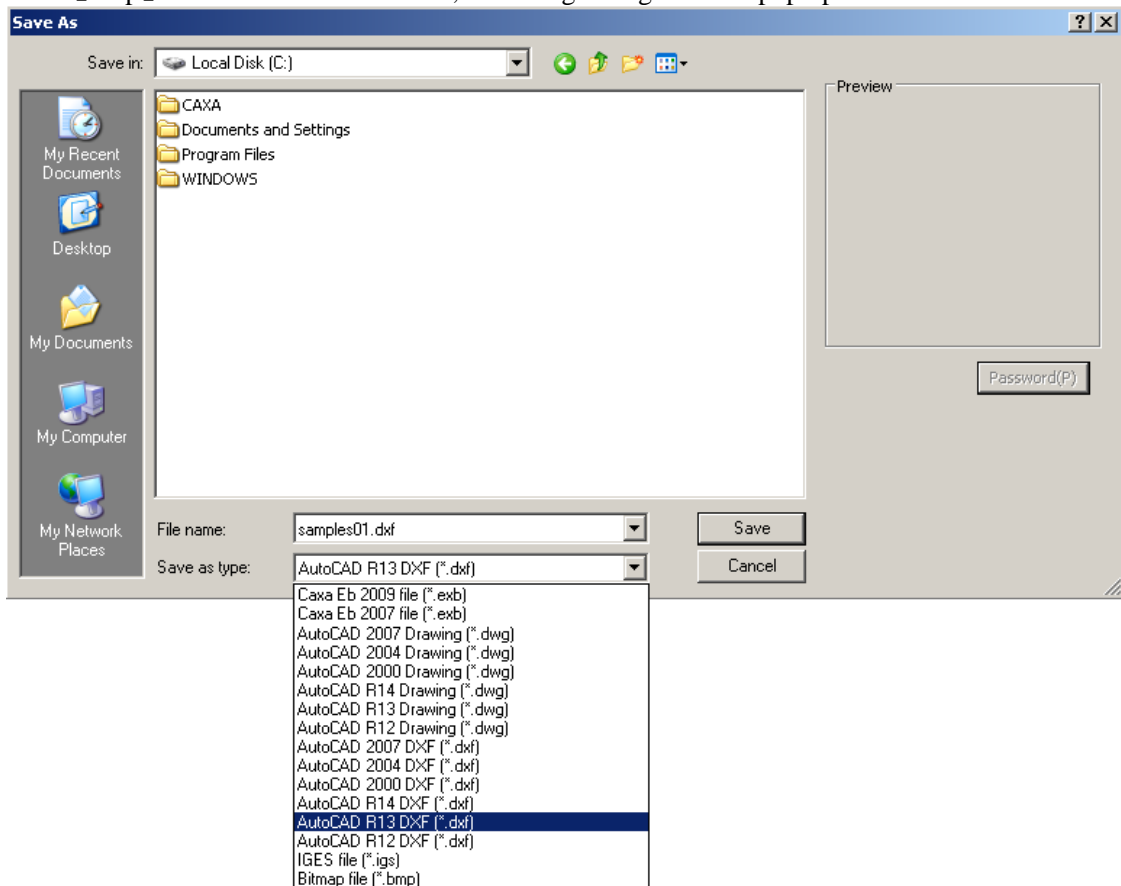
Pic9-2 Specify SHX font file

Then specify file that needs SHX font. The commonly used SHX font can be copied to the Font folder in installation catalog, then SHX font will be used in the newly opened DWG file.

9.1.1.2 Save DWG file

【Definition】: Export DWG/DXF file in different version.

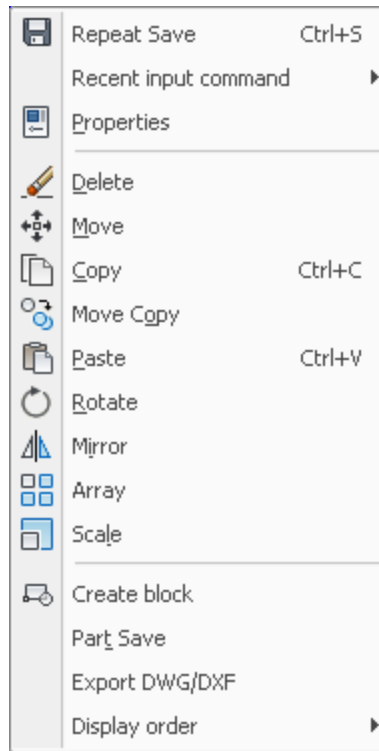
【Step】 Click command Save as, following dialog box will pop up.



Pic9-3 Save DWG file

In the save type, user can select DWG format with multiple version to save, input file name, click OK to confirm.

Or select object first, then click right key , following menu will pop up.



Pic9-4 Part save on right menu

Select part save, in the pop up dialog box, select DWG as save type.


9.1.2 Transfer DWG file by lot

【Command】 dwg

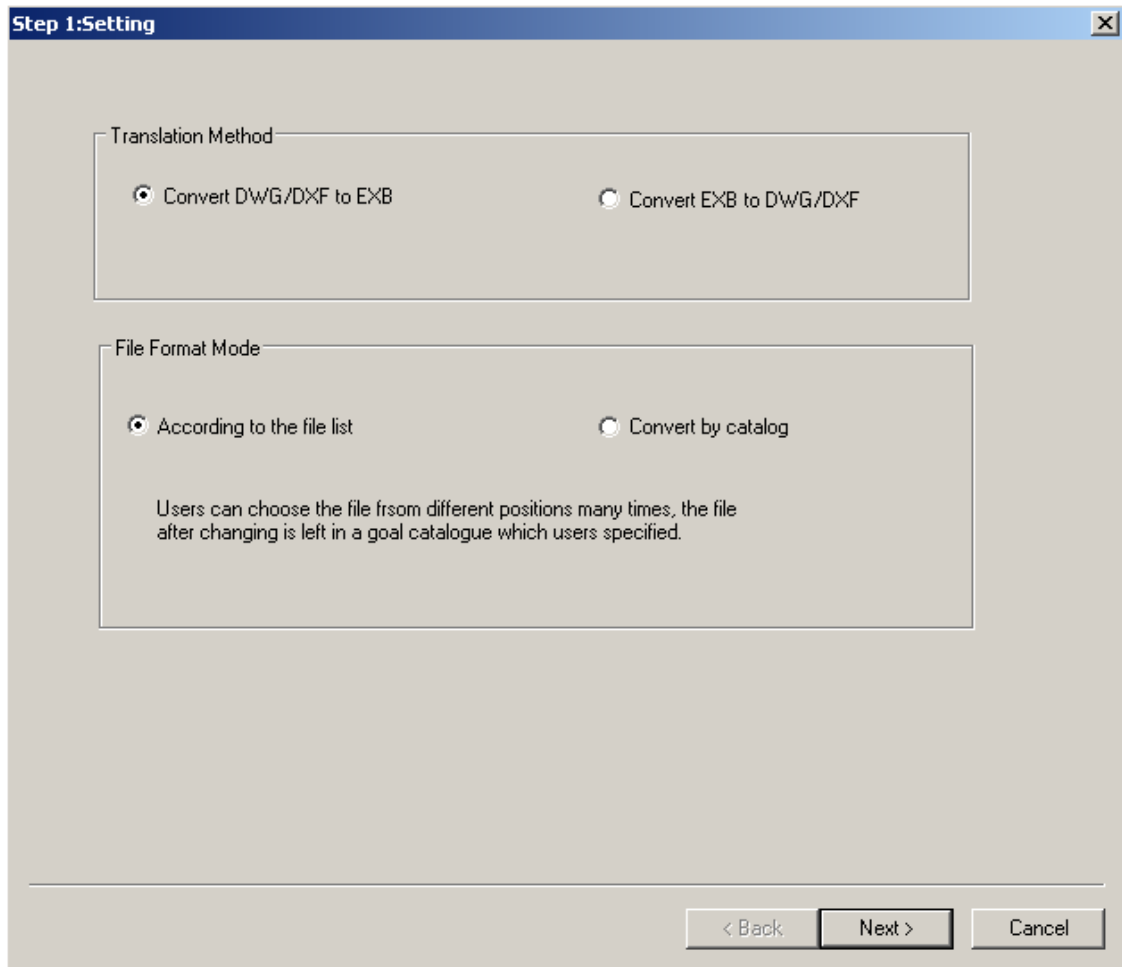
【Icon】 

【Definition】 DWG file in each version can be transferred to EXB file by lot, and EXB file with each version can be transferred to DWG file.

【Step】

User can execute command DWG in forms of: click button DWG in the main menu of file, click button  in the Tool panel of Tool option card, use command DWG directly.

Click command DWG, following dialog box will pop up.



Pic9-5 DWG convert tool

In the above dialog box, select Transfer mode and file structure mode, detail method is shown as follows:

- (1) Transfer mode: in the above dialog box, Select Transfer DWG/DXF file to EXB file , or Transfer EXB file to DWG/DXF file.

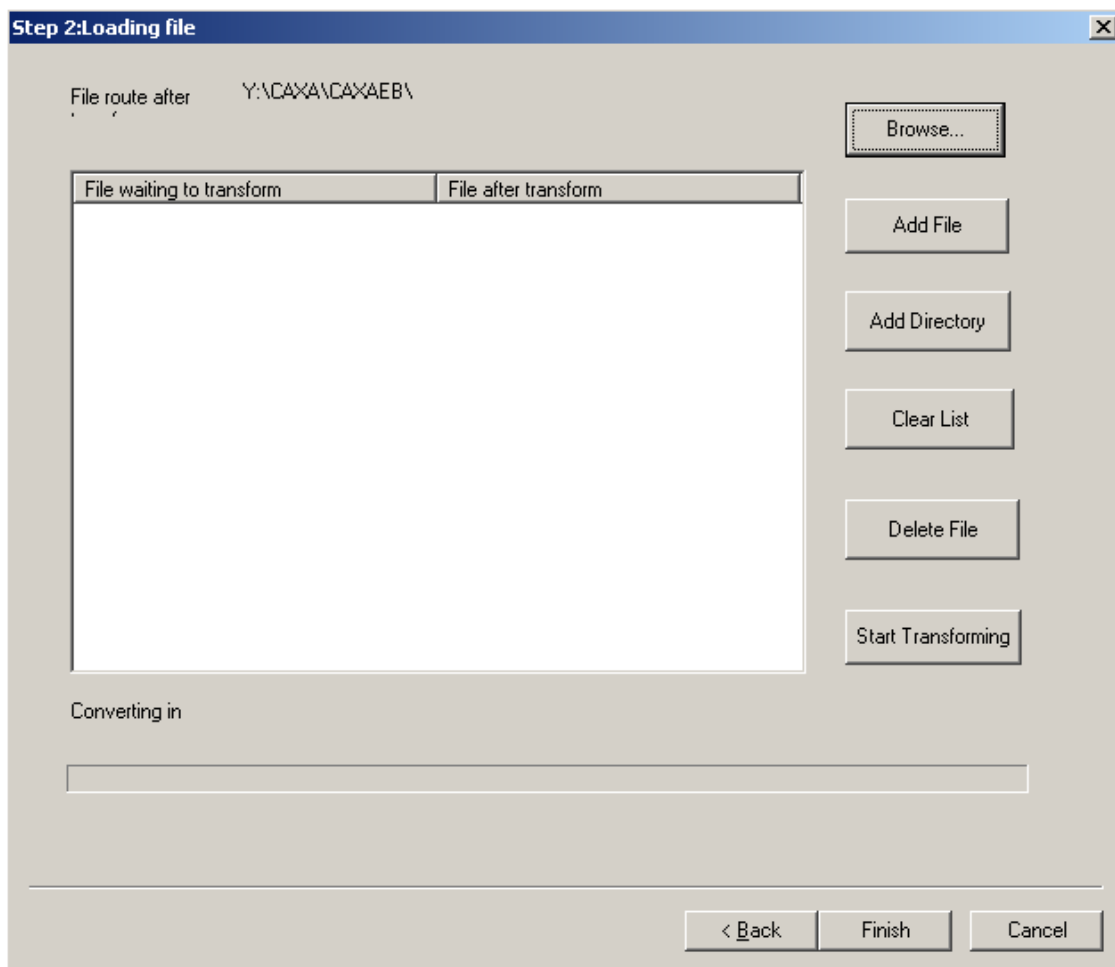
Once Transfer EXB file to DWG/DXF file is selected, click set to select DWG file version, as shown in figure 9-6.



Pic9-6 Select DWG/DXF file format

(2) **【File structure mode】** it is divided into two types ,they are transfer as per file list, transfer as per catalog structure.

● Transfer as per file list: Select file time after time at different position, the transferred file will be put in one specified target catalog, as shown in figure 9-7.



Pic9-7 File structure mode

Meaning and operation method for parametric in the above dialog box.

【File path after transferring】: it means save path when file is transferred.

【Add file】: add file to be transferred individually.

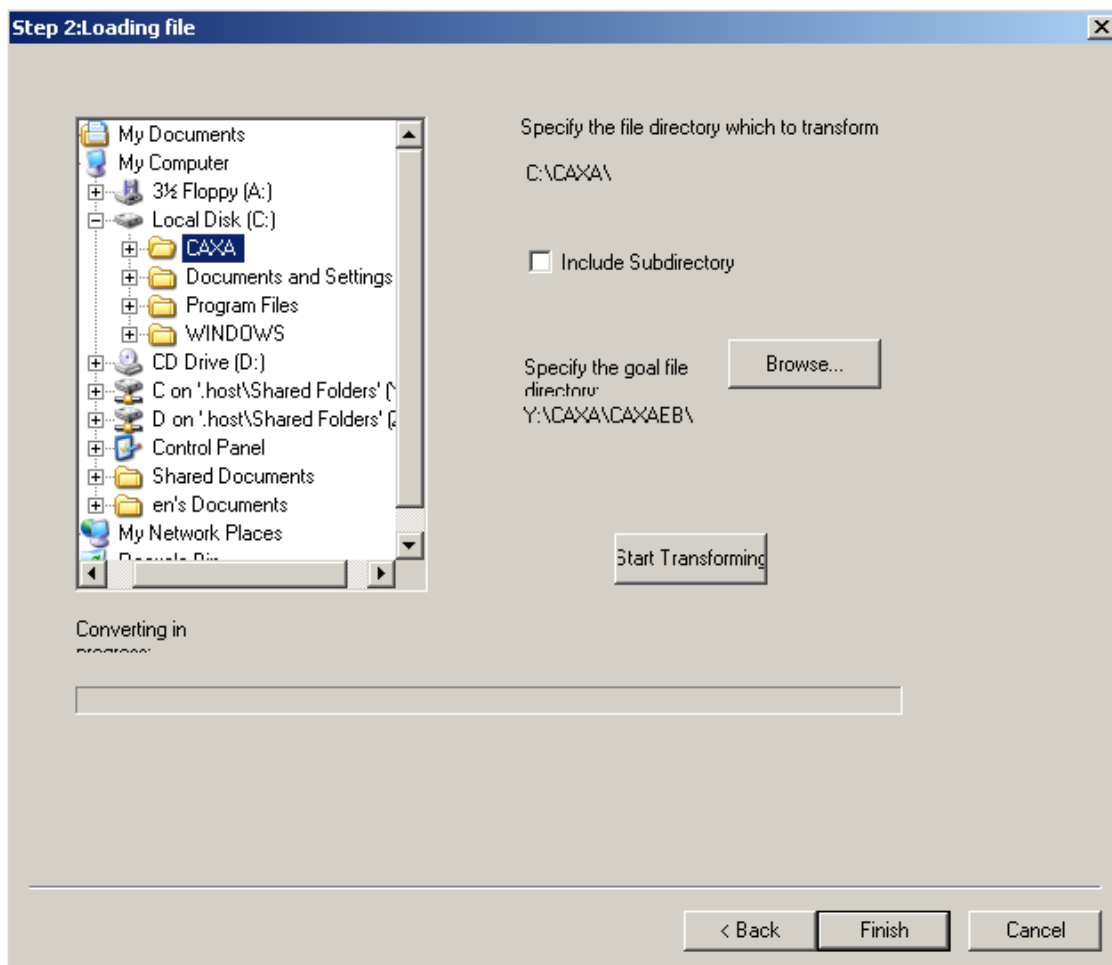
【Add catalog】: add file to be transferred, which meets request in selected catalog.

【Empty list】: empty file list.

【Delete file】: Delete selected file in the list.

【Start transferring】: Transfer file in the list, when it is transferred, it will query whether continue to operate. User can judge it as per request.

- **Transfer as per catalog structure:** Transfer data as per catalog form, all file in the catalog which meets requirement will be transferred by lot, as shown in figure 9-8.



Pic9-8 catalog structure mode

Meaning and operation method for parametric in the above dialog box.

【Transfer path】: Select catalog to be transferred on left of the dialog box.

【Including sub-catalog】: Select its check box, corresponding file in the sub catalog of selected catalog will be transferred together.

【File catalog after transfer】: Click button Browse to set save path for file after being transferred.

【Start transfer】: When all parametric is set, click this button to start to transfer.



9.2 EXB browser

【Icon】 

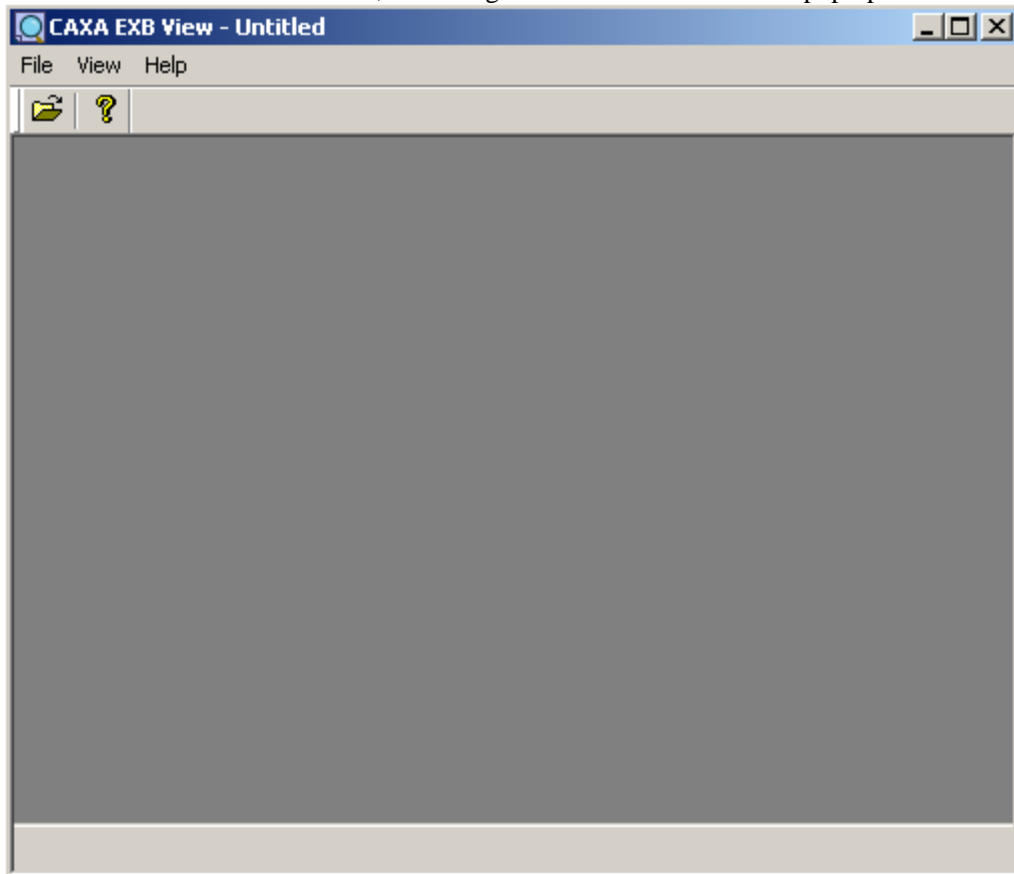
【Definition】 It is a tool for browsing EXB file.

EXB file browser is mainly used for viewing files with extension name exb. In browser, graphics can be zoomed in, zoomed out, window displayed and so on display operation.

【Step】

User can execute command EXB browser in forms of: click button  in outer tool from main menu of Tool, or click button  in outer tool from option card of Tool.


Execute command EXB browser, following file browser interface will pop up.



Pic9-9 interface of EXB View

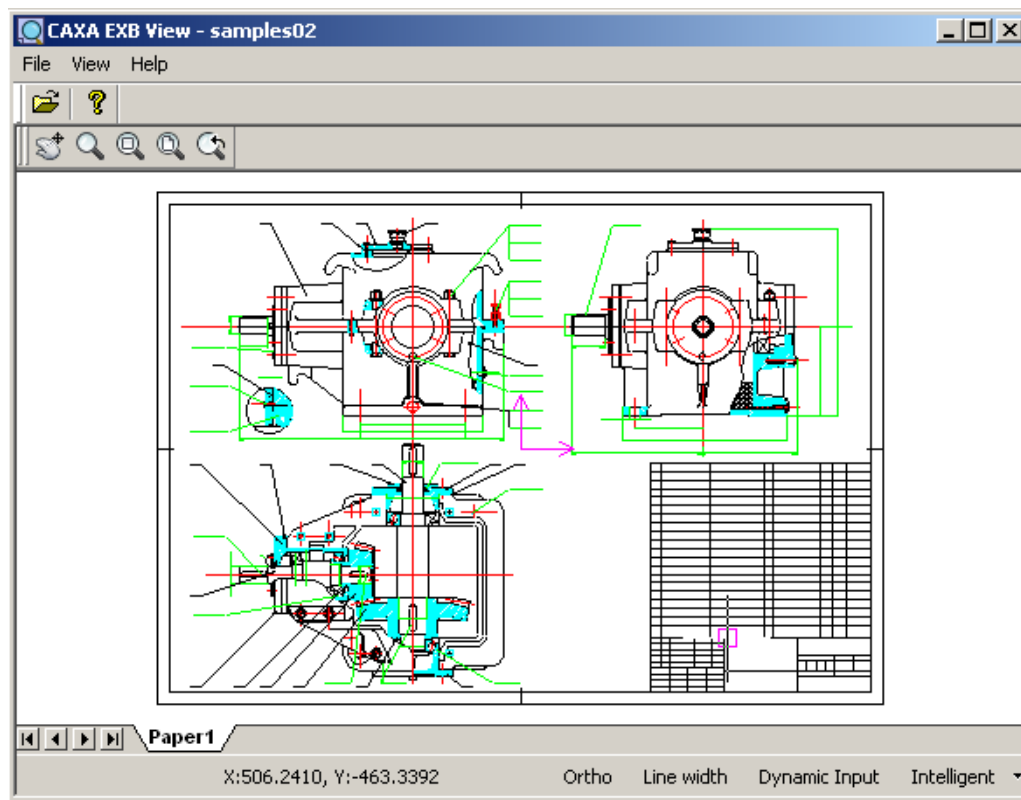
Method for using EXB browser.

- (1) Open file


Click  in the main menu of **file or in tool bar**, or press shortcut key **Ctrl+O**, corresponding dialog box will pop up, in which select one EXB file.

- (2) Browse file

When one EXB file is opened, following figure will be shown in the window.



Pic9-10 View EXB file

Then drag mouse wheel to zoom graph, or click corresponding button  to pan, dynamic pan, display window, display all, display previous etc.

9.3 System query

CAXA Draft provides function of query, by which user can inquire about coordinates of point, distance between two points, angle, element attribute, area, center of gravity, perimeter, inertia moment etc.




9.3.1 Query point coordinates

【Command】 id

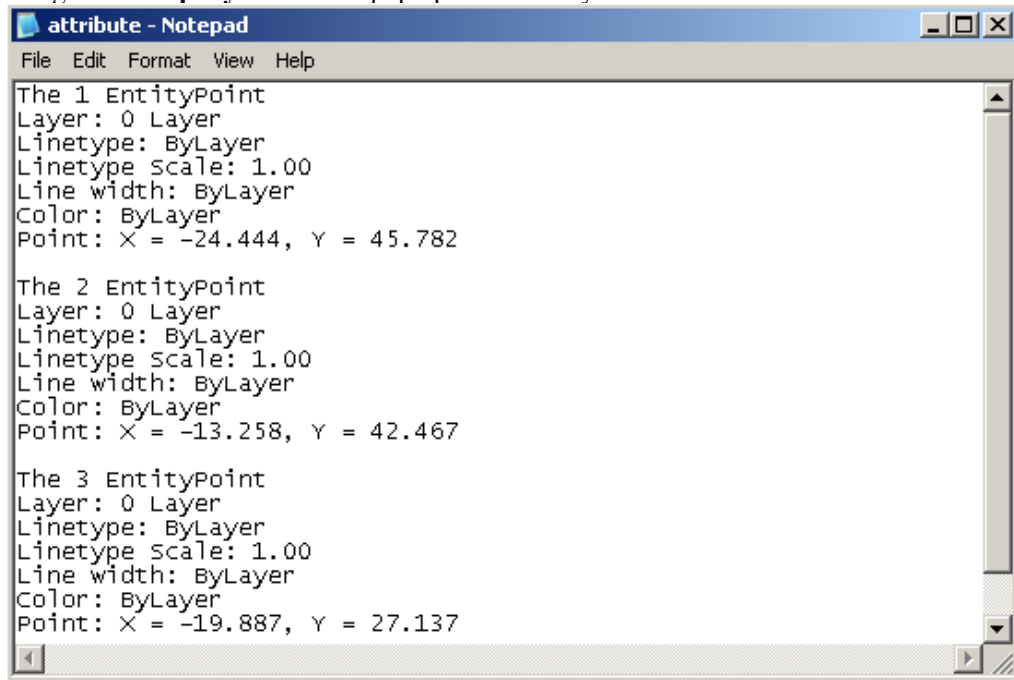
【Icon】 

【Definition】 User can inquire point coordinates under each kind of tool point style, and inquire several points simultaneously.

【Step】

User can execute command ID in forms of: click button  in Query from main menu of Tool. Click  in Query from option card of Tool, click button  in tool bar of Query tool. Use command ID directly.

Execute command id, Select the point to be inquired as per instruction, it will be red when it is selected, Continue to select other points, once the selection is finished, click right key to confirm, the dialogue box **inquiry result** will pop up immediately.



Pic9-11 Query point coordinates

All inquired point coordinates are listed as per selection order in the box. Click button Save to save query result in text file.




9.3.2 Query two points distance

【Command】 dist

【Icon】 

【Definition】 Inquire distance between any two points.

【Step】

User can execute command dist in forms of: click button  in Query from main menu of Tool. Click  in Query from option card of Tool, click button  in tool bar of Query tool. Use command dist directly.

Execute command dist, select two points to be inquired as per instruction, when second point is selected, the dialogue box **inquiry result** will pop up immediately, the inquired distance between two points and increment, in X axis and Y axis ,of second point relating to first point are listed in the box.




9.3.3 Query Angle

【Command】 angle

【Icon】 

【Definition】 Inquire center angle, intersection angle of two lines and intersection angle of three points.

【Step】

User can execute command angle in forms of: click button  in Query from main menu of Tool. Click  in Query from option card of Tool, click button  in tool bar of Query tool. Use command angle directly.

Click command angle, an immediate menu will pop up, in which user can select center angle, intersection angle of two lines and intersection angle of three point, specify query mode, then select object as per prompt.




9.3.4 Query Element attribute

【Command】 list

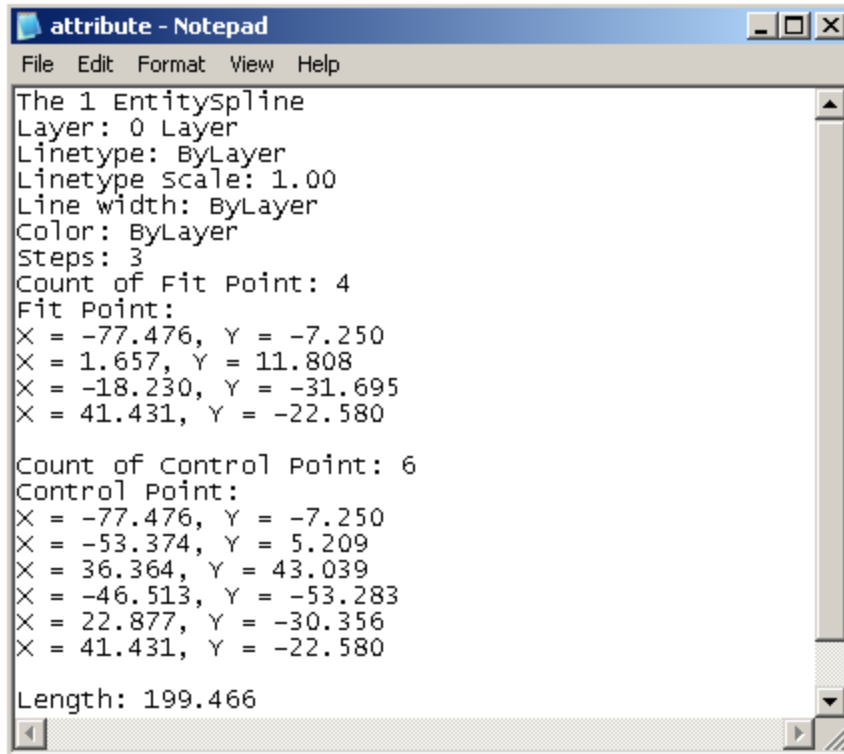
【Icon】 

【Definition】 query selected object attribute, and list its query result.

【Step】

User can execute command list in forms of: click button  in Query from main menu of Tool. Click  in Query from option card of Tool, click button  in tool bar of Query tool. Use command list directly.

Execute command List, Pick entity to be inquired as per instruction, click right key when the picking is finished. The dialog box of “inquiry result” will pop up, in which the attribute of picked element will be listed as per picking order. Following is query result for the selected spline.



Pic9-12 Query Element attribute

9.3.5 Query Perimeter

【Command】circum

【Icon】

【Definition】User can inquire about total length of a series of end to end curves, regardless of closed or opened.

【Step】

User can execute command circum in forms of: click button in Query from main menu of Tool. Click in Query from option card of Tool, click button in tool bar of Query tool. Use command circum directly.

Execute command Circum, pick curves as per instruction, the dialog box inquiry result will pop up immediately, in which the total length of a series of end to end curves and length of individual curve are listed orderly.




9.3.6 Query area

【Command】 area

【Icon】 

【Definition】 Inquire area of a closed region or complicated figure containing several closed regions. The region can be basic curve or closed region formed by super curves.

【Step】

User can execute command area in forms of: click button  in Query from main menu of Tool. Click  in Query from option card of Tool, click button  in tool bar of Query tool. Use command area directly.

Click command area, select **Add area or Reduce area**, **Add area** in immediate menu means adding the picked closed area to other areas, **reduce area** indicates subtracting area of the closed region from other areas. Complicated graphic area can be computed through this immediate menu.

Picking a point in the closed region to be computed, after selection, the curves composing closed loop will be highlighted in red. When picking is finished, click right key to confirm, the dialog box “ inquiry result” will pop up, total area will be shown for the selected closed regions.




9.3.7 Query Center of gravity

【Command】 barcen

【Icon】 

【Definition】 Inquire gravity of a closed region or complicated figure containing several closed regions. The figure can be basic curve or closed region formed by senior curves.

【Step】

User can execute command barcen in forms of: click button  in Query from main menu of Tool. Click  in Query from option card of Tool, click button  in tool bar of Query tool. Use command barcen directly.

Click command barcen, both inquiry operation is the save as that of query area. But when the picking is finished, the position of gravity center is shown in the dialog box of **inquiry result**.




9.3.8 Query Inertia moment

【Command】 iner

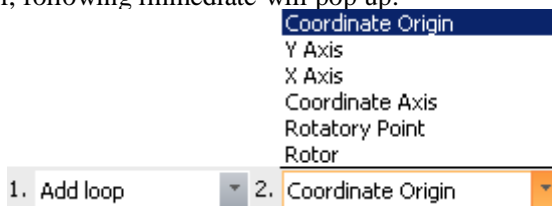
【Icon】 

【Definition】 Inquire inertia moment of a closed region or complicated figure containing several closed regions related to any rotor and turning point, the figure can be basic curve or closed region formed by senior curves.


【Step】


User can execute command iner in forms of: click button  in Query from main menu of Tool. Click  in Query from option card of Tool, click button  in tool bar of Query tool. Use command iner directly.

Click command iner, following immediate will pop up.



Pic9-13 Query Inertia moment

Click  in immediate menu, “add loop” and “reduce loop” can be switched, the operation method is the same as that of area inquiry.

Click  in immediate menu, there are origin of coordinates, Y coordinate axis, X coordinate axis, rotor, and turning point. Among which the former three modes are the selected distributing regions, corresponding to inertia moment of origin, X-axis, Y-axis respectively. User also could set the rotor and turning point by these two modes, then the system will compute inertia moment according to the settings.

When the closed region and rotor or turning point are picked as per instruction, the inertia moment will be shown in the **inquiry result** dialog box.

9.4 File retrieve

【Command】 idx



【Icon】 

【Definition】 Search proper file from local computer or from network computer as

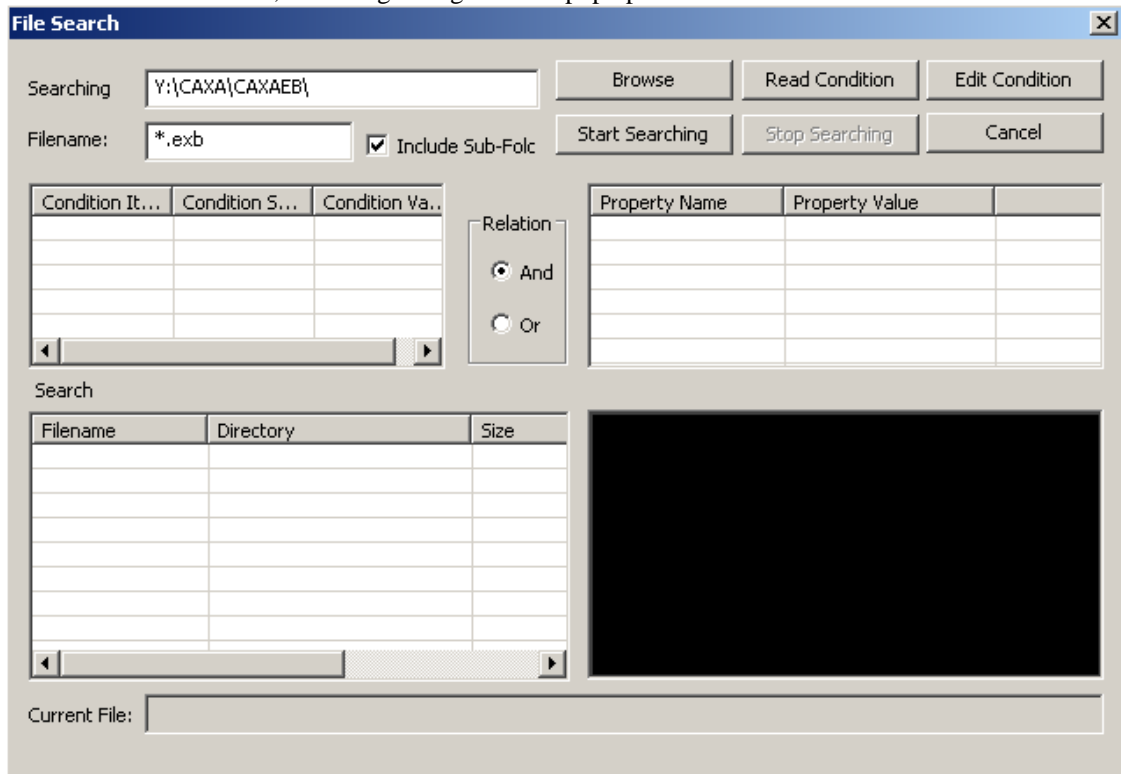
per retrieve condition.

Searching condition can be specified **path**, **file name**, or **attribute condition** in the title bar of CAXA Draft file.

【Step】

User can execute command idx in forms of: click button  in main menu of File. Click  in Tool from option card of Tool, Use command idx directly, press short cut key Ctrl+F.

Click command idx, following dialog box will pop up.



Pic9-14 File search

Parametric in above dialog box is explained as follows:

(1) Searching path

Nominate searching range, fill in by hand or and click button **browse** and select via path browsing dialog box. The option **Including subindex** can determine searching only in the current index or including the subindex. When searching according to file name or the extension name, the wildcard “*” is supported.

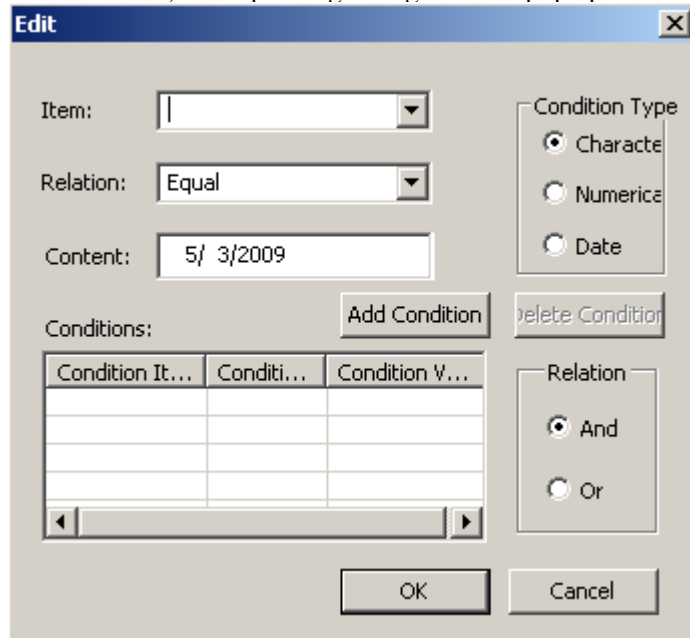
(2) Attribute condition

Attribute condition: display information condition in title bar, and nominate logical relations between conditions (**AND** & **OR**). The information condition in title bar can be edited by

activating editing-condition dialog box.

(3) Edit condition

Click button **edit condition**, corresponding dialog box will pop up for editing condition.



Pic9-15 edit condition

The added condition is shown in the condition-display area, user can edit the condition content in the lower condition-editing area or additional condition-editing area.

If condition needs to be added, it must be altered first, then click button Add condition, a new condition option is produced. The condition is made up of condition entry, condition character and condition value.

Condition entry indicates attribute title in title bar, such as design time, name etc. There are selectable attribute in the pull-down bar.

Condition character is divided into three types: character type, value type and date type. Each type has several options, select through pull-down bar of condition character.

Condition value is divided into three types accordingly: character type, value type and date type. Value can be inputted in edit box behind the condition value. If condition type is date type, current date will be shown in edit box, click arrowhead on the right, the date selection dialog box will be activated for selecting date.

For example: To search drawings designed before Aug. 20th, 2000. Select design date in pull-down bar of condition entry, select date type in condition type, and select before in condition character, select Aug. 20th, 2000 in condition value, click Add condition , and a

condition is produced and shown in the condition display area.

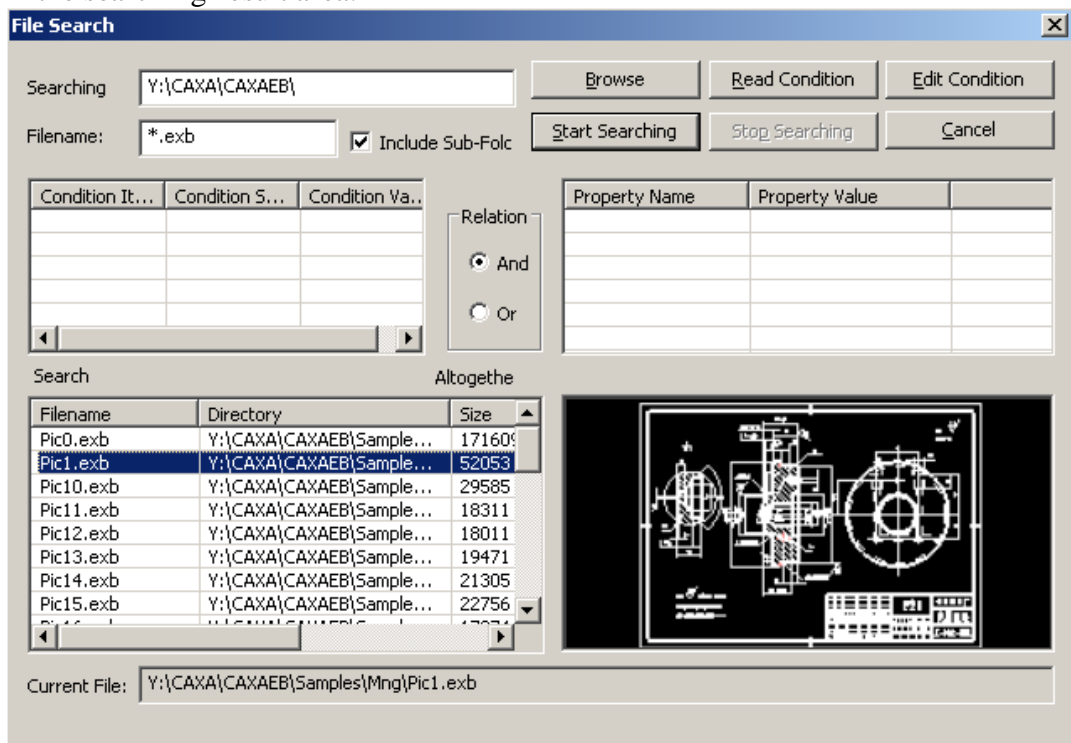
Condition relations: Select condition relations when two or more conditions are added, they are and & or.

The selected condition in condition display area can be deleted or be edited.....

When the condition is edited, click button add condition, corresponding condition content will be displayed in condition display box, click button ok, dialog box save will pop up, then the edited condition can be saved. In future, User can click button read condition directly, and the existing inquiry condition will be opened.

(4) Search result

Click button **Start searching**, and files match condition in the path are shown in the searching-result area.



Pic9-16Searching result

The searching result shows the information and total number of the searched file. If the file's total number is more than 100, the searching will be stopped. Select one searching result, the title bar content can be seen in the right attribute area, user can preview graphic in the preview area, double click it, the file will be opened by EB electronic board.

User can set check button Preview mode to determine mode of previewing graphics.

Bitmap: If select bitmap to preview graphics, only the whole graphics can be previewed, which can't be zoomed in, zoomed out or panned. But this preview mode will not occupy too much computer resource, and its display speed is fast.

Browser: If select browser to preview graphics, zoom in, zoom out, translate can be applied when previewing drawing. Since this preview mode will occupy some computer resource, the display speed is slow.



9.5 Module manager

【Command】 manage

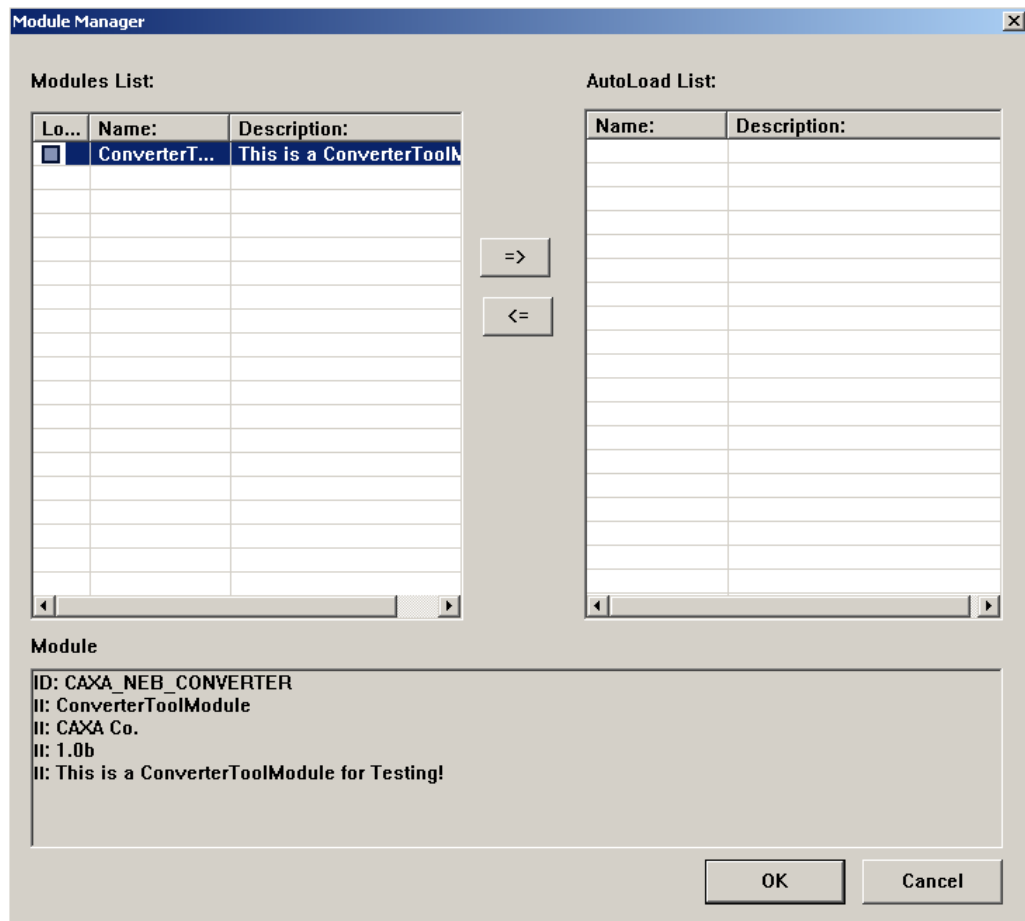
【Icon】 

【Definition】 Add-on and manage other function module.

【Step】

User can execute command manage in forms of: Click  in the main menu of File, click button  from Tool panel of Tool option card, click command Manage directly.

Click command Manage, following dialog box will pop up:



Pic9-17 dialog box of Module manager

Method for using module manager:

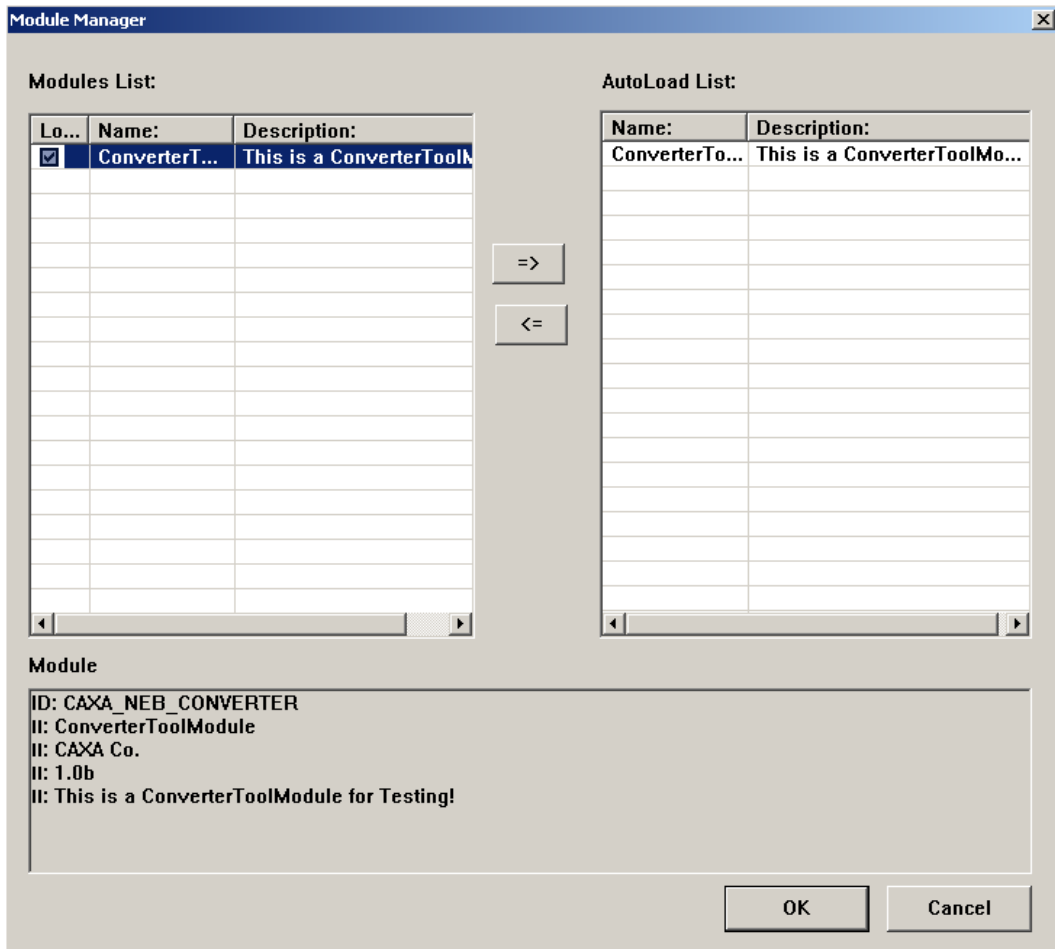
(1) Add-on and Uninstall

Usable module list is on the left side of above dialog box, select or cancel check box in front of module to add-on or uninstall module.


(2) Auto add on:

Select one module from the module list, then click button **=>**, and the module will be set as auto add on. Close this program and restart, the module will be add-on automatically, user can use it directly.

If the module is set as auto-add on, it will be displayed in auto-add on list. As shown in figure 9-18.



Pic9-18 Autoload mould

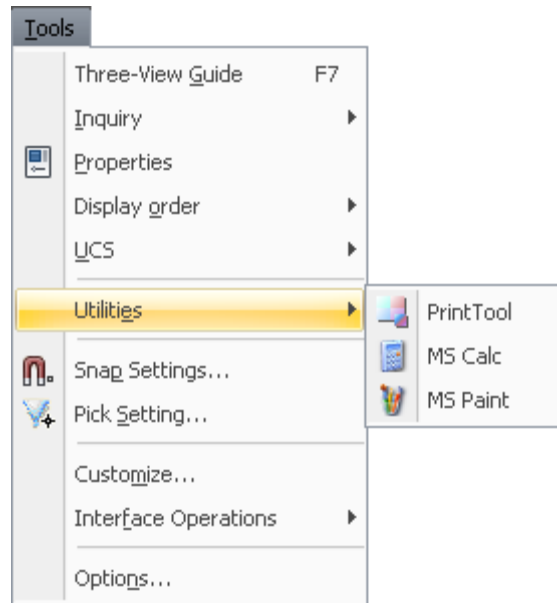
Select such module in the auto add on list, click button  to cancel auto add on setting.

9.6 External tool

User can configure other program as external tool via customize in section 8.9.2.

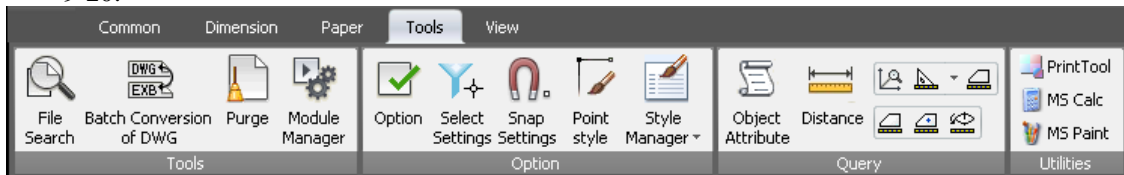
Configured external tool position is shown as follows:

- (1) Select external tool in the menu of Tool, as shown in figure 9-19,.



9-19 External tool

- (2) External tool in the place of function area is Tool option card, as shown in figure 9-20.



Pic9-20 External tool on Ribbon bar

Click button of external tool directly to start programe.

Appendix

Command list

Command name	Key command	Abbrev key	Keyboard
New	New		Ctrl+N
Open file	Open		Ctrl+O
Close file	Close		Ctrl+W
Save file	Save		Ctrl+S
Save as another file	Saveas		Ctrl+Shift+S
Merge file	Merge		
Part save	Partsave		
Plot	Plot		Ctrl+P
File retrieve	Idx		CTRL+F
Transfer DWG file by lot	DWG		
Module manager	Manage		
Purge	Purge		
Exit	Quit		Alt+F4
Undo	Undo		Ctrl+Z
Redo	Redo		Ctrl+Y
Select all	Selall		Ctrl+A
Cutclip	Cutclip		Ctrl+X
Copyclip	Copyclip		Ctrl+C
Copy with base point	Copywb		Ctrl+Shift+C
Pasteclip	Pasteclip		Ctrl+V
Paste as block	Pasteblock		Ctrl+Shift+V
Paste from	Specialpaste		Ctrl+R
Insert Object	Insertobj	OBJ	
Link	Setlink		Ctrl+K
OLE	OLE		
Clean	Delete		Delete
Erase all	Eraseall		
Refresh	Refresh		

Refresh all	Refreshall		
View Window	Zoom	Z	
Pan	Pan	P	
Zoom All	Zoomall	ZA	F3
Home	Home		Home
View scale	Vscale		
View Previous	Prev	ZP	
Next	Next	ZN	
Zoom in	Zoomin		PageUp
Zoom out	Zoomout		PageDown
Dynamic translate	Dyntrans		Mouse Center key/Shift+ Mouse left key
Dynamic Zoom	Dynscale		Mouse wheel/Shift+ Mouse right key
Layer setting	Layer		
Linear setting	Ltype		
Color setting	Color		
Line weight setting	Wide		
Dot style	Ddptype		
Text style	Textpara		
Dimension style	Dimpara		
Down-lead line style	Ldtype		
Geometric tolerance form	Fcstype		
Line	Line	L	
Draw two point line	Lpp		
Angle line	La		
Draw angular bisector	Lia		
Draw tangent line / normal	Ltn		
Bisector of two line segments	Bisector		
Parallel line	Parallel	LL	
Circle	Circle	C	
Draw circle via given centre and radius	Cir		

Draw two point circle	Cppl		
Draw circle via three point	Cppp		
Draw circle via two point-radius	Cppr		
Arc	Arc	A	
Three point arc	Appp		
Centre , start point and centre angle for drawing arc	Acsa		
Draw arc via given two point and radius	Appr		
Draw arc via given centre, radius , start and terminal angle	Acra		
Draw arc via given start point-end point-centre angle	Asea		
Draw arc via given start point-radius-start/end angle	Asra		
Spline	Spline	SPL	
Point	Point	PO	
Formula curve	Fomul		
Ellipse	Ellipse	EL	
Draw rectangle	Rect		
Draw regular polygon	Polygon		
Polyline	Pline		
Draw centre line	Centerl		
Draw offset line	Offset	O	
Draw hatching	Hatch	H	
Filling	Solid		
Text	Text		
Part enlarge	Enlarge		
Draw wave line	Wavel		
Dual fold line	Condup		
Draw arrow	Arrow		
Draw Arc fit spline	Nhs		
Insert image	Insertimage		

Image manage	Image		
Creat block	Block		
Insert block	Insertblock		
hide block	Hide		
Attribute definition	Attrib		
Paste as block	Pasteblock		Ctrl+Shift+V
Block edit	Blockedit	BE	
Block reigning edit	Refedit	RE	
Pick Symbol	Sym		
Symbol definition	Symdef		
Graphic Library management	Symman		
Drive symbol	Symdrv		
Graphic Library exchange	Symexchange		
Size dimension	Dim	D	
Basic dimension	Powerdim		
Base line dimension	Basdim		
Continuous dimension	Contdim		
Three point angle dimension	3parcdim		
Angle continuous dimension	Continuearcdim		
Half dimension	Halfdim		
Big arc dimension in measure dimension	Arcdim		
Radial dimension	Radialdim		
Gradient dimension	Gradientdim		
Curvature radius dimension	Curvradiusdim		
Coordinate dimension	Dimco	DC	
Origin dimension	Origindim		
Fast dimension	Fastdim		
Free dimension	Freedim		
Alignment dimension	Aligndim		
Hole site dimension	Hsdim		
Down-lead dimension	downleaddim		

Automatic list in coordinate dimension	Autolist		
Chamfer dim	Dimch		
Down-lead note	Ldtext		
Geometrical tolerance	Fcs		
Technique requirement library	Speclib		
Erase	Erase		Delete
Eraseline	Eraseline		
Move	Move	MO	
Copy	Copy		
Rotate	Rotate	RO	
Mirror	Mirror	MI	
Scale	Scale	SC	
Array	Array	AR	
Corner	Corner	CO	
Fillet	Fillet		
Multi Fillet	Fillets		
Chamfer	Chamfer		
Outside chamfer	Chamferaxle		
Inside chamfer	Chamferhole		
Multi Chamfer	Chamfers		
Sharp Corner	Sharp		
Trim	Trim	TR	
Edge	Edge	ED	
Break	Break	BR	
Stretch	Stretch	S	
Explode	Explode	EX	
Dimension edit command	Dimedit		
Dimension Drive	Drive		
Match	Match		
switch dimension style	Dimset		
edit text para	Textset		
Text finding and replace	Textoperation		

3D view navigation	Guide		
Query point coordinates	Id		
Query two points distance	Dist		
Query Angle	Angle		
Query Element attribute	List		
Query Perimeter	Circum		
Query area	Area		
Query Center of gravity	Barcen		
Query Inertia moment	Iner		
System statuse	Status		
Property option panel	Properties		
View to top	Totop		
View to bottom	Tobottom		
View to front	Tofront		
View to back	Toback		
Text to top	Texttotop		
dimension to top	Dimtotop		
text or dimention to top	Tdtotop		
Create UCS	Newucs		
Manage UCS	Switch		
print tool	Printool		
EXB View tool	Exbview		
CAXA calculator	Caxacalc		
calculator	Calc		
paint	Paint		
Capture setting	Potset		
Pick filter	Objectset		
Interface customize	Customize		
Interface reset	Interfacereset		
Add on interface configuration	Interfaceload		
Save interface configuration	Interfacesave		
System configuration	Syscfg		

close window	Close		
close all window	Closeall		
	Cascade		
	Horizontally		
	Vertically		
	Arrange		
Help	Help		F1
	Frminit		
	Headerfill		
	Tblhtrans		
	Tbltransform		
	Ptnoadd		
	Rehead		
	Refrm		
	Ortho		F8
	Showide		
	Showd		
	Catch		F6
	Interface		F9
	Blockin		
	Blockout		
	Blockonqwo		
	Blockonqws		
	Blockq		
			F4
			F5
			F2
			F7
			Ctrl+B
			Ctrl+E
			Ctrl+U
			Ctrl+M
			Ctrl+T

			Ctrl+Q
			Ctrl+I