CADian 2012 Utility

User Manual

Dec 24, 2011

http://www.cadian.com
CADian Soft Limited

* Applicable version : later than CADian2012

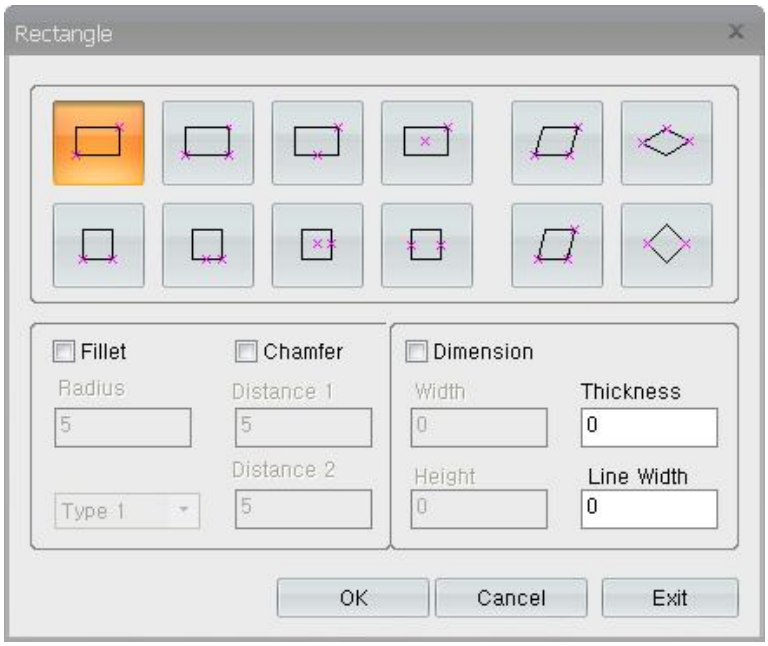
CADian 2012 Utility

1. Drawing

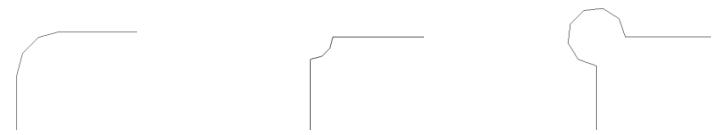
1. Drawing rectangles

This command draws a rectangle or a rhombus with a function of fillet, chamfer and scallop.

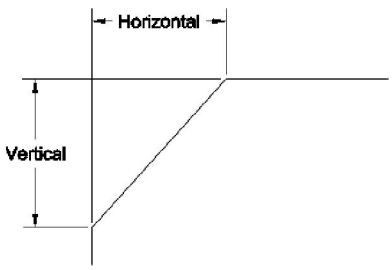
Command: DRECT
Dialog/<Start Point> : Enter "D". *(invoke a dialog.)*



How to draw: Select a shape and a method of drawing in the dialog. The first row in the dialog is for both rectangles and squares. The second row is only for drawing sqaures.
Fillet : select the shape of fillet and the radius value

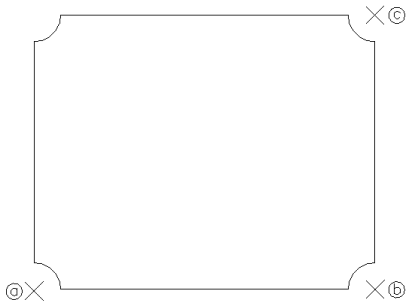


Chamfer : enter a vertical and horizontal distance from a vertex.

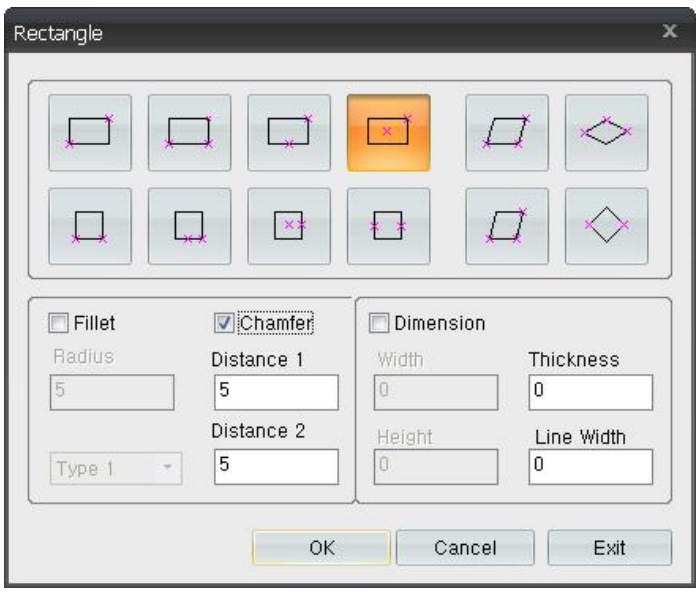


Dimension : function of drawing a box by entering its width and height.
- Width : set the horizontal length of the box.
- Height : set the vertical length of the box.
- Thickness : set 3-dimensional thickness of the box.
- Line Width : set the width of the line for the box.

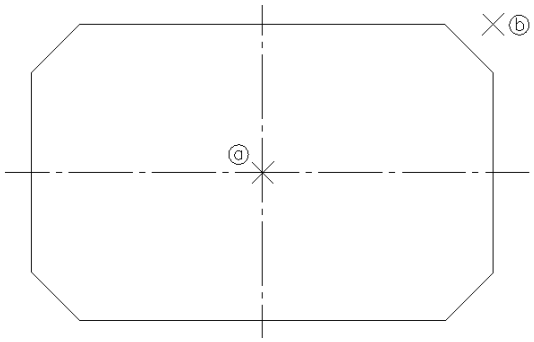
Dialog/<Start Point> : Select ⓐ point.
Dialog/<Start Point> : Select ⓑ point.
Dialog/<Start Point> : Select ⓒ point.



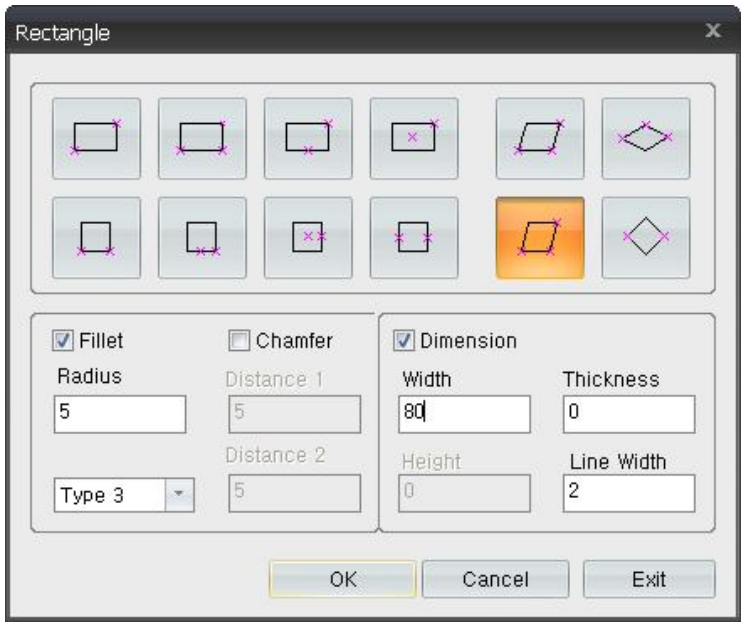
Command: DRECT
Dialog/<Start Point> : Enter "D". *(invoke a dialog.)*



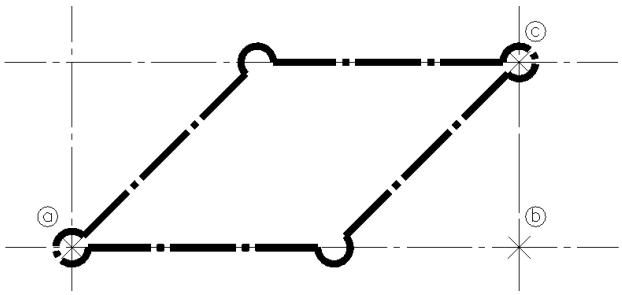
Dialog/<Start Point> : Select ⓐ point.
Dialog/<Start Point> : Select ⓑ point.



Command: DRECT
Dialog/<Start Point> : Enter “D”. *(invoke a dialog.)*



Dialog/<Start Point> : Select ⓐ point.
Dialog/<Start Point> : Select ⓑ point.
Dialog/<Start Point> : Select ⓒ point.

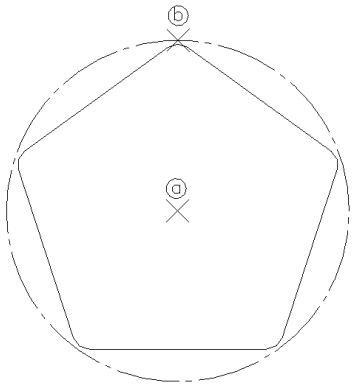


2. Drawing Polygons

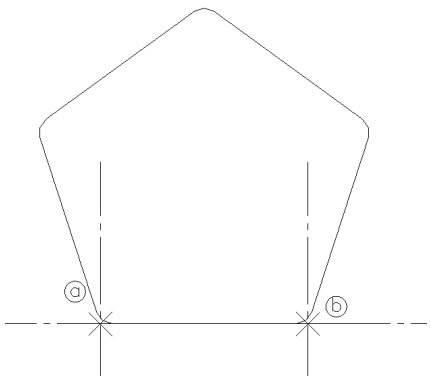
This command draws polygons with a function of fillet and chamfer automatically.

Command: PGON
Number of sides <5> : Enter “5” or press Enter key. *(Enter a number of sides for the preferred polygon.)*
Polygon Start Point or [Fillet/Champer] : Enter “F”.
Polygon Fillet Radius <0.0000> : Enter “5”. *(Enter the radious value for the fillet.)*

Polygon Start Point or [Fillet/Champer] : Select ⓐ point.
Polygon [Inscribed/Circumscribed/Edge/fixAngle]/<Circumscribed Radius>: Enter “I”.
Polygon [Inscribed/Circumscribed/Edge/fixAngle]/< Inscribed Radius> : Select ⓑ point. *(if the radius of the inscribed circle is known, enter the radius.)*



Number of sides <5>: Enter“5” or press Enter key. *(Enter the number of sides of the preferred polygon.)*
Polygon Start Point or [Fillet/Champer] : Enter “F”.
Polygon Fillet Radius <0.0000> : Enter “5”. *(Enter the radius of the preferred fillet.)*
Polygon Start Point or [Fillet/Champer] : Select ⓐ point.
Polygon [Inscribed/Circumscribed/Edge/fixAngle]/<Circumscribed Radius>: Enter “E”.
Polygon [Inscribed/Circumscribed/Edge/fixAngle]/< Inscribed Radius> : Select ⓑ point.*(entering the length of the polygon’s sides is also acceptable.)*

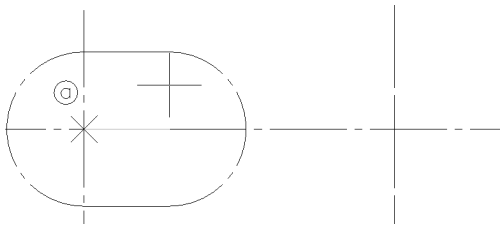


3. Drawing a slot hole

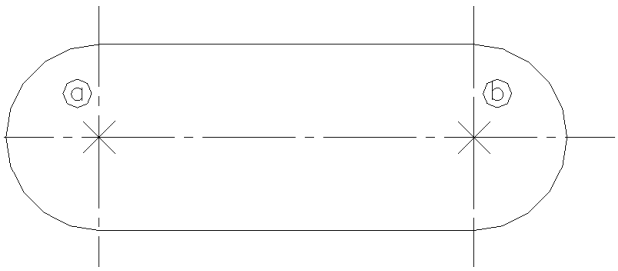
This command draws a slot hole automatically.

<Slot Hole Drawing 1>

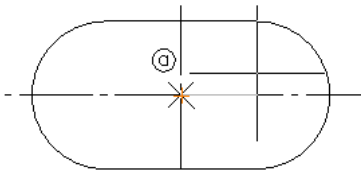
Command: DSLOT
Slot Hole[Slot/Flange/slde/Centers] / <Start Point> : Select ⓐ point.
Slot Radial : Enter “20”.*(enter the radius of the slot hole.)*



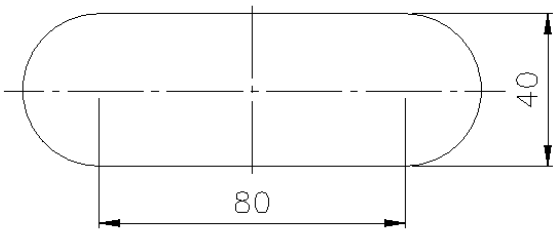
Slot Length : Select ⓑ point.*(enter the distance between ⓐ point and ⓑ point.)*



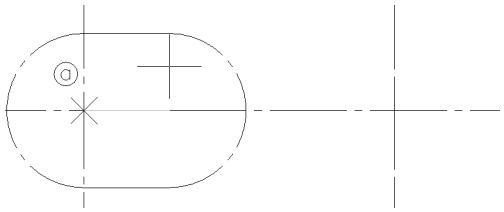
<Slot Hole Drawing 2>
Command: DSLOT
Slot Hole[Slot/Flange/sIde/Centers] / <Start Point> : Enter “C”.
Slot Hole[Slot/Flange/sIde/Centers] / <Start Point> : Select ⓐ point.
Slot Radial : Enter “20”.*(enter the radius of the slot hole.)*



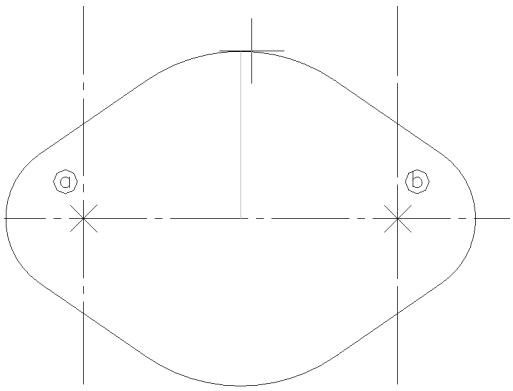
Slot Length : Enter ”40”.*(Enter the distance between ⓐ point and the center of the arc.)*



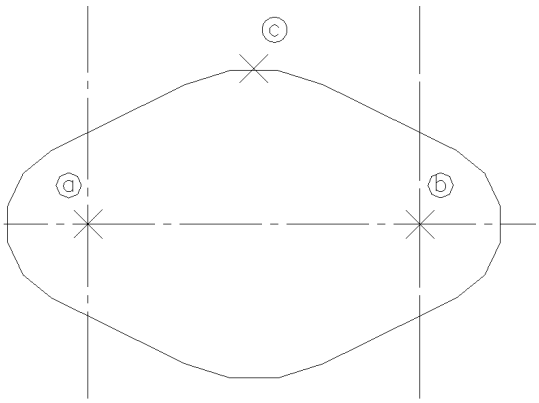
<Flange Drawing >
Command: DSLOT
Slot Hole[Slot/Flange/sIde/Centers] / <Start Point> : Enter “F”.
Slot Hole[Slot/Flange/sIde/Centers] / <Start Point> : Select ⓐ point.
Slot Radial : Enter “20”.



Slot Length : Select ⓑ point.



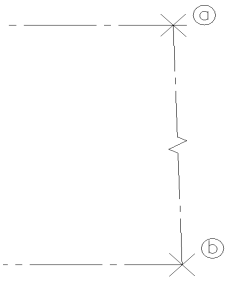
Slot Radius : Select ⓒ point.



4. Drawing a break line

This command draws a break line automatically on the drawing.

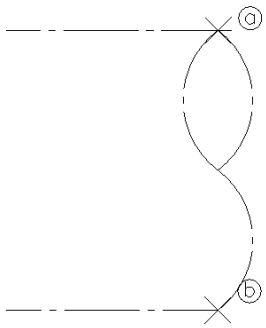
Command: BKLINE
Break Line [Size(5)]/<Start Point> : Enter "S".*(Enter the distance of the line to be broken)*
Size(5): Enter "10".
Break Line [Size(10)]/<Start Point> : Select ⓐ point.
End Point : Select ⓑ point.
Break Point<Mid point> : Press Enter key.*(it draws a break line at the mid point of the line. Drawing a break line at any point on the line is also acceptable. Select a point close to the point to draw a break line.)*



5. Drawing a cylinder section

This command draws a cylinder section of a round bar or a pipe.

Command : BKSEC
Cylinder Section [Single/Double]<Sing Start point> : select ⓐ point.*(Enter D to draw a cylinder section on the both sides of the bar/pipe.)*
End Point : Select ⓑ point.

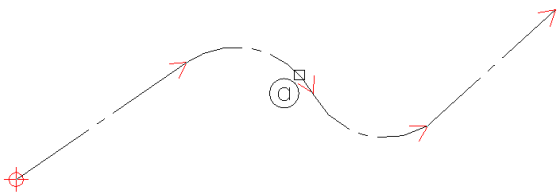


2. Drawing a curve line

1. Display the direction of the curve line

It displays the direction of the curve line. The arrows to illustrate the direction of the curve disappear automatically after drawing.

Command : CURVEDIR
Select Curve [Vertex disp/Definite disp] : Select ⓐ point to select a curve.



2. Change the direction of the curve line

Command : CURVEDIR
If the curve line is selected, the curve line changes its direction to the other way.

<This function cannot be performed due to API problem. To be added in CADian 2012 authenticity soon>

3. Change to a curve polyline

If you select a line, arc, circle or object, it will change the attribute to a polyline.

Command : CURVEPOLY

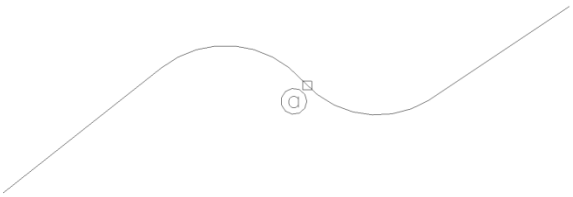
4. Change the polyline width

If you select a polyline, line, or arc, the polyline's width will change automatically. If you select a line and an arc, then selected object's attribute will change to polyline.

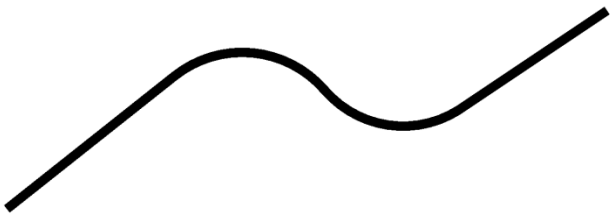
Command : POLYWIDTH

Select entities : Select ⓐ point to select a curve.

Select entities : Finish selecting by pressing Enter key.



Polyline Width <1> : Enter "2".

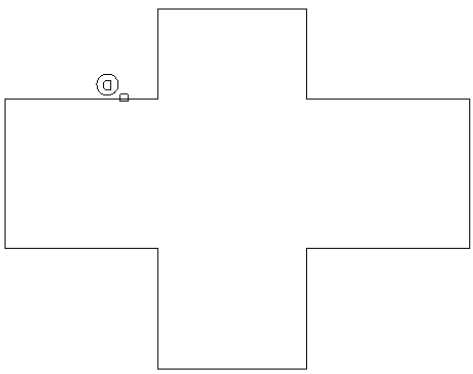


5. Edit a polyline

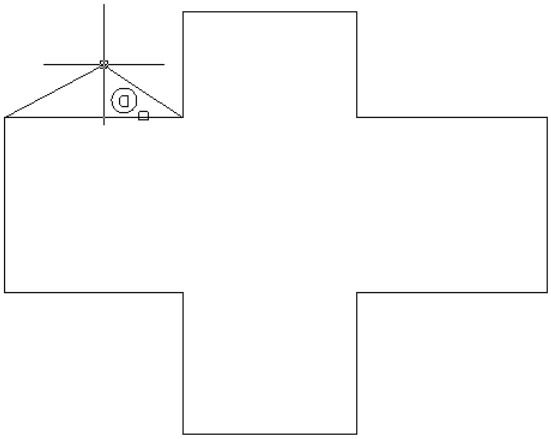
Drawn straight polylines can be edited to curved polylines and vice versa. Also, new object can be inserted/deleted in between two objects. It is easy to edit polylines when orthomode is turned off.

Command : POLYEDIT

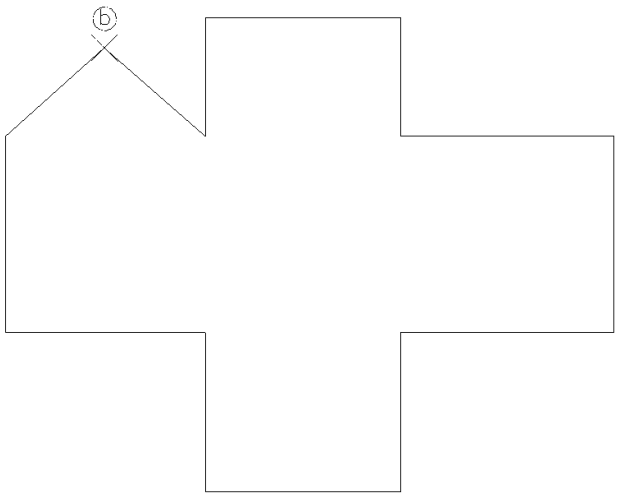
Select Object : select the polyline by selecting ⓐ point



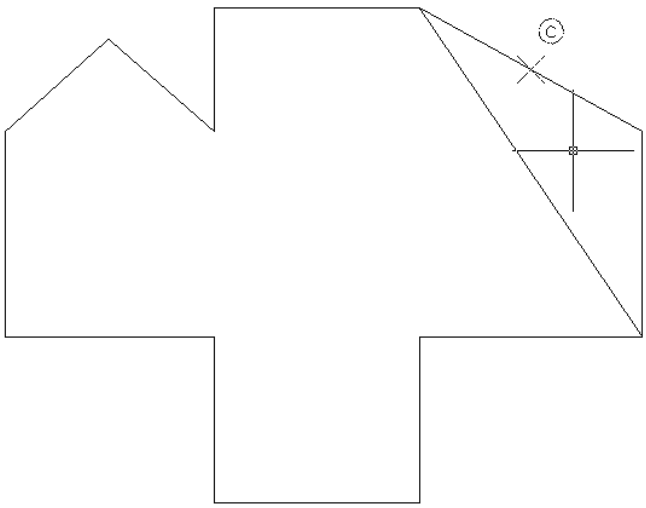
Polyline Insert [Remove/Move/Stretch/arC/Tanarc/Line/Fixed/eXit] <Append point> : *after selecting a polyline, dragging is performed in order to insert an object to the point where the cursor is located.*



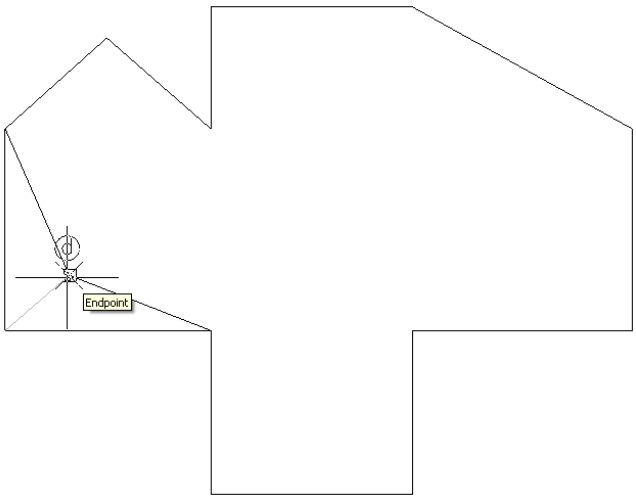
Polyline Insert [Remove/Move/Stretch/arC/Tanarc/Line/Fixed/eXit] <Append point> : select ⓑ point.



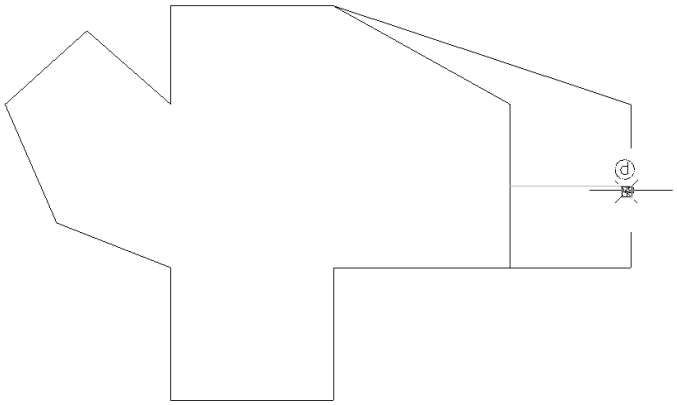
Polyline Insert [Remove/Move/Stretch/arC/Tanarc/Line/Fixed/eXit] <Append point> : enter "R".
Polyline Remove [Append/Move/Stretch/arC/Tanarc/Line/Fixed/eXit] <Remove point> : select ⓒ point. *(delete the object between two objects)*



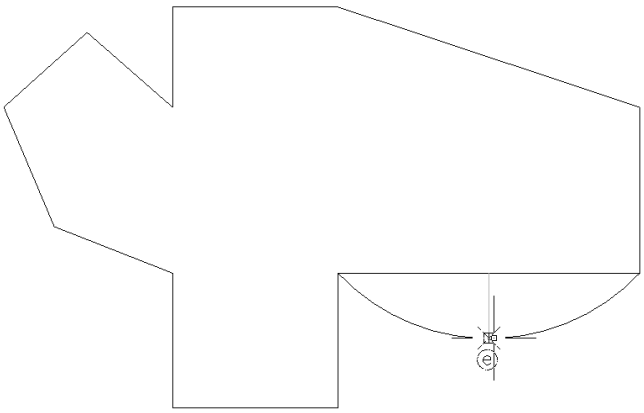
Polyline Remove [Append/Move/Stretch/arC/Tanarc/Line/Fixed/eXit] <Remove point> : Enter “M”.
Polyline Move [Append/Remove/Stretch/arC/Tanarc/Line/Fixed/eXit] <Move point> : select ⓓ point.



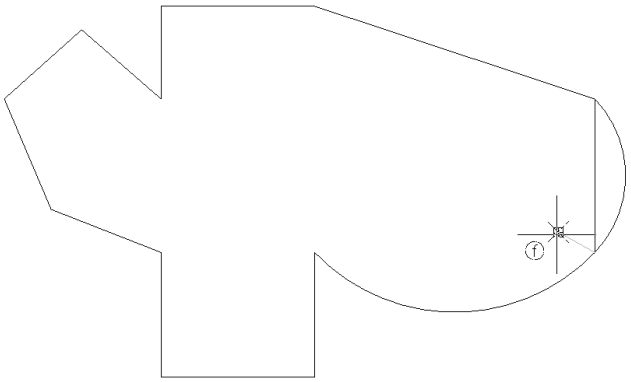
Polyline Move [Append/Remove/Stretch/arC/Tanarc/Line/Fixed/eXit] <Move point> : Enter “S”.
Polyline Stretch [Append/Move/Remove/arC/Tanarc/Line/Fixed/eXit] <Stretch point>: Select ⓓ point.



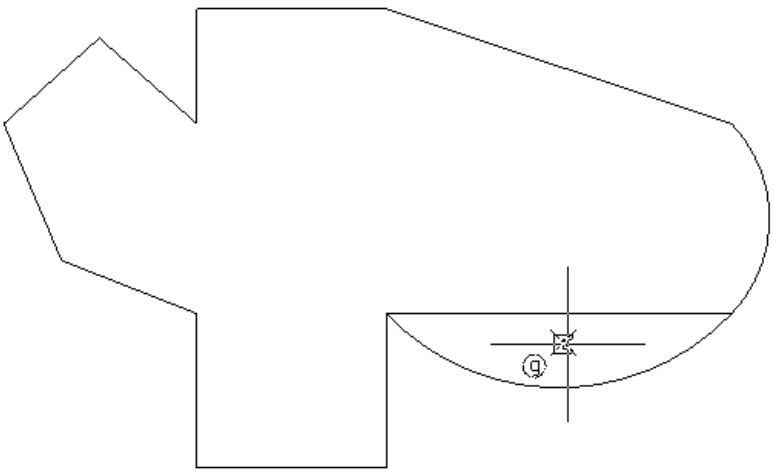
Polyline Stretch [Append/Move/Remove/arC/Tanarc/Line/Fixed/eXit] <Stretch point> : Enter “C”.
Polyline Arc [Append/Move/Remove/Stretch/Tanarc/Line/Fixed/eXit] <Arch point>: select ⓔ point.



Polyline Arc [Append/Move/Remove/Stretch/Tanarc/Line/Fixed/eXit] <Arch point> : Enter "T".
Polyline Tangent Arc [Append/Move/Remove/Stretch/arC/Line/Fixed/eXit] <Tanget point>: select ⓕ point. *(draws a inscribed circle with the nearest object to the cursor location.)*

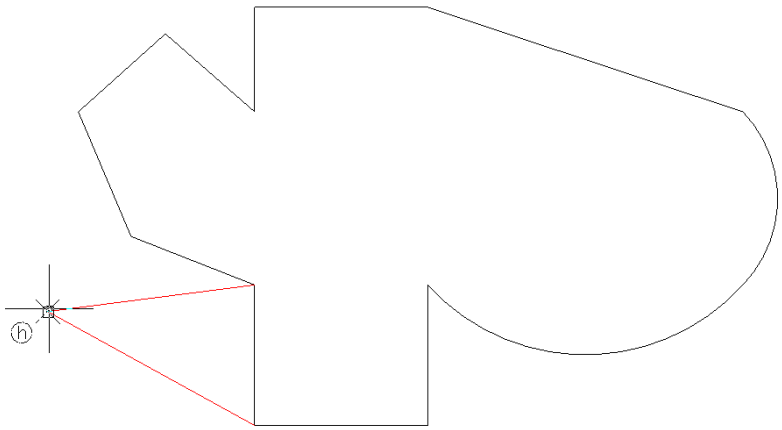


Polyline Tangent Arc [Append/Move/Remove/Stretch/arC/Line/Fixed/eXit] <Tanget point>: Enter "L".
Polyline Line [Append/Move/Remove/Stretch/arC/Tanarc/Fixed/eXit] <Line point> : select ⑨ point.*(when moving cursor to the arc, obejct changes to the straight line.)*



Polyline Line [Append/Move/Remove/Stretch/arC/Tanarc/Fixed/eXit] <Line point> : Enter "F".*(whenever the cursor is moving, insertion is performed at the cursor location. Fixed*

function helps fixing insertion location to where the cursor is.)
Polyline Line [Append/Move/Remove/Stretch/arC/Tanarc/Fixed/eXit] <Line point> : select ⓗ point.



Polyline Line [Append/Move/Remove/Stretch/arC/Tanarc/Fixed/eXit] <Line point> : press "X" to exit the command.

6. Change z value for objects
This command changes object's 3-dimensional coordinates values when entering 3-dimensional coordinates (z value) after selecting the object.

Command : CURVEZ

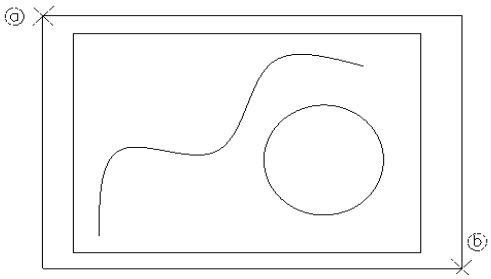
7. Connect lines
This command connects a polyline to objects that are connected to the straight line and an arc.

Command : LINEJOIN

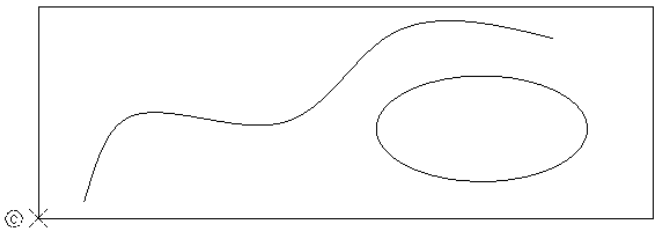
3. Modification

1. XY scale
Selected object's x and y scales can be changed. After the scale is edited, selected polyline object is divided into lines and arcs.

Command : XYSCALE
Select Entity : select an object by selecting ⓐ and ⓑ points



Base Point : select ⓒ point.
X Scale factor or Pick Point [Reference/Copy/ <2> : Enter “1”.
Y Scale factor <3> : Enter “2”.

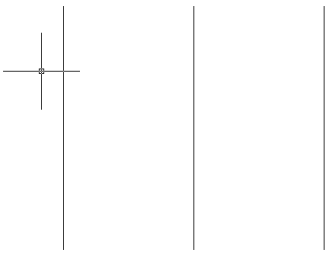


XY Scale factor <2,1> : *(displays the final applied scale factor.)*

2. Double Offset Distance
When selecting an object, two ends of the object has the offset distance automatically.

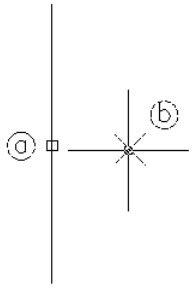
Command : DOST
Double Offset Distance <100> : Enter “1000”.
Select Entity : select the object by selecting ⓐ point.

Select Entity : press Enter key and quit the command.



3. Repeat Offset Distance
This command draws the selected certain number of objects with the repeated offset distance.

Command : RPOST
Repeat Offset Distance :<10> Enter “1000”
Select Entity : select the object by selecting ⓐ point.
Side for parallel : select the direction by selecting ⓑ point.



Repeat Count : Enter “3”.

Select Entity : Press Enter key and quit the command.

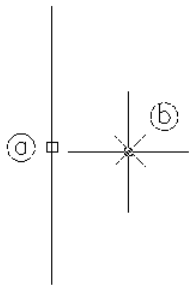
4. Increment offset distance

This command draws the selected object continuously with the offset distance.

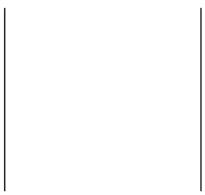
Command : INCOST

Select Entity : select the object by selecting ⓐ point.

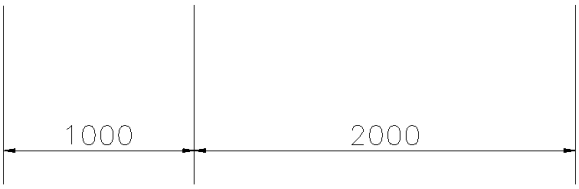
Side for parallel : select the direction by selecting ⓑ point.



Offset Distance : Enter “1000”.



Offset Distance : Enter “2000”.



Offset Distance : Quit the command by pressing Enter key.

5. Increment double offset

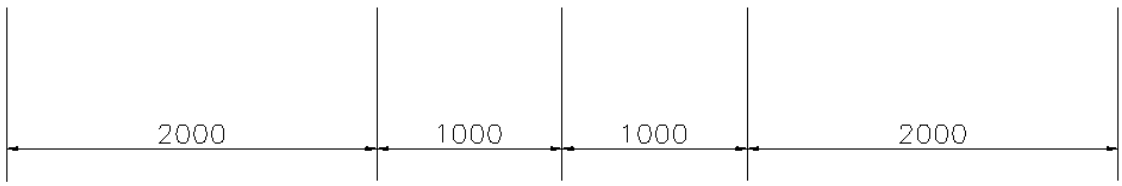
This command copies the object with the preferred offset distance on both sides continuously.

Command : INCDOST

Select Entity : select the object by selecting ⓐ point.

Offset Distance : Enter “1000”.

Offset Distance : Enter “2000”.



Offset Distance : quit the command by pressing Enter key.

6. Erase Empty Object

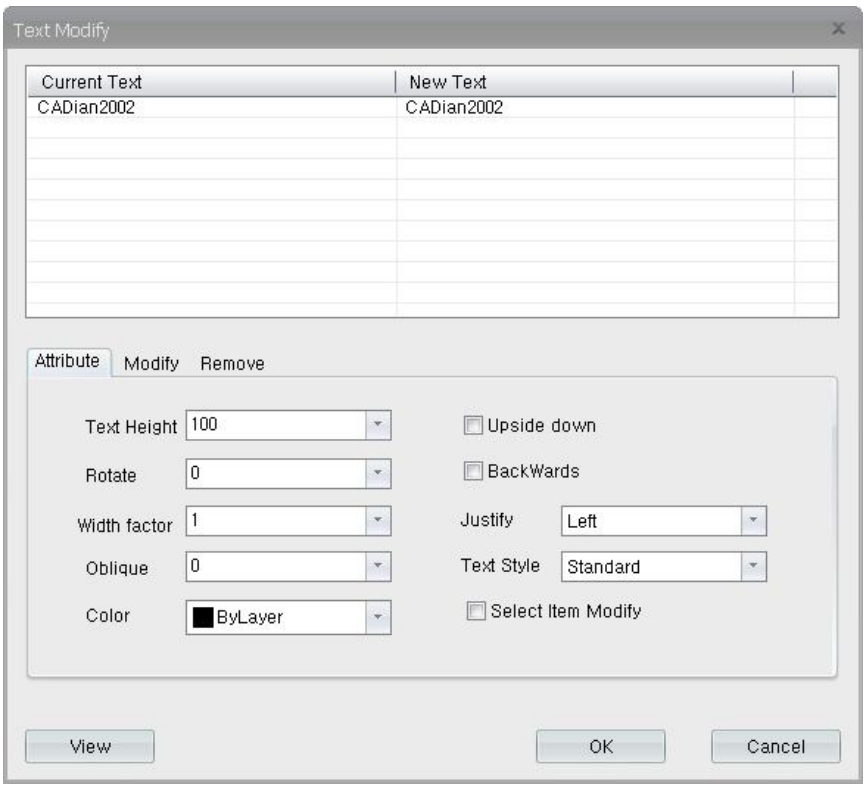
This command erases a line, polyline, arc, circle, (object with the same start point and end point), empty text, point(optional)

Command : ERASEEMPTYENT

4, Text

1. Text Modification

This command changes the attributes of the selected text (height, color, angle, etc), inserts the text into the selected location, and deletes the text. Dialog below pops up when selecting the object. Using the dialog's function makes text modification way easier.

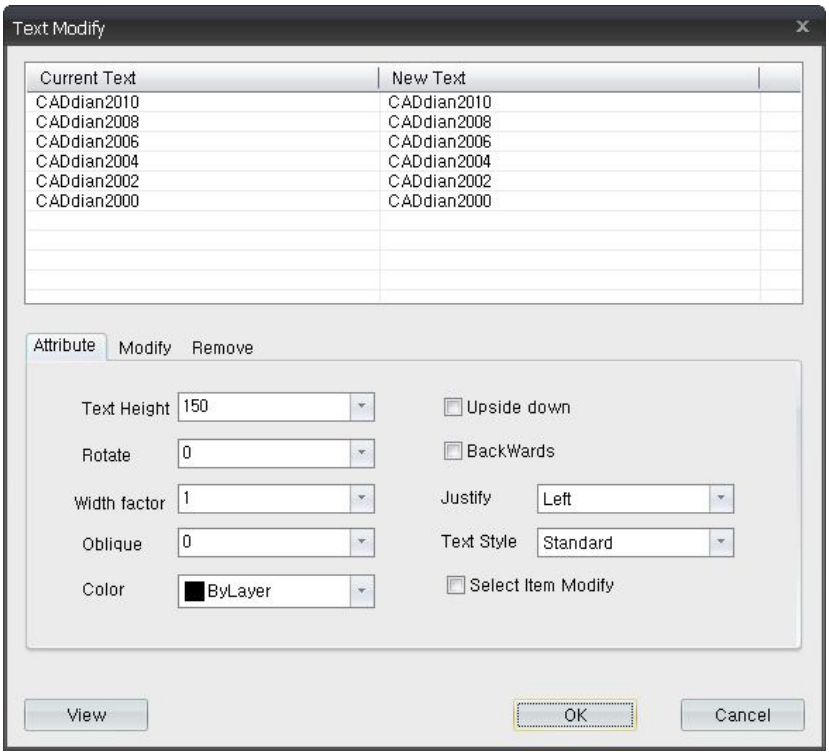


<Attributes Modification >

Command : TEXTMODIFY

Select Text : select the object by selecting ⓐ and ⓑ points.

Select Entity : Press Enter key and quit the selection. The dialog below pops up.



Text Box : Selected text is arranged in the table format and manual text modification is available for text on the right by clicking.

Text Height : When the dialog pops up, it displays the current text height. If text height is changed, all selected text height is automatically changed. If clicking on the toggle box on the bottom right, only the selected text height in the text box is changed.

Rotate : When the dialog pops up, current text rotation angle in the system is displayed. If text rotation angle is changed, all selected text rotation angle is changed accordingly. If toggle boxes on the bottom right are checked, only the selected text box rotation angle is changed.

Width factor : When the dialog pops up, current text width factor is the system is displayed. If text width factor is changed, all the selected text width factor is changed accordingly. If toggle boxes on the bottom right are checked, only the selected text box width factor is changed.

Oblique : When the dialog pops up, current text oblique in the system is displayed. If text oblique is changed, all the selected text oblique is changed accordingly. If toggle boxes on the bottom right are checked, only the selected text box oblique is changed.

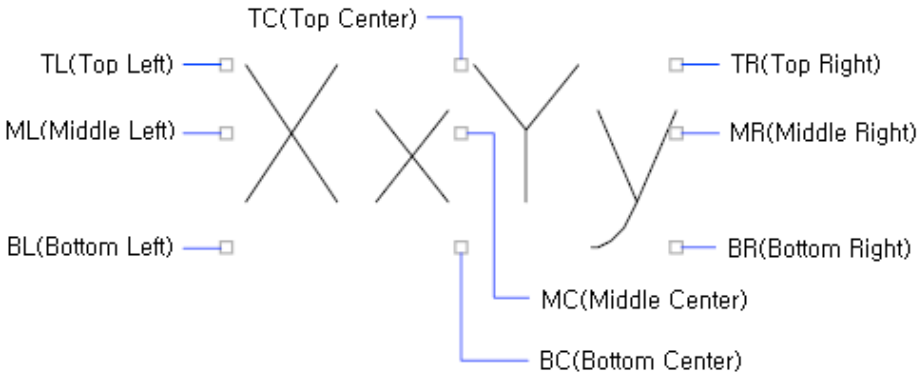
Color : When the dialog pops up, current text color in the system is displayed. If text color is changed, all the selected text color is changed accordingly. If toggle boxes on the bottom right are checked, only the selected text box color is changed.

Upside down : This function flips the text upside down when the checkbox is checked.

BackWards : This function switches the text backwards when the checkbox is checked.

Justify : When the dialog pops up, current text justify in the system is displayed. If text

justify is changed, all the selected text justify is changed accordingly. If toggle boxes on the bottom right are checked, only the selected text box justify is changed.



Text Style : When the dialog pops up, current text style in the system is displayed. If text style is changed, all the selected text style is changed accordingly. If toggle boxes on the bottom right are checked, only the selected text style is changed.

Select Item Modify : If checkboxes are checked, only the selected text attributes values are changed in the text box.

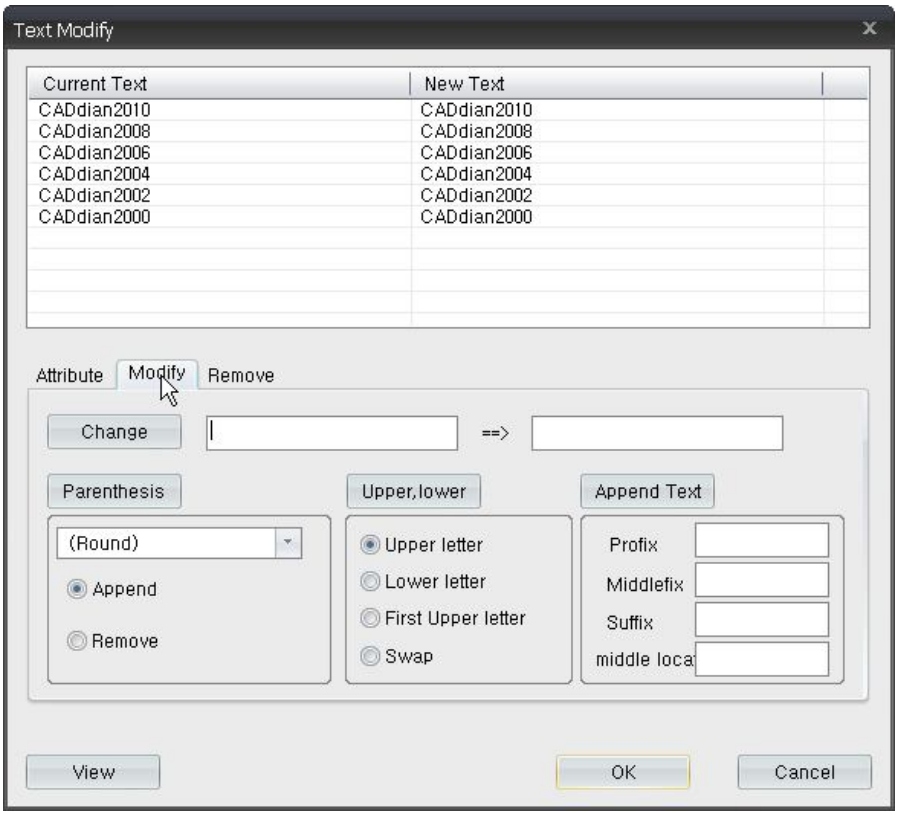
View : This function gives preview of the text attribute modification before it is applied to the drawing. If 'view' button is clicked, the dialog disappears and text's modified attribute is applied. Press "Esc" in order to switch back to dialog. Text attribute does not change before 'Ok' button is pressed.

<Text Modification >

Command : TEXTMODIFY

Select Entity : Select text object by selecting ⓐ and ⓑ point.

Select Entity : Press Enter key and quit the selection. The dialog below pops up. Click on the Modify tab.



Change : Specific text in all the selected text are automatically changed.
ex) 20 => 30 → CADian2002 => CADian3002

Parenthesis : If Parenthesis button is clicked, a parenthesis is added/deleted to all the selected text.
ex) CADian2000 => (CADian2000), {CADian2000} => CADian2000

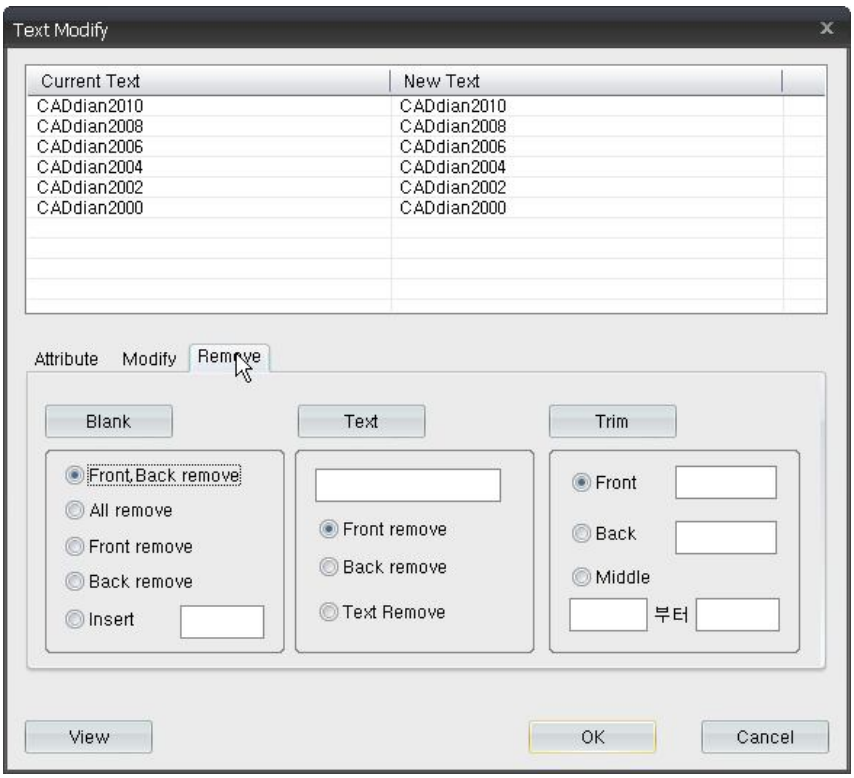
Upper,lower : If Upper, lower button is clicked, text is automatically changed to upper/lower letter depending on the option selected.
- Upper letter : All the selected text is changed to upper letter.
ex) CADian2000 => CADIAN2000
- Lower letter : All the selected text is changed to lower letter.
ex) CADian2000 => cadian2000
- First Upper letter : The first letter of all the selected text is changed to upper letter.
ex) cadian2000 => Cadian2000
- Swap : Upper letter in the selected text is changed to lower letter and lower letter in the selected text is changed to upper letter.
ex) CADian2000 => cadIAN2000

Append Text : Text can be added to the preferred part of the selected text. Text can be added as a prefix, midfix, or suffix to the text one by one or all together.
ex) prefix: added text : "ie" (CADian2000 => ieCADian2000)
midfix: added text : "yes" midpoint : 6 (CADian2000 => CADianyes2000)
suffix : added text : "ok" (CADian2000 => CADian2000ok)

<Text Delete>
Command : TEXTMODIFY

Select Entity : select the object by selecting ⓐ and ⓑ points.

Select Entity : Press Enter key and quit the selection. A dialog will pop up. Click on the remove tab.



Blank : A blank is added or removed from the selected location in the selected text.

Text : Once delete button is pressed, the specified text in the selected text is removed. If there are more than one of the specified text, then it is deleted from the left of the text. Once the button is pressed again, the next one from the left is deleted. Upper and lower cases should be distinguished.

Front Remove : Enter “a” and execute the function
ex) CADian2000 => an2000
Back Remove : Enter “a” and execute the function
ex) CADian2000 => CADia

Text Remove : Enter “a” and execute the function
ex) CADian2000 => CADin2000

Trim : Once the preferred digits of text is entered and trim button is pressed, text is automatically trimmed. Text’s upper and lower cases are distinguished.

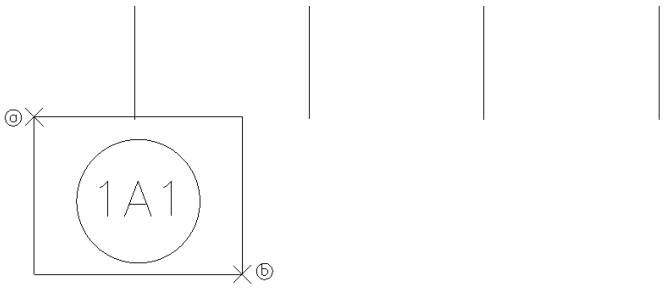
Front : Enter 3 and execute
ex) CADian2000 => ian2000
Back : Enter 4 and execute
ex) CADian2000 => CADian
Middle : Enter 3~6 and execute
ex) CADian2000 => CAD2000

2. Number Increment

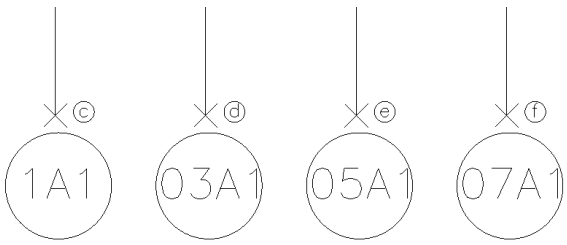
This command copies the text repeatedly by incrementing the selected number.

<Front Text Increment>

Command : TEXTINC
Select Entity : Select the object by selecting ⓐ and ⓑ points.

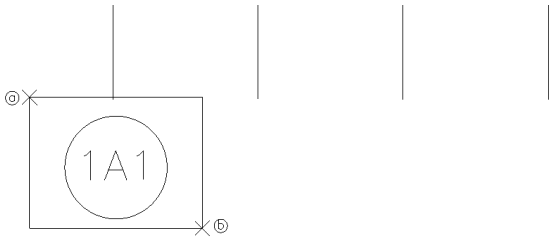


Select Entity : Press Enter key and quit the selection.
Incremental Value <1> : Enter “2”.
Digits <1> : Enter “2”.
Placement of Number[Front/Back] : Enter “F”.
Base Point : Select ⓒ point.
Next Point : Click ⓓ,ⓔ,ⓕ points repeatedly.
Next Point : Press Enter key and quit the command.

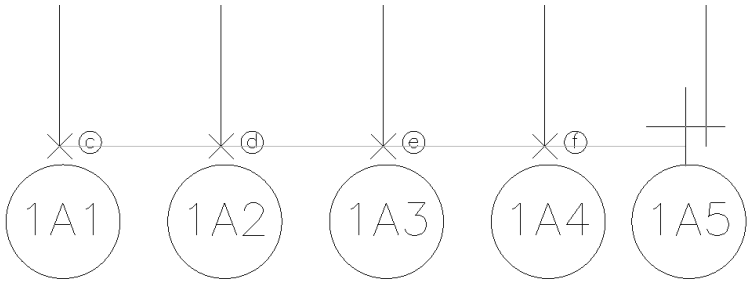


<Back Text Increment >

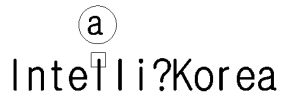
Command : TEXTINC
Select Entity : Select the object by selecting ⓐ and ⓑ point.



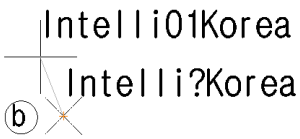
Select Entity : Press Enter key and quit the selection.
Incremental Value<1> : Enter “1”.
Digits<1> : Enter “2”.
Placement of Number[Front/Back] : Enter “B”.
Base Point : Select ⓒ point.
Next Point: Select ⓓ,ⓔ,ⓕ points repeatedly.
Next Point: Press Enter key and quit the command.



<”?” Text Increment>
If there is “?” in the selected text, “?” can be incremented by changing into numerical value. If both “?” and number are included, “?” is ignored and only number is incremented.
Select Entity : select the object by selecting ⓐ point.



Incremental Value<1> : Enter “1”.
Digits<1> : Enter “2”.
Placement of Number[Front/Back] : Press Enter key.

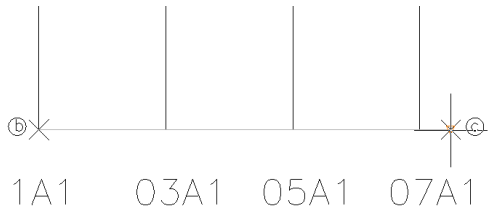


Base Point : Select ⓑ point.
Next Point: Select a point where the text should be drawn.*(If a point is continuously selected, then the number is incremented and text is automatically drawn.)*

3. Linear Continuous Text
Text is copied with the same distance that was determined by the number that is included in the text.

<Drawing 1>
Command : LINCTEXT
Select Entity : Select the text by selecting ⓐ point.

Select Entity : Press Enter key.
Incremental Value<1> : Enter “2”.
Digits<1> : Enter “2”.
Placement of Number[Front/Back] : Enter “F”.
Base Point : Select ⓑ point.
Space : Enter “1000”. (Distance can be entered by using a mouse)
Position : select ⓒ point.*(If dragging the crosshair to the right, the incremented text appears with 1000 space value. Text appears and text is created automatically after a point is selected.)*



<Drawing2>
If drawn text includes “?”, “?” can be switched to number and text is incremented. If “?” and number are included at the same time, “?” is ignored and only number is incremented.

Command : LINCTEXT
Select Entity : Select the text by selecting ⓐ point.

Select Entity : Press Enter key.
Incremental Value<1> : Enter “2”.
Digits<1> : Enter “2”.
Placement of Number[Front/Back] : Press Enter key.
Base Point : Select ⓑ point.
Space : Enter “1000”. (Distance can be determined by using a mouse)
Position : Select ⓒ point.*(If dragging the crosshair to the right, the incremented text appears with 1000 space value. Text appears and text is created automatically after a point is selected. If a number is included in the text, then “?” is ignored and only number is incremented.)*

4. Circular Continuous Text

This command draws a text in a circle direction with the incremented value of the number included in the text.

<Drawing1>

Command : PINCTEXT

Select Text : Select the number by selecting ⓐ point.

Select Entity : Press Enter Key.

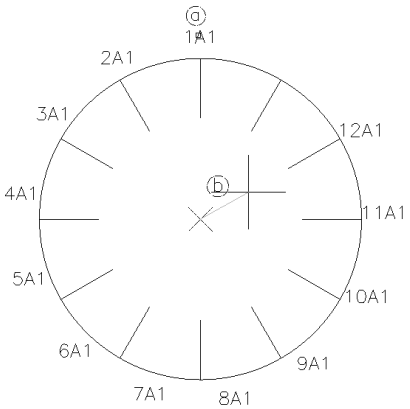
Incremental Value <1> : Enter “1”.

Digits <1> : Enter “1”.

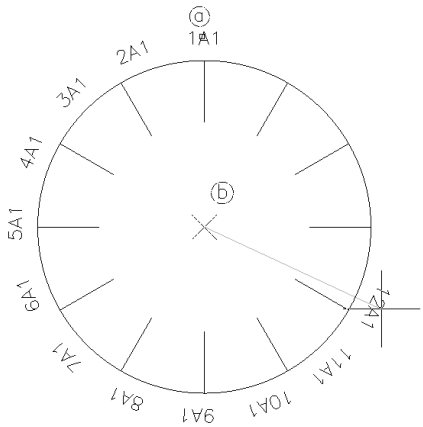
Placement of Number[Front/Back]: Enter “F”.

Center Point : Select ⓑ point. *(Select the center point of the circle by using OSNAP<“CEN”>.)*

Column [Angle/Rotate] <4>: Enter “12”.*(Enter the number of the rotation copy.)*



Draw Angle [Rotate] (+=CCW, -=CW) < 360 >: Enter “R”.



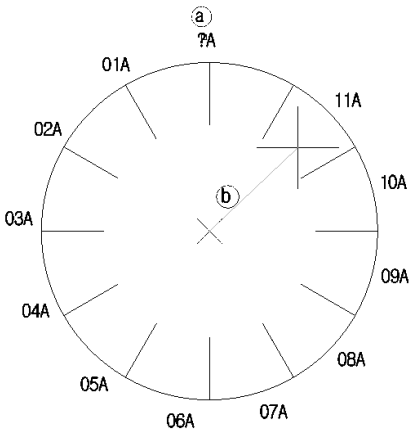
Draw Angle [Rotate] (+=CCW, -=CW) < 360 >: Enter “360”.

<Drawing2>

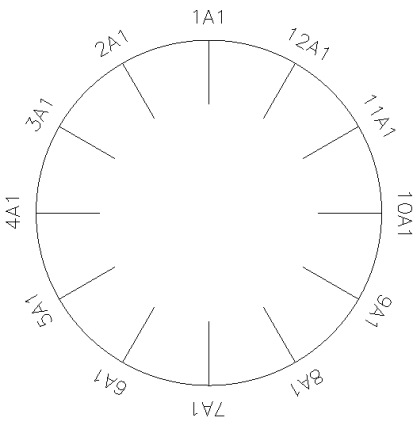
If “?” is included in the text, “?” can be incremented by changing it to the numerical value. If both “?” and number are included, “?” is ignored and only number is incremented.

Command : PINCTEXT

Select Text : Select the text by selecting ⓐ point.
Select Entity : Press Enter key.
Incremental Value <1> : Enter “1”.
Digits <1> : Enter “2”.
Placement of Number[Front/Back]: Enter “F”.
Center Point : Select ⓑ point. *(Select a center point of the circle using OSNAP<“CEN”>.)*
Column [Angle/Rotate] <4>: Enter “12”. *(Enter the number of the rotation copy.)*



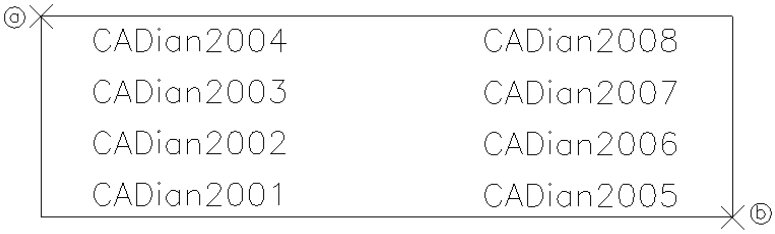
Draw Angle [Rotate] (+=CCW, -=CW) < 360 > : Enter “360”.



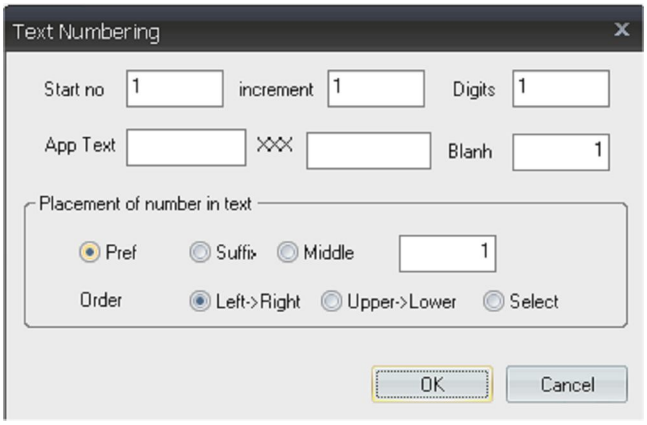
5. Numbering

This command adds a number with a rule in the text.

Command : TEXTNUMBERING
Select Text : Select ⓐ and ⓑ point.
Select Entity : Press Enter key.



Start No. [Dialog] < 1 > : Enter "D".*(dialog below pops up.)*



Start No. : Enter the first value of the Start Number.
Increment : Enter the number to be incremented.
Digits : Enter the digits of the added number.*(ex: 001)*
App Text : Enter the text to be added in the front or back of the text. *(ex: TA-001-L)*
Blank : Enter the number of blanks to be added in the front or back of the text.

Placement of number in text
- Prefix : Added in the front of the selected text *(ex: TEXT => T1L TEXT)*
- Suffix : Added in the back of the selected text *(ex: TEXT => TEXT T1L)*
- Middle : Added in the middle of the selected text *(ex: TEXT => TE T1L XT)*

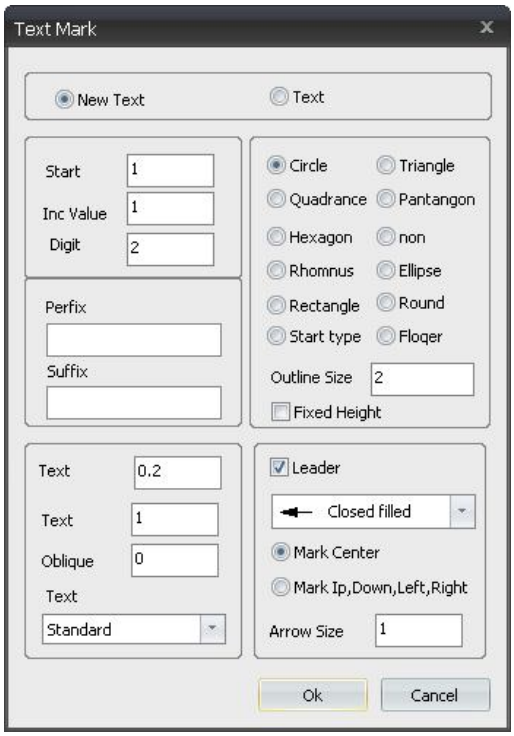
Order
- Left->Right : number is incremented from left to right starting from the base point.
- Upper->Lower : number is incremented from top to bottom starting from the base point.
- Select : Number is incremented by selection.

After pressing the ok button…….

T1L CADian2004 T5L CADian2008
T2L CADian2003 T6L CADian2007
T3L CADian2002 T7L CADian2006
T4L CADian2001 T8L CADian2005

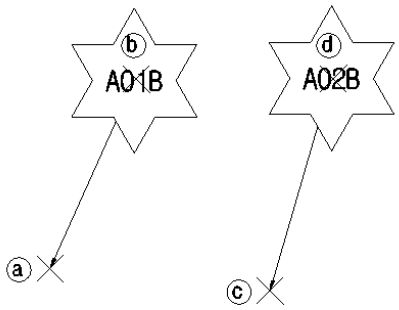
6. Text Outline

This command draws a shape automatically to the drawn text or draws the text on the shape.



New Make : Draws a shape and a text in the specified location.
Text : Draws a shape on the pre-drawn text.
Start No : Select the starting number.
Inc Value : Selecting the incremental value.
Cipher : Enter the digits.
Preface : Enter the text to be added in the front of the number. It is not necessary to enter the text.*(ex: AA001)*
Tail : Select a text to be added to the back of the number. It is not necessary to enter the text.*(ex: 001BB)*
Leader : Selects drawing function of a leader.
Mark Center : Draws the end point of the leader around the center of the circle.
Mark Up,Down,Left,Right : Depending on where the leader is drawn, the mid point of the shape's side is determined.

<After pressing ok button ….>
Leader Position : Select ⓐ point.
Mark Position : Select ⓑ point.
Leader Position : Select ⓒ point.
Mark Position : Select ⓓ point.
Leader Position : Press Enter key and quit.



7. Text Arrangement

This command arrange many selected texts horizontally, vertically, or in different arrangment.

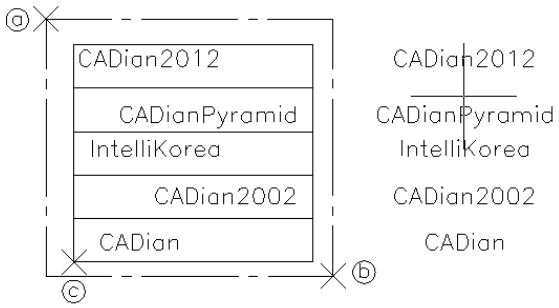
<Perpendicular Arrangement >

Command : TEXTARRANGE

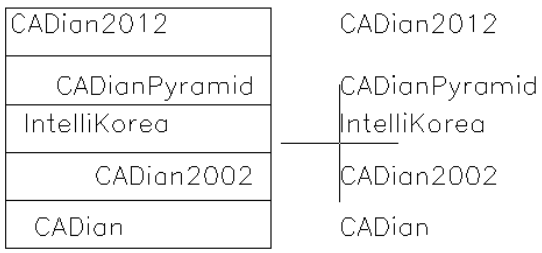
Select Text : Select the object by selecting ⓐ and ⓑ point.

Select Entity : Press Enter key.

Arrange Point : Select ⓒ point.*(Once selecting a point, Start dragging the crosshair with text arranged in the center.)*

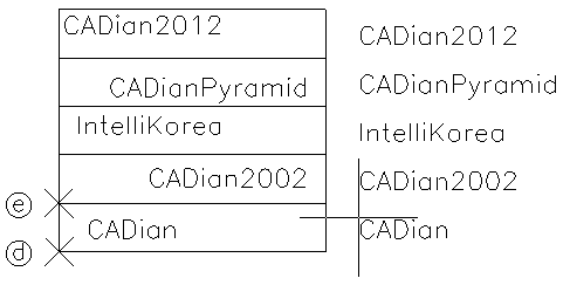


Text Arrange[Horizontal/Vertical/Spacing/Left/Center/Right/Bottom/Middle/Top /Attribute]: Enter "L". *(Start dragging the crosshair with the text arranged to the left.)*



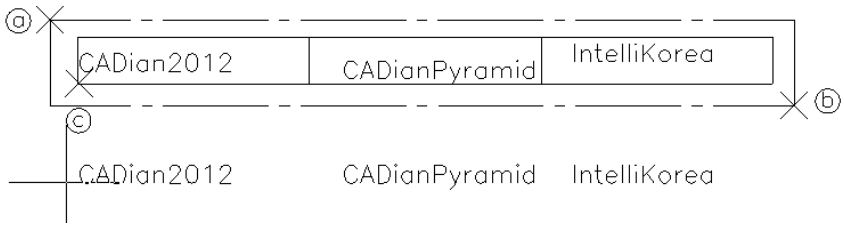
Text Arrange[Horizontal/Vertical/Spacing/Left/Center/Right/Bottom/Middle/Top /Attribute]: Enter "S".

Text Spacing : Select ⓓ and ⓔ point.*(Selected text is arranged with the same spacing as shown below.)*

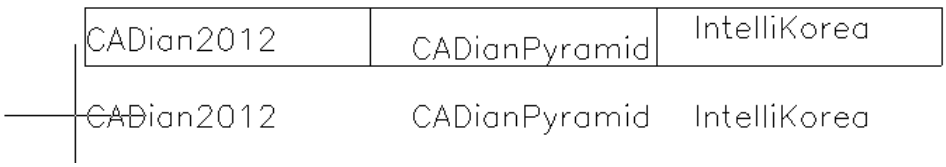


Text Arrange[Horizontal/Vertical/Spacing/Left/Center/Right/Bottom/Middle/Top /Attribute]: Enter ⓕ point.*(If dragging a mouse to ⓕ point, it automatically finds the arrangment point selected before and the text is automatically arranged.)*

<Perpendicular Arrangement>
Command : TEXTARRANGE
Select Text : Select the object by clicking ⓐ and ⓑ point.
Select Entity : Press Enter key.
Arrange Point : Select ⓒ point.*(Once selecting the point, the text is dragged in arrangement as shown below. Below is in the bottom arrangement.)*



Text Arrange[Horizontal/Vertical/Spacing/Left/Center/Right/Bottom/Middle/Top /Attribute]: Enter "M". *(Start dragging the crosshair to the center point in arrangement of the middle of the text as shown below.)*



Text Arrange[Horizontal/Vertical/Spacing/Left/Center/Right/Bottom/Middle/Top /Attribute]: Select ⓓ point.*(If selecting ⓓ point by using OSNAP<"MID">, text is arranged as shown below.)*

8. Text Copy
This command copies the text data of the original text without changing the text's attribute.

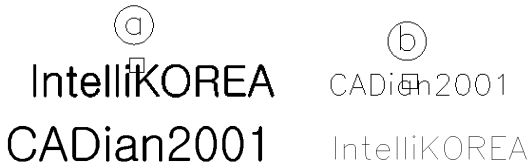
Command : TEXTCOPY
Select Origin Text : Select the original text by selecting ⓐ point.
Select Copied Text : Select the original text by selecting ⓑ point.

Select Copied Text : Press Enter key and quit the command.

9. Text Swap

This command swaps the text's data with the same text's attributes.

Command : TEXTSWAP
Select Text : Select the text by ⓐ point.
Select Changed Text : Select the text by ⓑ point.



10. Text Add

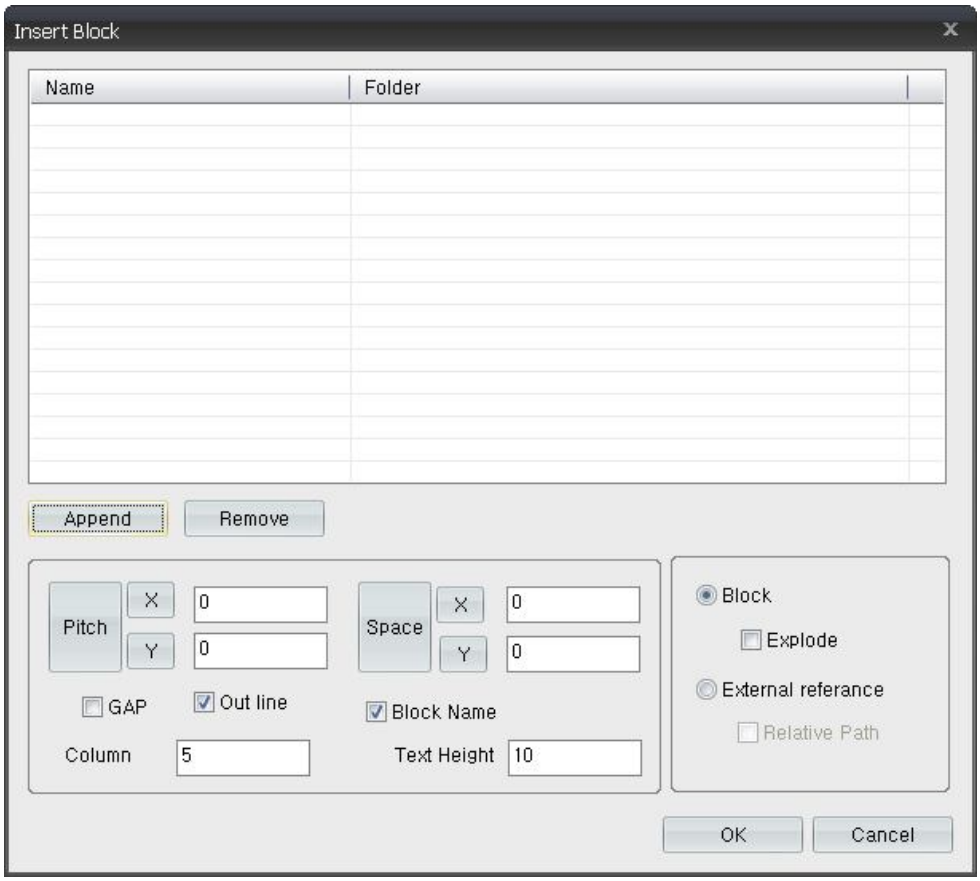
If two texts are selected, this command adds the second text to the back of the firstly added text. Added text follows the first text's attribute.

Command : TEXTADD
Select Text : Select the original text by selecting ⓐ point.
Select Entity : Press Enter key.
Select Append Text : Select the text to be added by selecting ⓑ point.

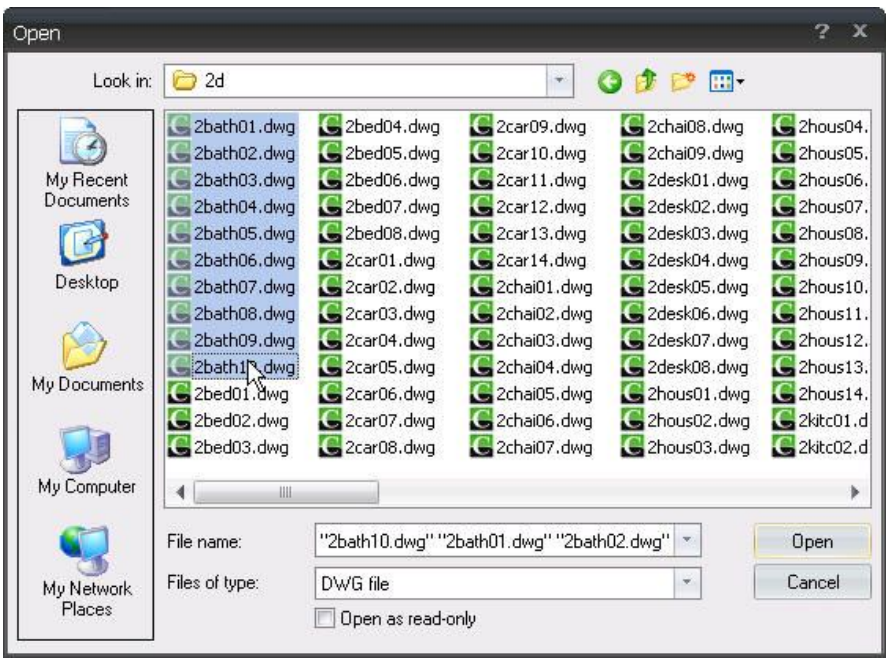
5. Block

1. Multiblock Insertion

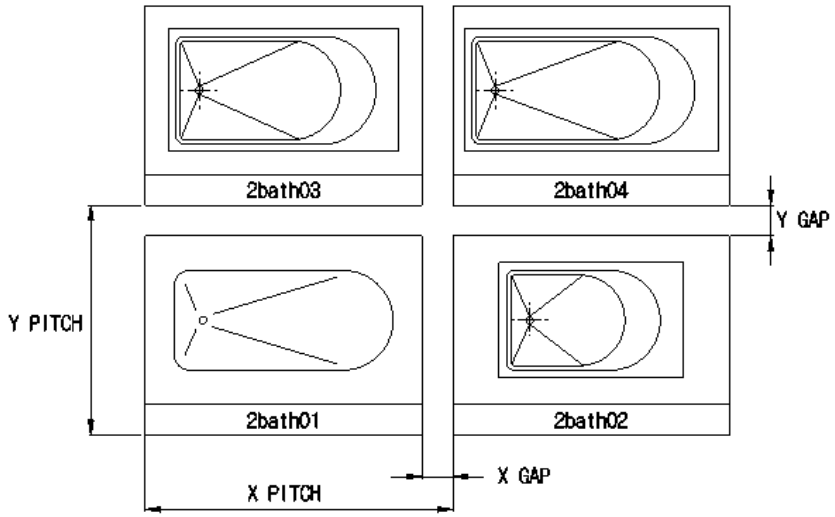
If selecting the file with block insertion repeatedly, this command inserts the block and creates the title accordingly.



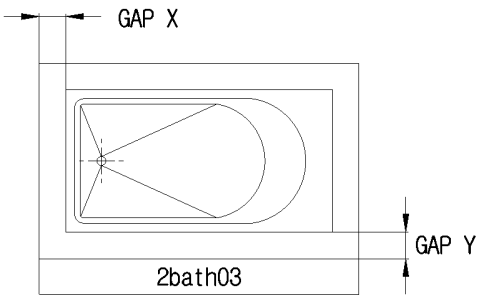
Append : If clicking on the add button, the dialog below pops up. Select the block to be inserted and click on the add button.



Remove : Deletes the selected block from the list.
Pitch : Dialog is closed when clicking the pitch button. Click two points for the rectangular box to obtain horizontal and vertical pitches.
Pitch X : Dialog is closed when X button is clicked. Click two points on the straight line to obtain the X pitch value.
Pitch Y : Dialog is closed when Y button is clicked. Click two points on the straight line to obtain the Y pitch value.
Space : Dialog is closed when clicking the space button. Click two lines for the rectangular box to obtain horizontal and vertical distances.
Space X : Dialog is closed when X button is clicked. Click two points on the straight line and obtain X distance.
Space Y : Dialog is closed when Y button is clicked. Click two points on the straight line and obtain Y distance.



GAP : Once you check in the checkbox, X and Y values of the pitch are set to the inner distance of the title as shown below.



Outline : If the checkbox is not checked, title box is not created.
Block Name : Sets whether the block name is entered in the title box.
Column : Enter the number of columns to be batched. If the number of blocks is more than the entered column number, then another line is added and creates the titlebox.
Text Height : Sets the text height of the box name.
Block : inserts the object with the block's attribute.
Explode : Once the checkbox is not checked, block is exploded and inserted into the drawing.
External reference : Inserts the block to be inserted with the external reference's attribute.

Relative Path : Inserts the block of external reference with the relative path's attribute.

Command : MINS

2. External Reference Insertion

Once selecting the external referenced block, attribute of the block is changed to the attribute of the block in the drawing.

Command : XREFINS

3. External Reference Detachment

Detaches the external referenced block from the drawing when it was selected.

Command : XREFDETACH

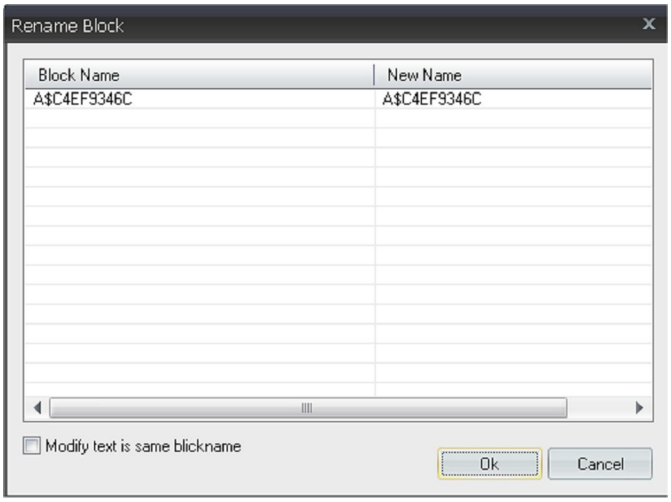
4. External Reference Unload

This function unloads when selecting the external referenced block. Loading can be performed by external reference command.

Command : XREFUNLOAD

5. Block Name Change

Once selecting the block object, dialog below pops up. Block name can be changed through the dialog.



Command : BLOCKRENAME

6. Block Origin Change

This command changes the attribute of the inserted block. Select the block first and then select the new origin of the block.

Command : BLOCKREORIGIN

7. Add Object to Block

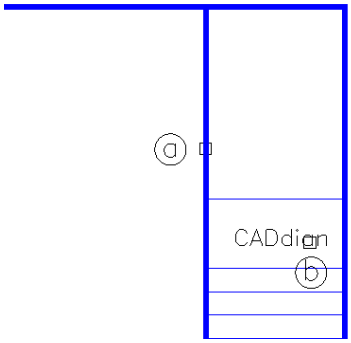
This command inserts the selected object to the inserted block.

Command : APPENDBLKENT

Select Block : Select ⓐ point to select the block.

Select Entity : Select the object by selecting ⓑ point. ("CADian" text object is added to

the title Block below.)



8. Remove Object in the Block

This command deletes the specific object in the inserted block. It has similar procedure to delete command.

Command : REMOVEBLKENT

9. Copy Object in the Block

This command copies the specific object in the inserted block. It has similar procedure to copy command.

Command : COPYBLKENT

10. Divide the multiple inserting blocks

This command explodes the inserted block using MINSERT.

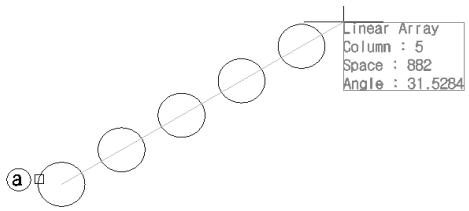
Command : MULTBLOCKEXP

6. Array

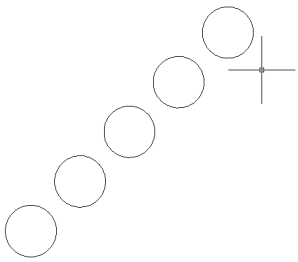
1. Linear Array

This command arranges the selected object linearly by dragging dynamically.
(Text height of the array information help is proportional to "PICKBOX" value.)

Command : LARRAY
Select Entity : Select the object by ⓐ point.
Array Column [Base] <10> : Enter "5".*(Once dragging the cursor, array distance and angle change dynamically as shown below.)*



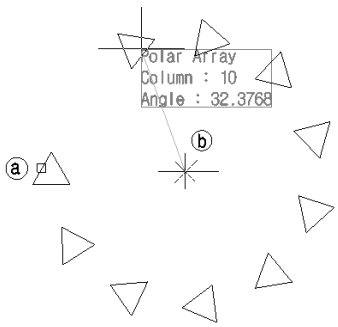
Array Space <100> : Enter "800".*(Also able to enter the distance by selecting a specific point.)*
Linear Array[Base/Column/Space/Rotate/eXit]Angle : <0.00>: Enter "45".



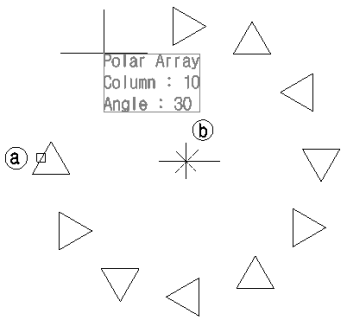
2. Circular Array

This command arranges the selected object in a circle shape.

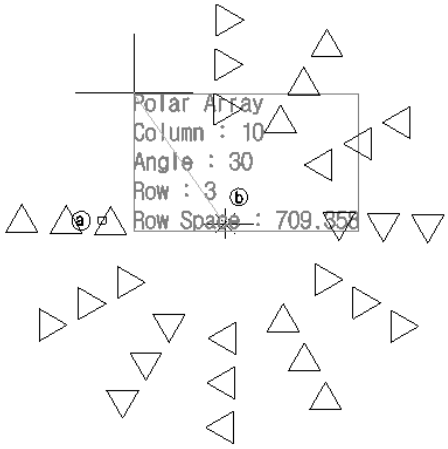
Command : PARRAY
Select Entity : Select the object by selecting ⓐ point.
Select Entity : Press Enter key.
Center Point : Select ⓑ point.
Item Column [Angle] <4> : Enter “10”.



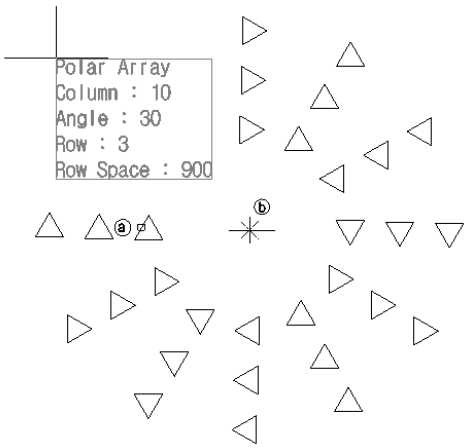
Draw Angle(+=CCW, -=CW) < 360 > : Enter “270”.



Polar Array [Base/Column/Angle/Draw angle/Row/Row Space/ROate/eXit] : Enter “R”.
Row <1> : Enter “3”.



Polar Array [Base/Column/Angle/Draw angle/Row/Row Space/ROate/eXit] : Enter “RS”.
Row Space <100> : Enter “900”.

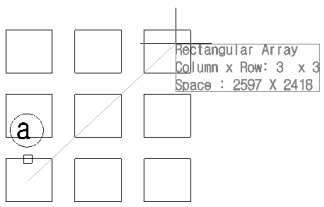


Polar Array [Base/Column/Angle/Draw angle/Row/Row Space/ROate/eXit] : Quit this command by entering “X”.

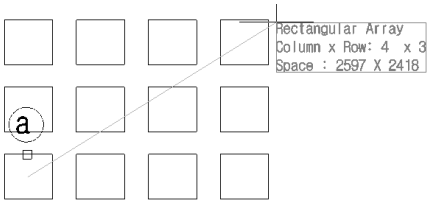
3. Rectangular Array

This command displays the object dynamically in rows and columns when dragging the cursor. Array can be performed fast and exactly when the array is displayed in the screen.

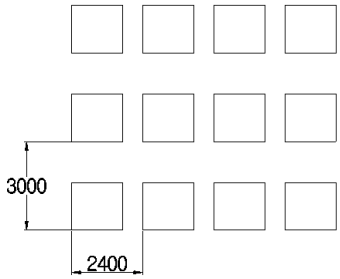
Command : **RARRAY**
Select Entity : Select the object by selecting ⓐ point.
Select Entity : Press Enter key.
Column [Angle/Column] : Enter “C”.



Column <3>: Enter “4”.
Row <1>: Enter “3”.



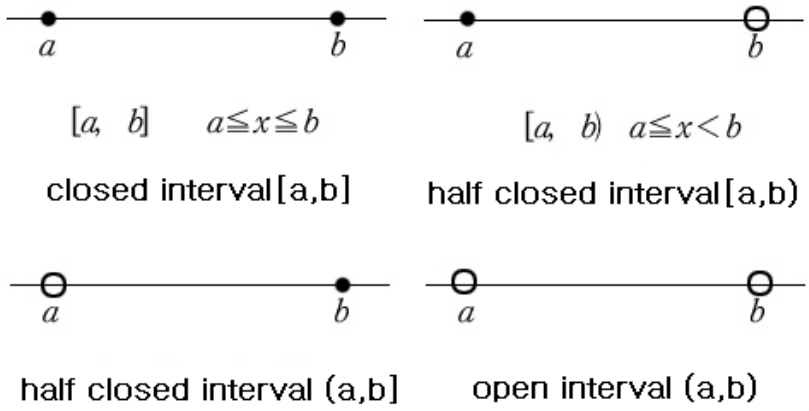
Column Space[Space] : Enter “S”.
Column Space [Length] <2418.1> : 2400
Row Space [Length)]<2418.1> : 3000



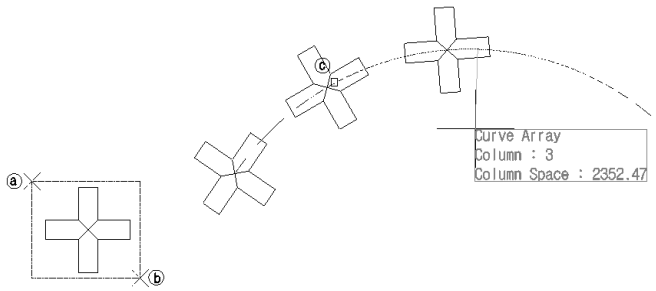
Rectangular [Column/Row/Column Space/Row Space/Angle/ROtate/eXit]<eXit> : Enter “X” or click.*(If the command is quitted forcefully, the array is not drawn properly.)*

4. Array the Curve line

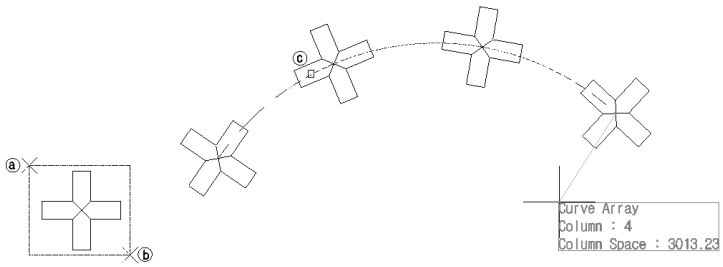
This command arrays the object in the selected curve dynamically.



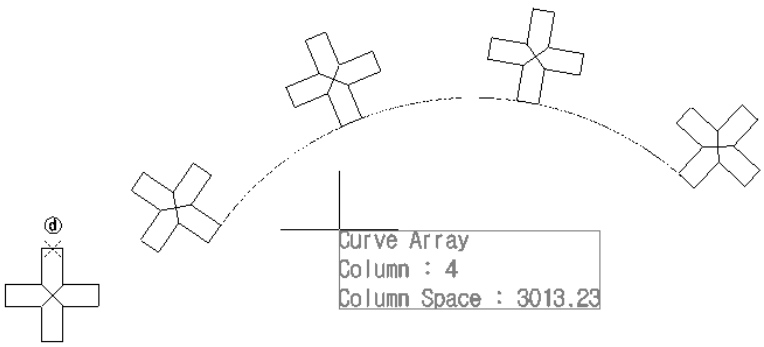
<Open Interval, The looped curve>
Command : CARRAY
Select Entity : Select the object by selecting ⓐ and ⓑ point
Select Entity : Press Enter key.
Select Curve : Select the curve by selecting ⓒ point.*(The object is automatically arranged starting from the curve start point with the default value once the curve is selected. The center point of the object is the base point. In order to change, enter "B" and change the base point.)*



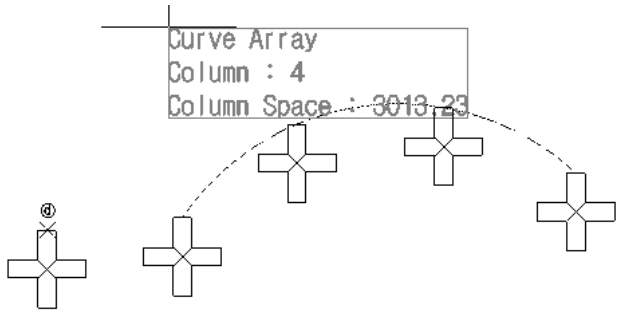
Column [Base/Start Pos)ROtate] <10> : Enter "4".*(Four objects are arranged dynamically.)*



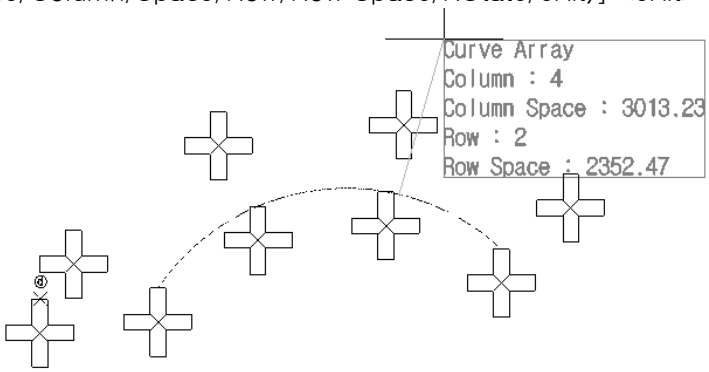
Column Space[Full] <3000> : Enter "F".(*Objects are arranged at points where the selected curve is divided into four section. If the command is not quitted, array command is not fully done. Image above is drawn virtually.)*
Curve Array [Base/Column/Space/Row/Row Space/ROtate/eXit)] <eXit> : Enter "B".
Base Point : Select ⓓ point.*(Depends on the starting direction of the curve, the base point can be flipped over.)*



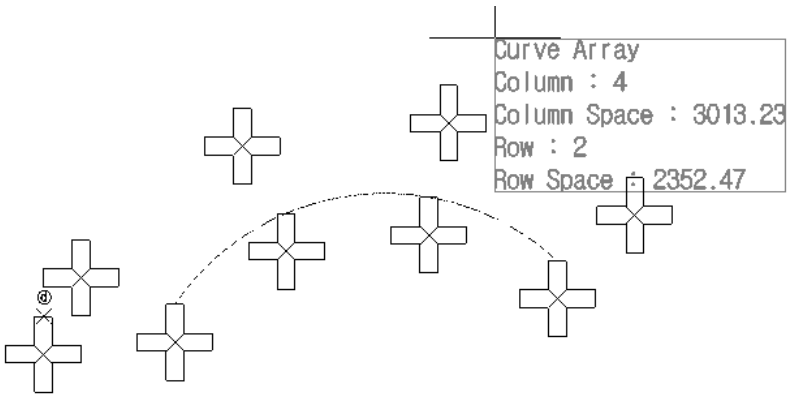
Curve Array [Base/Column/Space/Row/Row Space/ROtate/eXit)] <eXit> : Enter "ROT".



Curve Array [Base/Column/Space/Row/Row Space/ROtate/eXit)] <eXit> : Enter “R”.



Row (1) : Enter “2”.



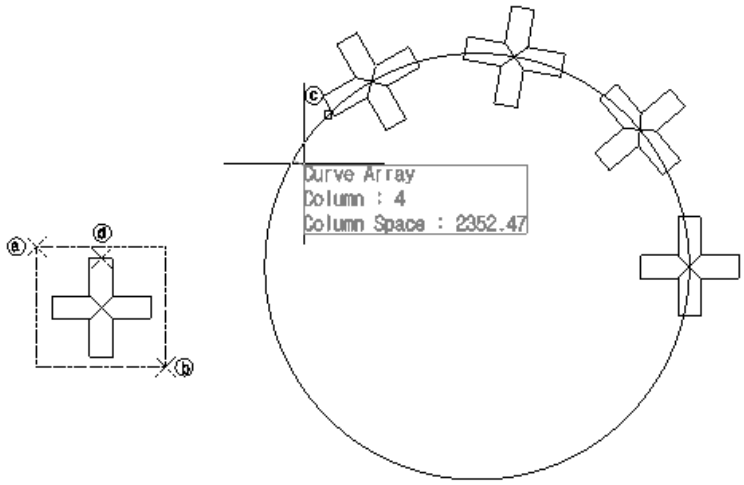
Curve Array [Base/Column/Space/Row/Row Space/ROtate/eXit)] <eXit> : Enter “X” and quit the command.*(Draws the arranged object when quitting the command. If the command is quitted forcefully by pressing “Esc” key, then virtual objects are deleted and array command is cancelled.)*

<Close Interval, The looped line>

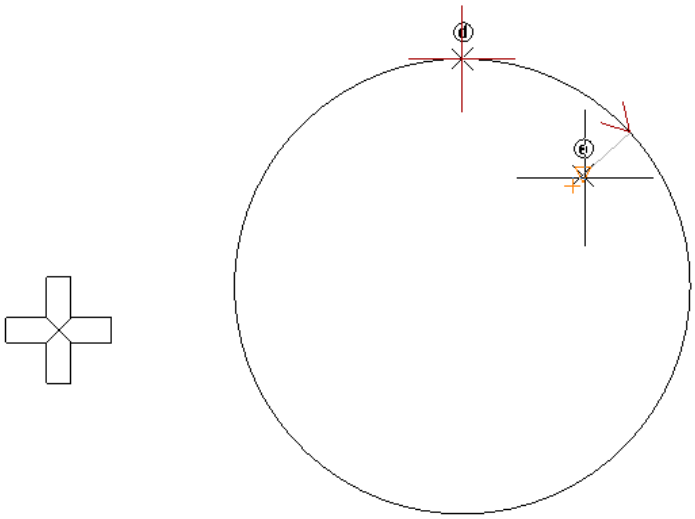
Command : CARRAY

Select Entity : Select the object by selecting ⓐ and ⓑ points

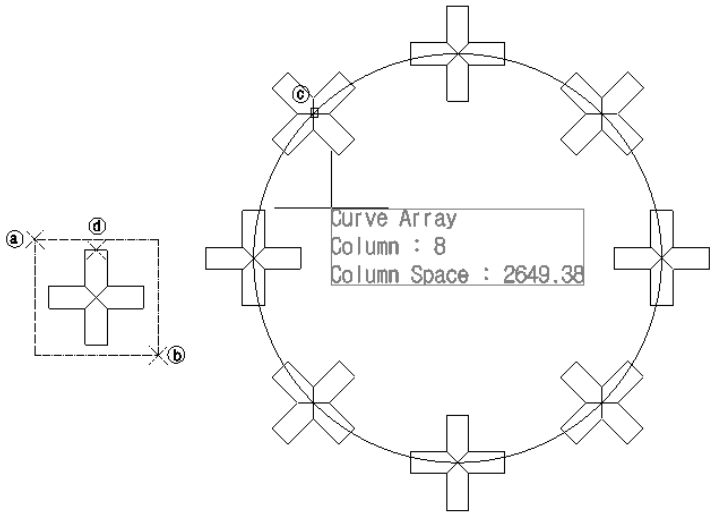
Select Entity : Press Enter key.*(For circles, start point is set to “0” degree. In order to change the start point, enter “SP” and change the start point.)*



Column [Base/Start Pos)/ROtate] <10> : Enter “SP”.
Start Pos : Select ⓓ point.
Array Direction : Select ⓔ point.*(If the crosshair is brought to the opposite direction, the direction of the arrow is changed to the opposite direction.)*



Column [Base/Start Pos/ROtate] <10> : Enter “8”.
Column Space[Full] <2352.47> : Enter “F”.

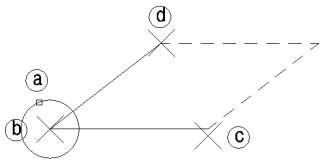


Curve Array [Base/Column/Space/Row/Row Space/ROtate/eXit] <eXit> : Enter “X” and quit the command.

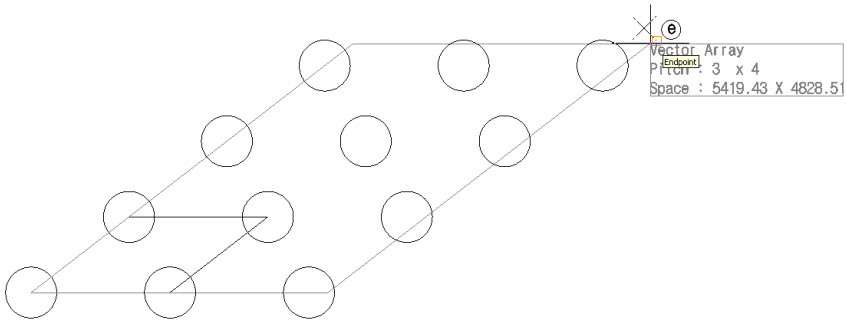
5. Vector Array

This command arrays the selected object with rhombus shape in rows and columns.

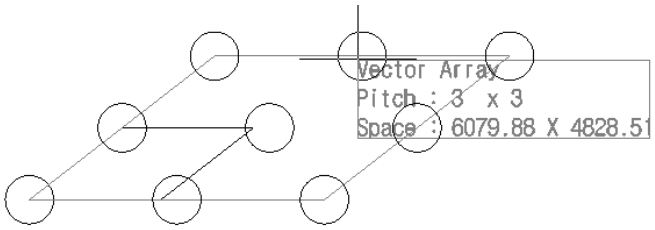
Command : VARRAY
Select Entity : Select the object by selecting ⓐ point.
Select Entity : Press Enter key.
Base Point : Select ⓑ point.
X Axis Vector : Select ⓒ point.
Y Axis Vector : Select ⓓ point.



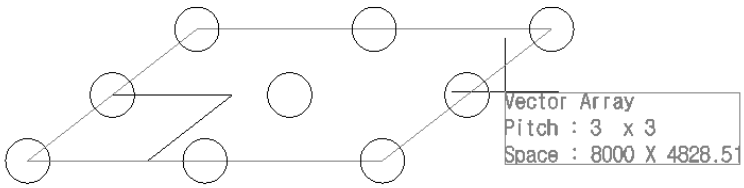
Array Position : Select ⓔ point.*(If moving the crosshair, objects are arranged as shown below in a rhombus shape. Row distance is set to the distance between ⓑ point and ⓒ point. Column distance is set to the distance between ⓑ and ⓓ point.)*



Vector Array [Column/Row/Column Space/Row Space/eXit] : Enter “R”.
Row Space<4>: Enter “3”.



Vector Array [Column/Row/Column Space/Row Space/eXit] : Enter “CS”.
Column Space [Length] <5000> : Enter “30”.



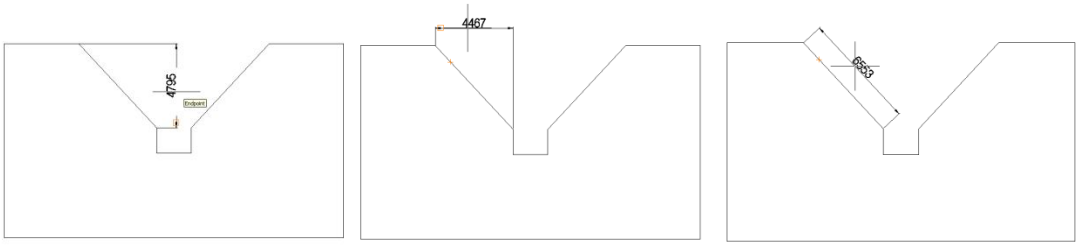
Vector Array [Column/Row/Column Space/Row Space/eXit] : Enter “X” and quit the command.

7 Dimension

1. Linear Dimension

   Click on two points where the dimensions should be created, and move the cursor as shown below, then dimension is automatically changed horizontal, vertical, and slanted way. Once the preferred dimension is displayed, dimension is drawn on the selected point.

   Command : LDIM



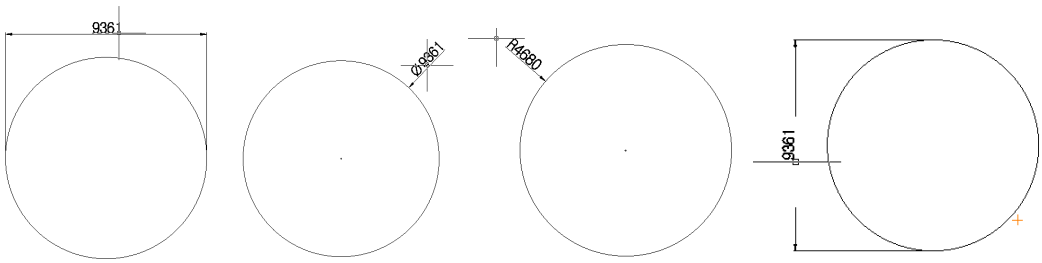
2. Circular Dimension

   If a circle or arc is selected, line weight, diameter, angle values are determined e depending on the crosshair’s location or option setting.

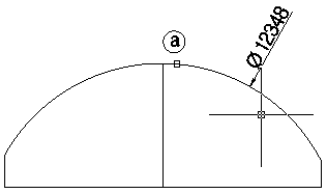
   <When a circle is selected >

   Dimension shape is determined according to the crosshair’s location as shown below. Diameter and radius are automatically changed when “R” or “D” keyword is entered.

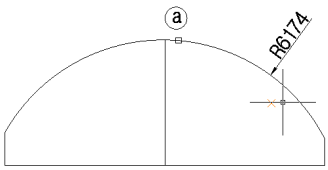
   Command : CDIM



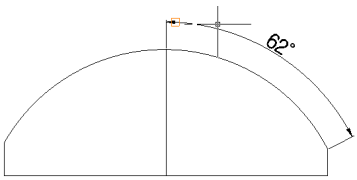
<When an arc is selected >
Command : CDIM
Select arc or circle : Select an arc by selecting ⓐ point.



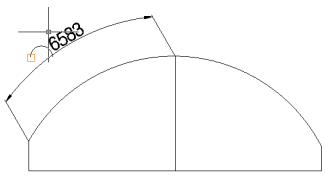
Specify dimension line location[Diameter/Radius/Angular/arcLeng/Chordleng/eXit]:
Enter "R".*(Dimension is changed to the radius value as shown below.)*



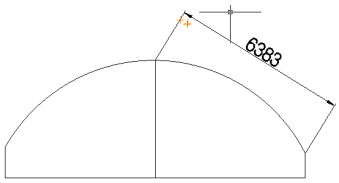
Specify dimension line location[Diameter/Radius/Angular/arcLeng/Chordleng/eXit]:
Enter "A".*(Dimension is changed to angle as shown below. If there is a intersection object in the arc, intersection point is found automatically and the angle dimension line is displayed.)*



Specify dimension line location[Diameter/Radius/Angular/arcLeng/Chordleng/eXit]:
Enter "L".*(Arc length is displayed automatically on the point where the crosshair is.)*



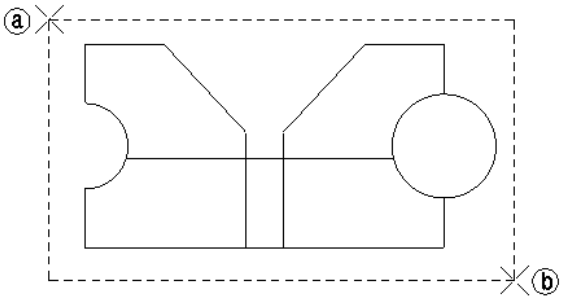
Specify dimension line location[Diameter/Radius/Angular/arcLeng/Chordleng/eXit]:
Enter "C".(*Chord length is displayed automatically on the point where the crosshair is)*



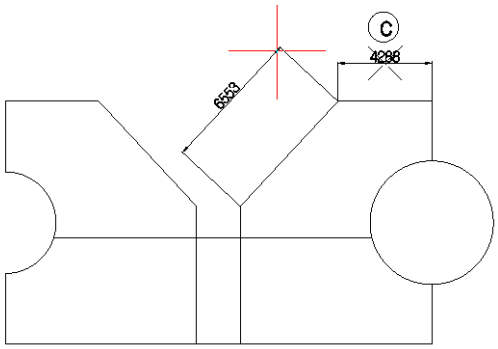
3. Object Dimension

This command finds the object where the crosshair is and enters the dimension automatically when objects are all selected at once.

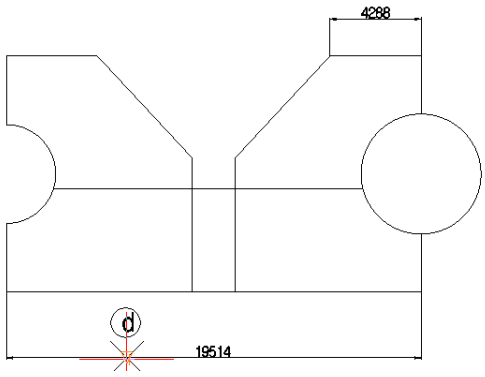
Command : **OBJDIM**
Select Entity : Select the object by selecting ⓐ and ⓑ points.



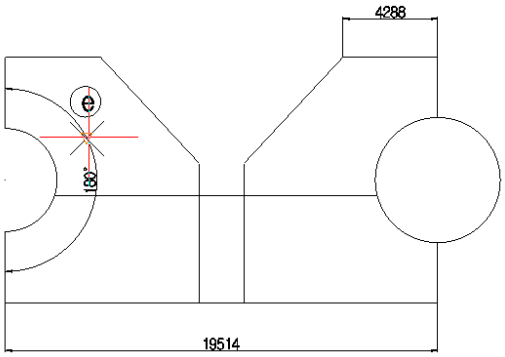
Specify dimension line location [Horizontal/Vertical/aliGned/Diameter/Radius/Angular /arcLeng /Chordleng/Intersect/Pick/eXit] <aliGned,Diameter> : Select ⓒ point and draws the dimension.*(After the dimension is created, the next virtual dimension is created where the cursor location is.)*



Specify dimension line location [Horizontal/Vertical/aliGned/Diameter/Radius/Angular /arcLeng /Chordleng/Intersect/Pick/eXit] <aliGned,Diameter> : Enter "I".*(A dimension is created up to the point where the object is automatically intersected. If "I" option is applied, the intersection point is ignored and entire dimension can be created.)*
Specify dimension line location [Horizontal/Vertical/aliGned/Diameter/Radius/Angular /arcLeng /Chordleng/Intersect/Pick/eXit] <aliGned,Diameter> : Select ⓓ point to draw the dimension.*(after the dimension below is created, in order to create the dimension between the dimension, enter "I". Then the function changes to intersection option and the dimension is automatically drawn.)*



Specify dimension line location [Horizontal/Vertical/aliGned/Diameter/Radius/Angular /arcLeng /Chordleng/Intersect/Pick/eXit] <aliGned,Diameter> : Enter "A". *(If the crosshair is moved to the object, angle dimension can added automatically.)*
Specify dimension line location [Horizontal/Vertical/aliGned/Diameter/Radius/Angular /arcLeng /Chordleng/Intersect/Pick/eXit] <aliGned,Angular> :After selecting ⓔ point, angle dimension line is created.



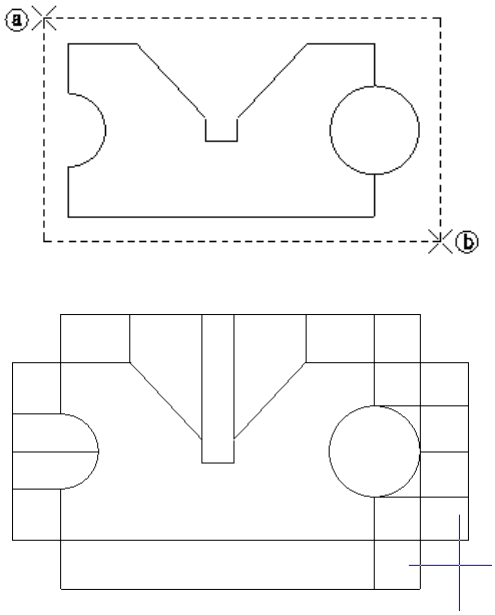
Specify dimension line location [Horizontal/Vertical/aliGned/Diameter/Radius/Angular /arcLeng /Chordleng/Intersect/Pick/eXit] <aliGned, Angular > : Enter "X" and quit the command.

"As mentioned before, many different dimension styles can be applied if each function keys are applied during the command"

4. Outline Dimension

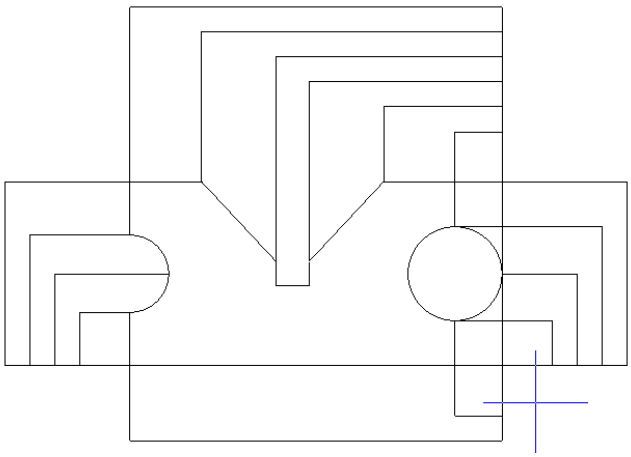
This command finds the outline of the selected object and creates the dimension. Use the crosshair to draw the preferred dimension and once it is drawn, click on the place and the dimension will be created. This function can be applied in real-time, so while the dimension is virtually drawn, it can be applied accordingly. Dimension line is drawn with the current dimension style.

Command : **OUTDIM**
Select Entity : Select the object by selecting ⓐ and ⓑ points.

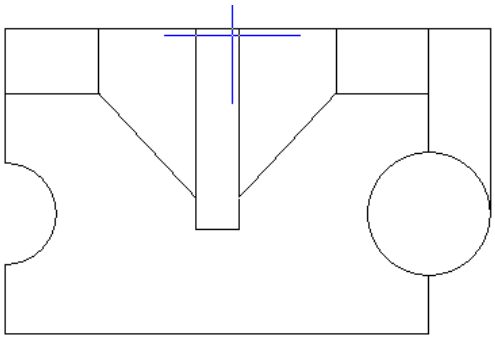


<Once the object is selected, outline dimension is drawn virtually as shown above. If the crosshair location is changed, the dimension's location can be determined. Once the final location is selected, dimension is created.>

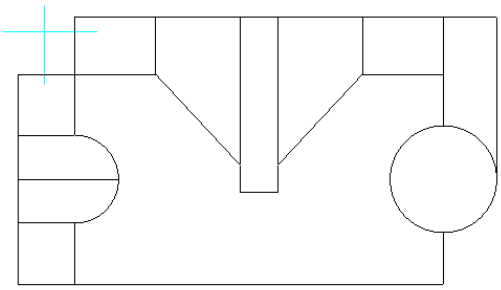
Specify dimension line location[Continuous/Zigzag)/Baseline/Stack/Ordinate/oriGin)/All /sIngle/Double/eXit] : Enter "B".*(Based on the crosshair location, dimension creation method differs automatically if the dimension on base line is created. If the point is not selected to the preferred location, then the dimension is not created)*



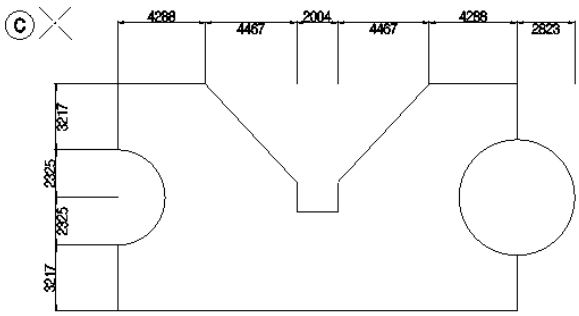
Specify dimension line location[Continuous/Zigzag)/Baseline/Stack/Ordinate/oriGin)/All /sIngle/Double/eXit] : Enter "I".(*dimension is virtually created in the direction of the crosshair as shown below.)*



Specify dimension line location[Continuous/Zigzag)/Baseline/Stack/Ordinate/oriGin)/All /sIngle/Double/eXit] : Enter "D".*(Double dimension is virtually created in the direction of the crosshair as shown below.)*



Specify dimension line location[Continuous/Zigzag)/Baseline/Stack/Ordinate/oriGin)/All /sIngle/Double/eXit] : Enter "G".*(Extension line's origin is sorted as shown below)*
Specify dimension line location[Continuous/Zigzag)/Baseline/Stack/Ordinate/oriGin)/All /sIngle/Double/eXit] : Enter "Z".*(Continuous dimension text is created in zig zag pattern. This function can be useful when the dimension distance is too dense.)*
Specify dimension line location[Continuous/Zigzag)/Baseline/Stack/Ordinate/oriGin)/All /sIngle/Double/eXit] : Once ⓒ point is clicked, dimension line is created and command is closed.

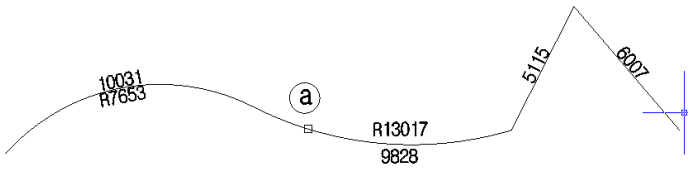


5. Display Curve Length

This function draws the lengths and radius automatically when selecting a curve(polyline, spline, ellipse) (Current text height style is applied to the text height (Text height X dimension scale "DIMSCALE").)

<When selecting a polyline >
Command : CURVELEN

Length,Radius [All/LEngth/Radius/Intersect] : Select a polyline object by selecting ⓐ point.

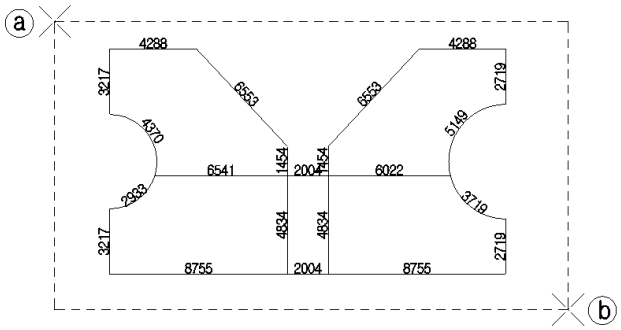


<When selecting many intersecting objects >

Command : CURVELEN

Length,Radius [All/LEngth/Radius/Intersect] : Enter "I".

Intersect [All/Length/Radius/Intersect] : Select the object by selecting ⓐ and ⓑ points. *(Dimensions are automatically created for all the intersected lines in the object.)*



If there is any inquiry, please contact CADian Helpdesk by phone or by email :

. Phone : +82 (70) 8240-4291 / Mail help@cadian.com

This manual is subjected to change without any prior notice for function and quality improvement.