

TOOLS INSTRUCTIONS

LINES: 

IMPORTANT NOTE:

When drawing an entity with a tool icon you can change snap between each click required to complete the entity

2 Points:

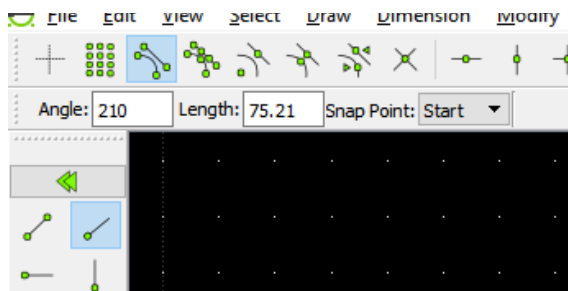


When selected it will prop the tool options: Close, Undo, Redo. Draws a line from a first point to a second point. If one draws more than two lines, clicking on Close will automatically close the entity from the last drawn point to the first.

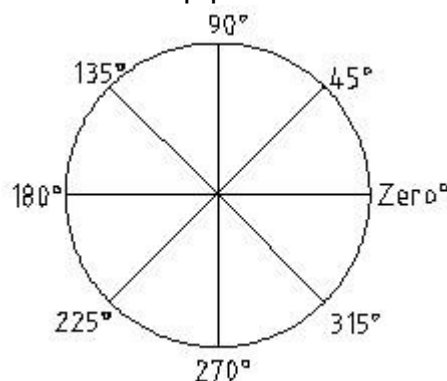
Angle:



When selected it will prop the tool options: Angle, Length, Snap Point:



It will draw a line from the snap point chosen at the angle and the length set.



The example above shows 8 lines drawn with "Snap Center", length equal to radius and the angles as indicated.

Horizontal: 

When selected it will prop the tool options: Length, Snap Point. It will draw a horizontal line from the snap point chosen at the length set.

Vertical:

When selected it will prop the tool options: Length, Snap Point. It will draw a vertical line from the snap point chosen at the length set.

Rectangle:

Draws a rectangle or a square from two opposite points: the lower left corner and the upper right corner, or vice and versa. The sides will be parallel to the X and Y axes. The entity created is a polyline, see # Polyline below.

If one needs to modify it to 4 lines, the Modify Explode (see Modify below) tool will do it.

Parallel:

When selected it will prop the tool options: Distance, Number. It will draw the set number of lines parallel to any line on the drawing at the given distance when the mouse is moved over said line, left click when the parallels are located where one wants them to be. It will also work with circles and arcs, not with ellipses or polylines.

Parallel through point:

When selected it will prop the tool options: Number. It will draw the set number of lines parallel to any line on the drawing, the first one going through a given point depending on the Snap option, left click when the parallels are located where one wants them to be. It will also work with circles and arcs.

Bisector:

When selected it will prop the tool options: Length, Number. It will draws the bisector(s) of two crossing lines. Whether or not the cross point of the original lines is visible, the bisector will start from it. With n number, it will draw n new lines dividing the angle in n+1 equal angles.

Tangent (P,C):

Draws a line tangent to an existing circle, arc or ellipse from a chosen point.

Tangent (C,C):

Draws a line tangent to two existing circles, arcs or ellipses or a combination of them.

Tangent Orthogonal:

Draws a line square to another one and tangent to a circle, arc or ellipse.

Orthogonal:



When selected it will prop the tool option: Length. It will draw a line square to another one, the length will be equal to the value inserted either side of the crosshairs, as a result the new line will be twice as long as defined.

It can also be used with a circle, in this case it will draw a line square to a radius.

It can also be used with an arc, in this case it will draw a line square to the arc itself.

Relative angle:



When selected it will prop the tool option: Angle, Length. It will draw a line at the given angle to another line, the length will be equal to the value inserted either side of the crosshairs, as a result the new line will be twice as long as defined.

It can also be used with a circle, in this case it will draw a line at given angle to a radius.

It can also be used with an arc, in this case it will draw a line at given angle to the arc itself.

Polygon (Cen, Cor):



When selected it will prop the tool option: Number. It will draw a polygon with the given number of sides, defining center and one corner.

Polygon (Cor, Cor):



When selected it will prop the tool option: Number. It will draw a polygon with the given number of sides, defining two corners.

Freehand Line:



It will draw a freehand line so long as one keeps the left mouse button down.

ARCS:



Center, Point, Angles:



When selected it will prop the tool option: Anticlockwise, Clockwise. First select the center, then the radius, the next click will determine the starting point of the arc and the last click the end of the arc.

3 Points:



It will draw an arc subtended by three points.

Concentric:



It will draw a concentric arc to one previously drawn. The distance by default is 1 unit, to change the value requires typing the new value in the Command Line. Moving the mouse close to the existing arc will show the new arc either inside or outside the original.

NOTE: on later version of LC this icon no longer appears, use the Line Parallel icon instead or Modify Offset one.

Arc Tangential:



When selected it will prop the tool option: Radius, Angle. It will draw an arc tangent to an extremity of an existing line or arc with the set radius or angle (the central angle is equal to the value checked in the angle value). At the moment of writing it seems that the use of it is unstable.

CIRCLES:



Center, Point:



It will draw a circle from the position of the center and the radius given by the position of the mouse. The value of the radius can also be typed in the Command Line.

Center, Radius/



When selected it will prop the tool option: Radius. It will draw a circle with the set value of the radius from the selected position of the center.

2 Points:



It will draw a circle the diameter of which is determined by the two points chosen.

2 Points, Radius:



When selected it will prop the tool option: Radius. It will draw a circle with the set value of the radius, passing through the two points. If the set diameter (radius x 2) is smaller than the distance between the points no circle will be drawn.

3 Points:



It will draw a circle passing through the three points.

Concentric:



It will draw a concentric circle to one previously drawn. The distance by default is 1 unit, to change the value requires typing the new value in the Command Line. Moving the mouse close to the existing circle will show the new circle either inside or outside the original.

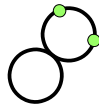
NOTE: on later version of LC this icon no longer appears, use the Line Parallel icon instead or Modify Offset one.

Circle Inscribed:



It will draw a circle inscribed in a triangle. Note that the sides of the triangle do not necessarily need to meet. In case the sides are too short, use "Draw tangential circle of 3 given points, lines, arcs/circles" described below.

Tangential, 2 Points:



It will draw a circle tangential to a line, a circle or an arc and passing through two points.

Tangential 2 Circles, Radius:



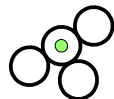
When selected it will pop the tool option: Radius. It will draw a circle of the set radius tangent to 2 other circles. If the set diameter (radius x 2) is smaller than the distance between the 2 circles, no circle will be drawn.

Tangential 2 Circles, 1 Point:



It will draw a circle tangent to 2 other circles and passing through a point.

Tangential 3 circles:



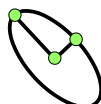
Draw tangential circle of 3 given points, lines, arcs/circles:

It will draw a circle tangent to a combination of 3 of any points, lines arcs/circles.

ELLIPSES:



Ellipse (Axis):



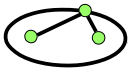
It will draw an ellipse starting with the center then the long axis ending with the short one.

Ellipse Arc (Axis):

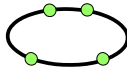


It will draw an ellipse starting with the center then the long axis ending with the short one. Once the ellipse is in place, setting the arc will require 2 more clicks.

Ellipse Foci Point:

It will draw an ellipse  with its two foci and one point on the ellipse itself.

Ellipse 4 Point:



It will draw an ellipse going through 4 points.

Ellipse Inscribed:



It will draw an ellipse inscribed in a quadrilateral.

Points:



This tool will just draw a point.

NOTE: The point drawn will have the size defined in the layer and could be uneasy to see. It is advisable to enlarge it through Properties or Attributes, this will not affect the actual position of the point should you draw from, to or through it, only its center will be considered.

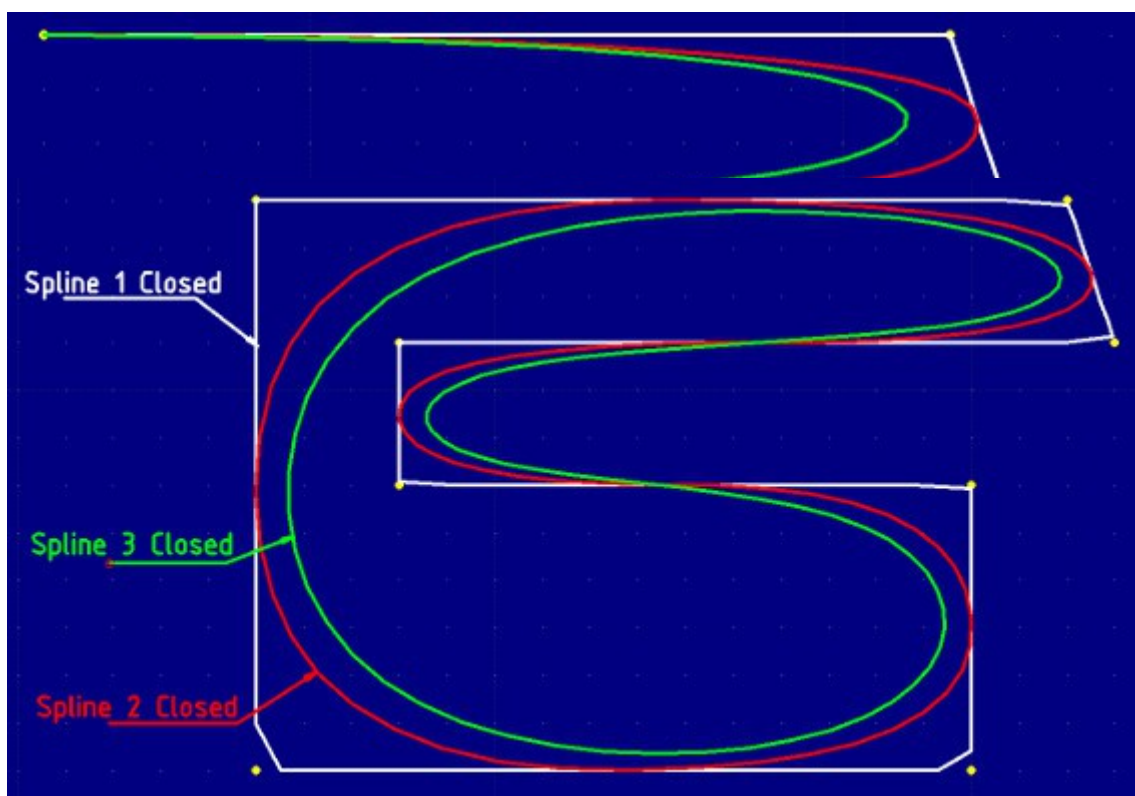
SPLINES:



Spline:



When selected it will prop the tool options: Degree:, Closed, Undo. It will draw a spline the curvature of which will be influenced by the Degree chosen. Checking the Closed checkbox will draw a closed spline. Each click on Undo will cancel the last segment created, eventually back to the starting point. The illustrations below show the 3 degrees of open splines and the 3 degrees of closed splines produced using the same nodes in yellow.

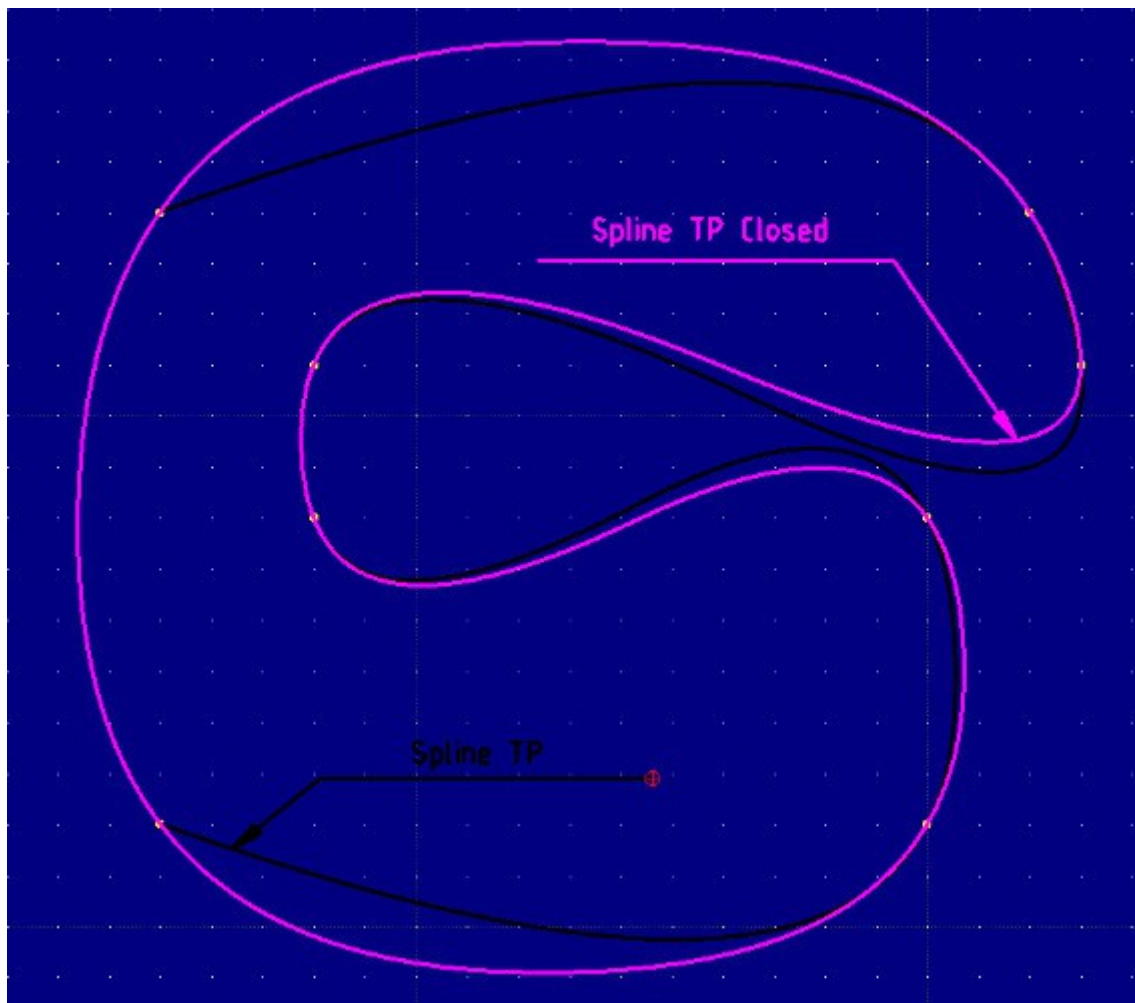


NOTE: most of times, a degree 1 spline will show beveled angles as seen on the Spline 1 Closed.

Spline through points:



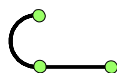
When selected it will prop the tool options: Closed, Undo. It will draw a spline going through the given points. Checking the Closed checkbox will draw a closed spline. Each click on Undo will cancel the last segment created, eventually back to the starting point. The illustration below show an open spline and a closed spline produced using the same nuds as above in yellow (TP stands for Through Pints).



POLYLINES:



Polyline:



When selected it will prop the tool options: Close, Undo, then a choice between: Line or Tangential or Tan Radius or Angle.

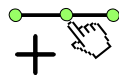
Line: this will draw a broken line going through the points chosen. It is a better option than Spline 1 as it does not create beveled angles.

Tangential: this will draw a line with a tangential arc at the end, more arcs can be added that will be tangential to the previous one.

Tan Radius: here the radius must be entered in the appropriate checker box. As above it will draw a line with a tangential arc the radius of which will be fixed and more arcs can be added and the radius changed at will.

Angle: here the radius must be entered in the appropriate checker box as well as the orientation. This will draw an arc for which the central angle is equal to the value checked in the angle value.

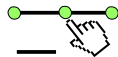
Add node:



This will add a node to an existing polyline. The resulting effect varies with the type of curve/line it applies to:

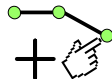
- if it is added to an arc it will transform it into two broken lines,
- if it is added to a line there will be no immediate effect but once the polyline is selected the node can be moved about just like any other.

Delete node:



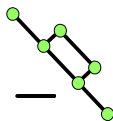
This will delete a node from an existing polyline. In order to better visualize where nodes are it is better to first select the polyline so it turns into a dotted line with the nodes visible. Snap Intersections and Endpoints maybe required to best select the nodes.

Append node:



It will allow to add a node at the beginning or the end of a polyline. The new segment created will be straight. More than one segment can be added. If the polyline is closed, it will add a new segment to the closing point of the polyline.

Delete between two nodes:



It will delete nodes located between two nodes to be selected. It will have no effect if two adjacent nodes are selected.

Trim segments:



This tool will extend two segments to their crossing point deleting all other elements situated between their two closest ends.

Create Equidistant Polyline:



When selected it will prop the tool options: Spacing, Copies. It will replicate an existing

polyline spaced with the set value and with as many copies as imposed. Left clicking close to a polyline will create the duplicate(s) on the side where one clicks.

Create Polyline from Existing Segments:



This tool allows to make a single polyline out of several connected segments, lines, arcs or polylines.

M TEXT:

M Text:

M Text stands for Multi-line Text.

This opens a dialogue box with the controls for the text to be inserted into the drawing.

Dimensions:

Aligned:

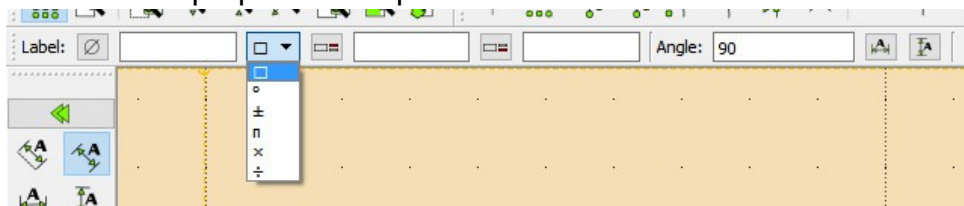


This will draw a dimension parallel to its origins, i.e., used on an oblique line it will give its linear length.

Linear:



When selected it will prop the tool options:



Label: the first case provide 6 choices, the rectangle is actually the Ø sign, the others are explicit. The second case will write the text in superscript. The third case will write the text in subscript.

Angle: this will determine the way the dimension is positioned, one can choose Vertical or Horizontal from the icons on the right or impose another angle in the Angle case. If no label is chosen, the value will be calculated according to the Angle chosen.

Horizontal:

This is the same as above having chosen the Horizontal icon.

Vertical:



This is the same as above having chosen the Vertical icon.

Radial:



This will draw the radius dimension with the center of the circle or arc as origin. The dimension created can be positioned at any chosen angle.

Diametric:



This will draw the diameter dimension inside the circle or arc. The dimension created can be positioned at any chosen angle.

Angular:



This will draw the angle dimension formed by two lines. The dimension created can be positioned at will. It will not work for parallel lines.

Leader:



This will draw a leader on any chosen entity. To write a text on the Leader one will need to use the Text or MText functions.

SELECT:



Deselect all:



Clicking on this icon will instantly deselect any selected entities.

Select All:



Clicking on this icon will instantly select all entities.

Select Entity:



Allows to select any chosen entity.

NOTE: this can also be done before selecting any icon when the mouse is represented by an arrow (or whatever sign chosen in your OS).

(De-) Select Contour:



This icon allows to select contours made up of lines, arcs and ellipse arcs. It will not select polylines or splines.

Once selected it can also be used to deselect the same contours.

Select Window:



This icon will ask to draw a window defined by its two opposite corners that will select any entity totally included in it. Any entity only partially included will not be selected.

Deselect Window:



This icon will ask to draw a window defined by its two opposite corners that will deselect any entity totally included in it. Any entity only partially included will not be deselected.

Select Intersected Entities:



This will let one draw a line that will select all entities it crosses. It can be used repeatedly so long as the right mouse button is not depressed.

Deselect Intersected Entities:



This will let one draw a line that will deselect all entities it crosses. It can be used repeatedly so long as the right mouse button is not depressed.

(De-) Select Layer:



This will permit to select, or deselect, all entities of a layer when doing a right click over or close to any entity of that layer.

Invert Selection:



A click on this icon will immediately invert the selection: if nothing is selected it will select everything (similar to Select All), if something is selected it will deselect it and select the rest.

HATCH:



One click on this icon will prompt the Select tools in order to select the item(s) to be hatched. Once done, one can either depress the Enter key or click on the double arrow below the Select tools, this will prompt the Choose Hatch Attribute window from which to choose said attributes.

NOTE: only polygons, circles, ellipses and closed polylines can be hatches, splines and non-closed entities cannot be hatched.

INSERT IMAGE:



One click on this icon will prompt the file explorer in order to select the image to be inserted.

MODIFY:



Move / Copy:



This will permit to move and/or copy selected entities. Once selected it will prop the Select icons so one can select the desired entities, once done either press Enter or click on the Continue arrows below the Select icons. Choose a reference point (left click on the chosen point), move it to the new position and left click, this will prop the Move/Copy Options window giving the following options:

- Delete Original,
- Keep Original,
- Multiples Copies: will copy the number of entities desired equidistant from one another, keeping the original.

One will also need to choose between "Use current attributes" and/or "Use current layer". In the first case the copies will be created on the same layer as the original, in the second case, and providing another layer was selected beforehand, the copies will be created in the selected layer. In case both are ticked, "Use current layer" will prevail.

NOTE ABOUT ROTATIONS: note and remember that nominal rotations are anticlockwise. For instance if one rotates an entity 90°, it will be moved anticlockwise, for a 90° clockwise rotation one can either note "-90" or "270".



Rotate:

This will permit to rotate selected entities. Once selected it will prop the Select icons so one can select the desired entities, once done either press Enter or click on the Continue arrows below the Select icons. Choose the rotation center (left click on a specific point of the entities), then choose a reference point (left click on the chosen point), move it to the new position and left click, this will prop the Rotation Options window (in case the rotation center and reference point are the same, the "Rotation Options" window will prop as soon as the reference point is positioned) giving the following options:

- Delete Original,
- Keep Original,
- Multiples Copies: will copy the number of entities desired equidistant from one another, keeping the original.

The Angle can be kept as is or altered at will.

One will also need to choose between "Use current attributes" and/or "Use current layer". In the first case the copies will be created on the same layer as the original, in the second case, and providing another layer was selected beforehand, the copies will be created in the selected layer. In case both are ticked, "Use current layer" will prevail.

Scale:



This will permit to scale, down or up, selected entities. Once selected it will prop the Select icons so one can select the desired entities, once done either press Enter or click on the Continue arrows below the Select icons. Choose a reference point (left

click on the chosen point), this will prop the "Scaling Options" window giving the following options:

- Delete Original,
- Keep Original,
- Multiples Copies: will copy the number of entities desired using the scaling factor in a cascade.

One will need to tick, or not, "Isotropic Scaling". This will determine if the scaling X Y is identical (Isotropic) or different. If not Isotropic, both X and Y factors must be entered. One will also need to choose between "Use current attributes" and/or "Use current layer". In the first case the copies will be created on the same layer as the original, in the second case, and providing another layer was selected beforehand, the copies will be created in the selected layer. In case both are ticked, "Use current layer" will prevail.

Mirror:



This will permit to mirror entities. Once selected it will prop the Select icons so one can select the desired entities, once done either press Enter or click on the Continue arrows below the Select icons.

The command line will ask to choose the first point of the mirror line, left click where desired, the command will then ask for the second point of the line, at this point one will be able to visualize the position of the new entity, left click when the position is correct.

Move and Rotate:



This will permit to move/copy and rotate entities. Once selected it will prop the Select icons so one can select the desired entities, once done either press Enter or click on the Continue arrows below the Select icons. This will prop the tool option "Angle:" where it is possible to write the angle needed, note that the angle can also be altered in the "Move/Rotate Options" window at a later stage.

The command line will ask to choose the reference point, left click where desired, the command will then ask for the target point, now one will be able to visualize the position of the new entity, left click when the position is correct. This will prop the "Move/Rotate Options" window. giving the following options:

- Delete Original,
- Keep Original,
- Multiples Copies: will copy the number of entities desired equidistant from one another, keeping the original.

The Angle can be kept as is or altered at will.

One will also need to choose between "Use current attributes" and/or "Use current layer". In the first case the copies will be created on the same layer as the original, in the second case, and providing another layer was selected beforehand, the copies will be created in the selected layer. In case both are ticked, "Use current layer" will prevail.

NOTE: this tool and the tool Rotate can often be used indifferently to obtain the same result.

Rotate Two:



This will permit to rotate entities according to two different angles around two reference points. Once selected it will prop the Select icons so one can select the desired entities,

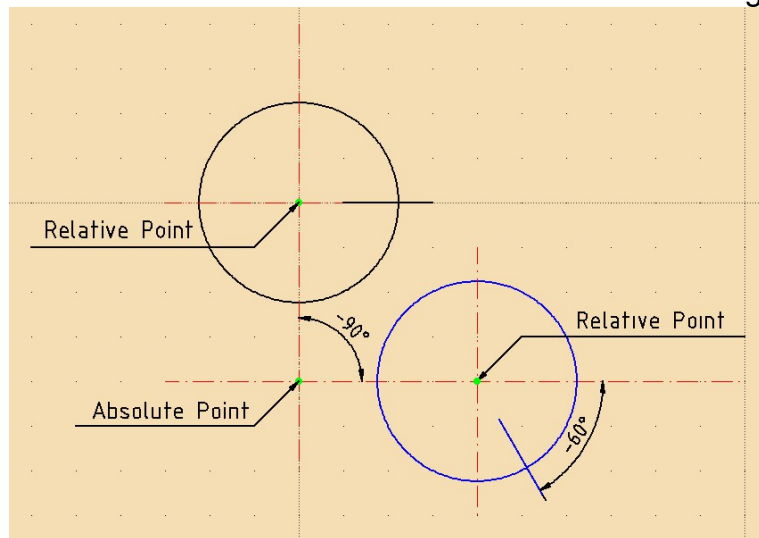
once done either press Enter or click on the Continue arrows below the Select icons. The command line will ask to specify an absolute reference point, left click where desired. The command line will then ask to specify a relative reference point, left click where desired. The Rotate Two Options window will pop giving the following options:

- Delete Original,
- Keep Original,
- Multiples Copies: will copy the number of entities desired equidistant from one another, keeping the original.

The first angle to be specified is the rotation around the absolute reference point, the second one is the rotation around the relative reference point.

One will also need to choose between "Use current attributes" and/or "Use current layer". In the first case the copies will be created on the same layer as the original, in the second case, and providing another layer was selected beforehand, the copies will be created in the selected layer. In case both are ticked, "Use current layer" will prevail.

The picture below illustrates the double rotation of a circle with a segment at 3 O'clock.



The original entity is in black, the result of the double rotation is blue. The angles used are respectively -90° and -60° . The reference points are named.

Revert direction:



Apparently this is used for tool path generation for CNC machines. I cannot describe here what exactly it does.

Trim:



This will permit to adjust the length of a segment, circle, arc, ellipse or arc of ellipse when it crosses a similar entity. First select the entity to be used as the limiting element, then left click on the element to be adjusted in the following way:

- two lines are crossing: select the limiting one and click on the part of the other line that will have to remain, the other part of it will be deleted;
- two lines potentially cross but are drawn short of the crossing point: select one and click on the other, it will be extended to the cross point. Right click, select the second line just extended and then click on the first, it will also be extended

to the cross point;

- a line crosses a circle and one wants to reduce the circle in an arc limited by the line: first select the line then click close to one end of the desired arc and then to the other end of it, the other part of the circle will be deleted;
- an arc and a line potentially cross but are drawn short of the crossing points, they need to be extended to the crossing points: first select the arc then click close to one end of the line, it will be extended to the cross point, click again on the other end of the line, it will extend to the other cross point, right click, select the line, click close to one end of the arc, it will be extended to the cross point, click on the other end of the arc, it will be extended to the other cross point.

These are just some examples, possible combinations can be many.

Trim Two:



Much like the Trim above this will adjust the length of a segment, circle, arc, ellipse or arc of ellipse when it crosses a similar entity. The difference is that it is aimed at adjusting the two entities at once. It works best on two crossing segments in the following way:

- two lines are crossing: select the first one and the other the extension beyond the crossing point will be deleted;
- two lines potentially cross but are drawn short of the crossing point: select the first one the other, both will be extended to the cross point.

It does not seem so efficient when it comes to circles and ellipses though it does save a few clicks.

Lengthen:



This will permit to adjust the length of a line, arc or arc of ellipse. Once selected it will prop the tool option to set the length and a tick case.

If one needs to extend the length of a line by 20 units, enter 20 in the "length" box and uncheck the checkbox; click on the end of the line where the extension is required.

If one needs to shorten the line, just enter -20 and follow the same procedure.

If one needs to set the line length at 20 units, the line being longer, enter 20 in the "length" box and check the checkbox, click close to the end that will be removed.

If one needs to set the line length at 20 units, the line being shorter, enter 20 in the "length" box and check the checkbox, click close to the end that will be extended.

Offset:



This will draw a parallel line to the entity/entities (line, arc and circle but not ellipse or polyline) selected at the given distance.

At the moment of writing it is best to select the entity/entities first and then click on the Offset icon this will prop the tool option: Distance. It will draw a parallel to any line on the drawing at the given distance when the mouse is moved over said line, left click when the parallels are located where one wants them to be.

Bevel:



This will draw a bevel (or chamfer) between two lines and/or arcs. Once selected it will prop the tool options: Trim, Length 1, Length 2.

Checking Trim will remove the lines between the bevel and the lines junction.

Length 1 is the distance between the lines junction and the start of the bevel.

Length 2 is the distance between the lines junction and the end of the bevel.

Fillet:



This will draw a fillet between two lines and/or arcs. Once selected it will prop the tool options: Trim, Radius.

Checking Trim will remove the lines between the fillet and the lines junction.

Radius is this of the fillet.

Divide:



This will divide an entity in more than one segment. It can be used on lines; circles, arcs, ellipses, arcs of ellipses and through points splines.

Once selected the command line will ask: "Specify entity to cut", once the entity is

selected the command line will ask: "Specify cutting point", it can be the intersection with a line or another entity or any chosen point.

Stretch:



This will stretch/deform such entities as lines, polygons and polylines other than "angle", it does not work on circles, ellipses, polylines angle and splines.

When selected the command line will ask to select a first corner, then a second corner and then a reference point which can be located anywhere on the drawing. The principal is that the two selected corners will be displaced via the reference point. The new entity will replace the original one.

Here are a few specific cases:

1. A line:

- if both extremities are chosen as the two corners, it will be moved keeping its original attributes (length, angle).
- If one extremity is chosen as both corners, only that extremity will be moved.

2. A rectangle or parallelogram:

- If two adjacent corners are chosen as the two corners, they will be moved.
- If two opposite corners are chosen, the rectangle or parallelogram will be moved keeping its original attributes.

Properties:



This will detail the properties of a single entity.

When selected one must click on the desired entity, a window will open giving the following details

- The type of entity, i.e., Circle, line etc.
- The name of the layer on which the entity is drawn. The layer can be modified.

- The property of the line, color, width and type. All elements can be modified.
- The geometry where applicable. All elements can be modified.

With this tool one can identify the type of entity but also modify it, for instance the radius of a circle can be changed or the position of its center.

Attributes:



This will permit to change layer and line properties of several entities at the same time. When selected the command line will ask "select to modify attributes", at this point the selection must be made by clicking on each individual entity, once done press the Enter key or the Attributes tool again.

If one wants to use the Select tools for the selection of the entities, that need to be done first and then click on the Attributes tool.

Either way this will open the Attributes window where Layer and Pen will show "Unchanged", not the actual attributes as in the previous tool. At this point any of these can be changed.

Explode Text into Letters:



This will permit to turn a text into each individual letters.

When selected the command line will ask "select to explode text", at this point the selection must be made by clicking on the text concerned and then complete the action by pressing the Enter key or the Explode Text into Letters again.

One can also first select the text concerned and then press the Explode Text into Letters.