Modifying designs

This chapter covers the use of common editing commands available in the gCADPlus Modify drop down menu, the command line and on the modify toolbar.

Tip Do not forget that whenever an entity is selected, many of these editing commands are available from the right click 'pop up' menu.

When an object is selected and ready for the operation of a ‘modify’ operation, the entity shows in a magenta color.
The Erase command

Used to remove an entity from drawing editor. To erase an entity, do one of the following:

- Select the entity and right click, then choose Erase.
- Type E in the command area and press ENTER.
- Type ERASE in the command area and press ENTER.
- Select the object and select the scissors tool on the standard toolbar.
- Select the entity and hit the DEL key.

The Copy command

Makes a copy of an entity or group of entities. To copy an entity, do one of the following:

- Choose Copy from the Modify drop down menu (do not choose the CopyToClipboard option)
- On the Modify toolbar, click the Copy tool.
- Type COPY in the command line and then press Enter.
- Select the entity right click and select Copy.

When asked for a base point, select a point on the entity to be copied. Then hit ENTER

Follow this an indicate the point of displacement.

Tips: Beginners often fail to mark the base point correctly - it need to be directly on the entity to be copied.

Right click in the middle of the copy command in order to control whether or not to multiply the copy, specify the displacement, input a pair of coordinates or call up a snap override.

Use copy together with Tracking. Set additional tracking angles as required.

The COPY command can be used to generate a circular array. Hold the SHIFT key down while copying around a center point. More on the use of the copy command can be found on YouTube - type ‘gCADPlus copy’ in the search box.

Use copy and copy 400mm pavers a fixed distance.
Move

An entity can be moved from one position (selecting a base point) to another using the move command. The entity (or group of entities) is shifted from one location to another. To move an entity, do one of the following:

Choose Move from the Modify drop down menu (do not choose the CopyToClipboard option) or,
on the Modify toolbar, click the Move tool, or
type MOVE or M in the command line and then press ENTER, or
Select the entity right click and select Move.

Here is a short movie showing the use of the move command in operation.
Rotate

The ROTATE command spins an object(s) about the Z axis. To rotate an object, do one of the following:

Choose Rotate from the Modify drop down menu.
On the modify toolbar, click the Rotate tool.
Type ROTATE or RO in the command line and then press Enter.
Select the entity right click and select Rotate.

Tip: The ROTATE command can be used to generate a circular array. This small movie illustrates the use of the ROTATE command.

More example of the use of the rotate command can be found on YouTube - type gCADPlus rotate in the search box.
SCALE

The SCALE command can be used to make an object (or group of objects) smaller or larger. To scale an entity, do one of the following:

Choose Scale from the Modify drop down menu. In gCADPlus, the SCALE command is commonly used to scale symbols representing plant specimens. If you are working in the USA, the symbols in the drop down menu Library > Plants > Standard group are drawn with a canopy diameter of 3 feet. If you are working in the metric system, the canopy diameter is 1000 mm. As a symbol is inserted, you have the ability to scale the drawing as it is inserted. Typing a factor of 2 in the USA system where the base unit is decimal feet brings the new symbol in with a canopy diameter of 2 X 3 feet, i.e. 6 feet. Similarly, in the metric version of gCADPlus, a scale of 2 results in a symbol 2000 units (mm) wide - 2 meters.

This small movie shows how the scale command can be used in a drawing where the base unit is decimal feet.

It is worth experimenting with scaling symbols dynamically as shown in the movie. Select the symbol, respond to the prompts by marking a base point (in the center of the symbol is best) when asked for the scale factor or reference point, move your cursor to the edge of the canopy and click, finally drag in to reduce the size of a symbol or out to increase its size.

The symbols available in the Library > Plants > Sized groups (Climbers, Grasses, Ground Cover, Hedge, Large Shrubs, Medium shrubs, Mini Shrubs, Small Trees, Tall Trees have already been sized, but that does not preclude the use of the SCALE command to further adjust canopy diameter.

On the modify toolbar, click the Scale tool.

Type SCALE in the command line and then press ENTER.

Select the entity right click and select Scale from the pop up menu.

Scaling can be dynamic (drag) or type a value in the command line. Values > 1.0 magnify the entity and values <1.0 shrink the object.

Tip: Whole drawings can be scaled. We prefer to scale about coordinate 0,0 as a base point if scaling and entire drawing. More options are available on the right click pop up menu in the middle of the SCALE command.
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Mirror

This command is used to gather a group of entities and to copy (mirror) them about a mirror line. To mirror an entity object, do one of the following:

Choose Mirror from the Modify drop down menu.

On the modify toolbar, click the Mirror tool.

Type MIRROR in the command line and then press ENTER.

Select the entity right click and select Mirror.

Here is a short movie showing how the mirror command can be used to rapidly build up a design for part of an entrance portico. We show how to handle text when using the mirror command.
Explode

The EXPLODE command allows you to change a grouped item into its individual elements so it can be edited. Putting it another way, the EXPLODE command breaks a compound object like a BLOCK or even an entire drawing up into its constituent parts (separate entities).

To explode an object, do one of the following:
Choose Explode from the Modify drop down menu.
On a toolbar, click the Explode tool.
Type EXPLODE in the command line and then press Enter.
Select the entity, right click and select explode.

A word of caution - if complex objects like hatching are exploded, the file size of the drawing may increase dramatically.

This small movie illustrates the use of the explode command.
Join

The Join command can be used to stitch together a series of individual entities such as line segments into one polyline entity. This makes operations like hatching so much more convenient.

To join an entity object, do one of the following:

- Choose Join from the Modify drop down menu.
- On the modify toolbar, click the Join tool.
- Type JOIN in the command line and then press ENTER.
- Select the entity right click and select Join.

We use the JOIN command to weld together several lines. The resultant polyline is then easily hatched.

In the figure below, the Join command can be used to weld the line and the arc (because they have the same endpoint).
Trim

The TRIM command can be used to clean up overhanging entities over a boundary. To trim an entity object, do one of the following:

Choose Trim from the Modify drop down menu.

On the modify toolbar, click the Trim tool.

Type TRIM in the command line and then press ENTER.

We trim several overhanging lines and create a new layout to display a design for the front entrance of a house.
Extend

This command enables entities to be extended to a boundary. Start the command with a menu item, a toolbar icon or by typing the command. The first prompt asks for the boundary to which the entity is to be extended, right click, then choose the entity to extend.

To extend an entity to a boundary, do one of the following:

Choose Extend from the Modify drop down menu.

On the modify toolbar, click the Extend tool.

Type EXTEND in the command line and then press ENTER.

Here is a short movie illustrating the use of the EXTEND command.
Offset

The OFFSET command makes copies of object(s) parallel to the current position at a distance you nominate. The lines at right and top have been offset 230 mm to create a double brick wall.

The offset command can be started by:

- Choosing Offset from the Modify drop down menu.
- Type OFFSET or O in the command line and then press Enter.
- Select the entity, right click and select Offset.
- Select from a toolbar option.

You will be asked for an offset distance. Type the distance in the command area and hit <ENTER> or select the ‘Through’ option and simply click the point in the drawing where you want the offset to appear.

This short movie shows the offset command in action in the metric environment.

This short movie shows the offset command in action in the Imperial environment.

Use a pop up menu item to set the offset distance.
ARRAY

This command creates copies of objects arranged in a pattern. It is possible to create copies of objects either in a regularly spaced rectangular or polar array. A rectangular array distributes copies of the selected object into any combination of rows and columns while a polar array evenly distributes copies of the object in a circular pattern around a center point or axis of rotation.

Creating a polar array.

Using the array command to quickly place some bricks around a garden bed. We create a rectangular array.

Create the design shown below.
STRETCH

The STRETCH command is poorly understood by CAD users. It allows you to adjust/move objects like a door or a window along a wall with little effort. It is a little tricky to use and the most important thing to keep in mind is to start with a crossing window at the first prompt. Perhaps you are already using Stretch without knowing it. If an entity such as a line is selected and you click on a handle to make it hot, moving the handle utilizes the stretch command without the need to call it.

To STRETCH, do one of the following:
Choose Stretch from the Modify drop down menu.
On the modify toolbar, click the Stretch tool.

This movie illustrates the use of the STRETCH command.
BREAK
The BREAK command slices a segment from an entity such as a line.
To BREAK an entity, do one of the following:
Choose Break from the Modify drop down menu.
On the modify toolbar, click the Break tool

This movie shows the break command in action. We break twice at a point on a line to remove a doubled up segment.

Remove a segment from a line or polyline entity with break.
EXPLODE

The explode command is usually used to break a complex, compound symbol or group of entities (a block) into a separate set of entities that made up the block.

The explode command can be started by:

Choosing Explode from the Modify drop down menu.

Type Explode or X in the command line and then press Enter.

Select the entity, right click and select Explode.

Select from a toolbar option.

The figure below shows a complex block (plant symbol) highlighted in magenta with a single pick, and a copy of the symbol exploded on the right where a single click selects one entity.
Explode polylines

This command allows you to edit a section of a continuous polyline by “exploding” the line and morphing it into individual lines. This command is important, as it lets you select certain areas of a line without selecting a whole shape.

The figure below shows an intact polyline with three vertices on the right with an explode polyline on the left. There are so many individual segments that editing the polyline becomes exceedingly difficult.

Exploding a polyline.
FILLET

The fillet command enables users to create a rounded corner (or fillet) between various entities such as lines, polylines, arcs etc. by extending them, then trimming.

It is well worth experimenting with the FILLET command in gCADPlus. It can save much editing time. For example, the fillet command can be used to square off a corner in one click without the need for a double trim. Alternatively, the resultant corner can be given a radius.

This movie shows the FILLET command in action.

The fillet command can be used to clean up corners.
ORDER command (Draw order)

gCADPlus provides DRAWORDER tools to allow control over the stacking order of entities. Although we have placed the options in the Tools drop down menu, it is available via the Modify toolbar.

One (entity) object can be set to lie under or above another if required. Options are: send to the back, bring to the front, and for more subtlety, send above or below an entity.

These small movies shows some of the issues involved in the use of the draw order tools.

The tall tree symbol is above the understory planting (and paving).
DIVIDE

If you have been used to drafting with AutoCAD, you may have used the DIVIDE and MEASURE commands. We have not implemented a measure command because measuring along a line or other entity a fixed distance in gCADPlus can be carried out easily using tracking, but a DIVIDE command is another matter.

A DIVIDE command is available in gCADPlus and it is called from the modify drop down menu. The first prompt asks for the entity to be divided and the second asks for the number of segments. The line (or other entity) is then divided by placing appropriately spaced points along the line.

Tip: You may need to adjust the size of the point marker if the divisor points do not show clearly. This is done from the Format drop down menu.

Here is a short movie showing how the DIVIDE command is used. We show how to space lighting an equal distance apart in a low voltage supply line.
ELEVATION

This command stores the Z value with an entity. For example, if a rectangle used to denote a raised paved area is assigned an elevation of 300 mm (when working in metric base units, 1.5 feet when working in imperial units) then when the drawing is passed to SketchUp to build a 3D model, the rectangle will automatically placed at that level. This greatly speeds up 3D model making.
Set fixed scale

It is usual to print drawings from layout space. The scale within a floating viewport on a layout sheet is set in the properties box. Changing the scale factor to meet standards like 1/96, 1/100, etc. can get a little trick. This command allows for selection of a desired scale directly from the command line.

Fixed scale off

This command clears a scale setting associated with a floating viewport.

Set scale factors in floating viewport frames is simplified with this tool.
MATCH PROPERTIES

This tool is used to select an entity, capture its properties and apply the same set of properties to one or more other entities. It is a great time saver when editing.

Select the option from the Modify drop down menu, indicate a source entity, and then an entity to be matched.

A movie showing the operation of the Match Properties tool. The colour and layer properties of one set of hatching in a bush regeneration mix is applied to other hatched areas.

MatchProperties - one of the most valuable editing tools in gCADPlus.
Align objects

If there is a need to lining up a group of entities when drafting - the underground water tanks in the example below - this ALIGNOBJECT tool can be used.

This movie shows how to use the align command to simplify the placement of a group of underground water storage tanks.

Align objects - command line options

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<thead>
<tr>
<th>Command Line</th>
<th>SNAP</th>
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<th>ORTHO</th>
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Aligning a group of objects (entities).
Align text

This command is used to line up text. Various options such as left and right justify are available.

Tip: the tool works well in layout space and can be used there to neaten up the text in a presentation sheet.

This very short movie shows the AlignText tool in action in layout space.

Aligning a group of text entities in layout space.
Introduce variation to symbols

This tool can be used to vary the size and rotation of a group of symbol thereby giving the drawing a more ‘organic’ feel.

A short movie showing how the ‘introduce variation‘ tool works.
COPSE command

This command takes a series of overlapping symbols and converts them into a cleaner group symbol (a copse).

This small movie shows the operation of the COPSE tool.

The COPSE command simplifies groups of symbols.