Innovative EDA Solutions
Quadcept

Quadcept V10.5.0 Release Notes

Release Date 2021/10/25

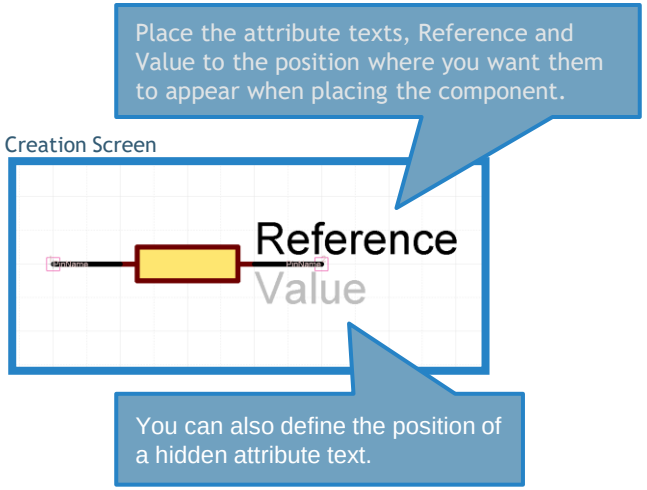
SCH

Attribute Text Positions Now Definable During Symbol Creation

Innovative EDA Solutions Quadcept

- You can now define the location of the attribute texts including Reference and Value when creating symbols.
- This allows you to adjust the positions of the attributes in accordance with the symbol shape you are creating.

Symbol Creation Screen

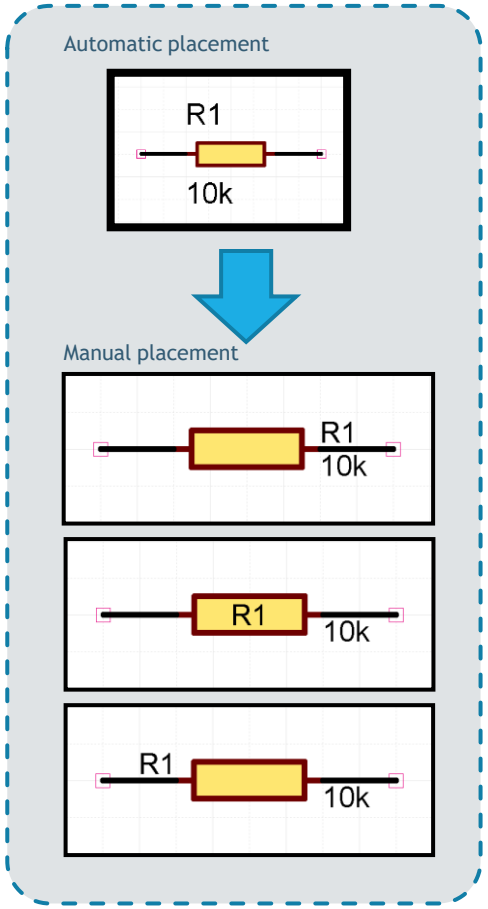


* By default, Reference and Value are set to Show and Hide respectively. Change the setting to Show as necessary.

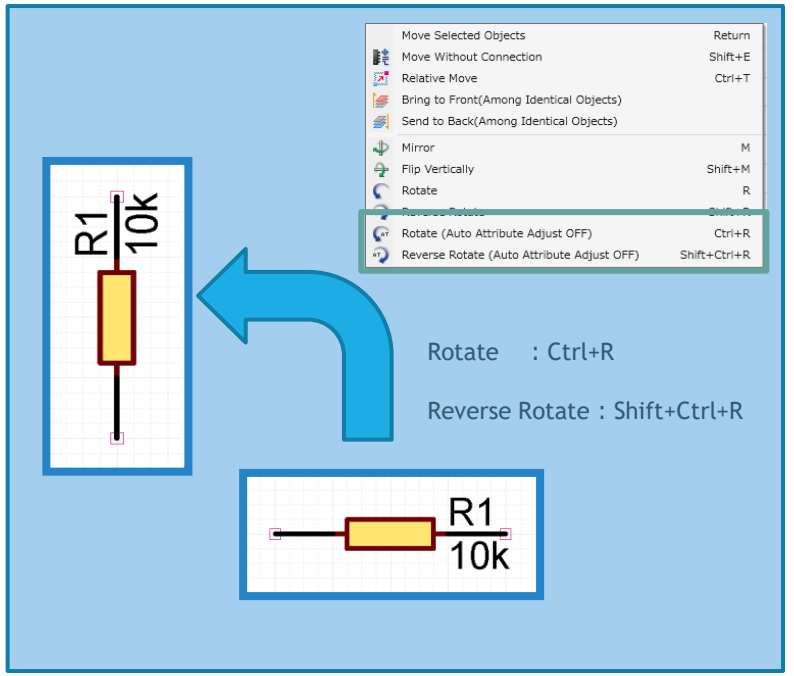
* You can also delete Reference and Value. If you delete them, their positions will be automatically adjusted by the system.

* You can also place user-defined attribute texts.

* The user-defined attribute texts will be added as component attributes when creating components.



Run **Rotate(Auto Attribute Adjust OFF)** if you want to rotate the symbol without the attribute texts being automatically adjusted.



Online Manual

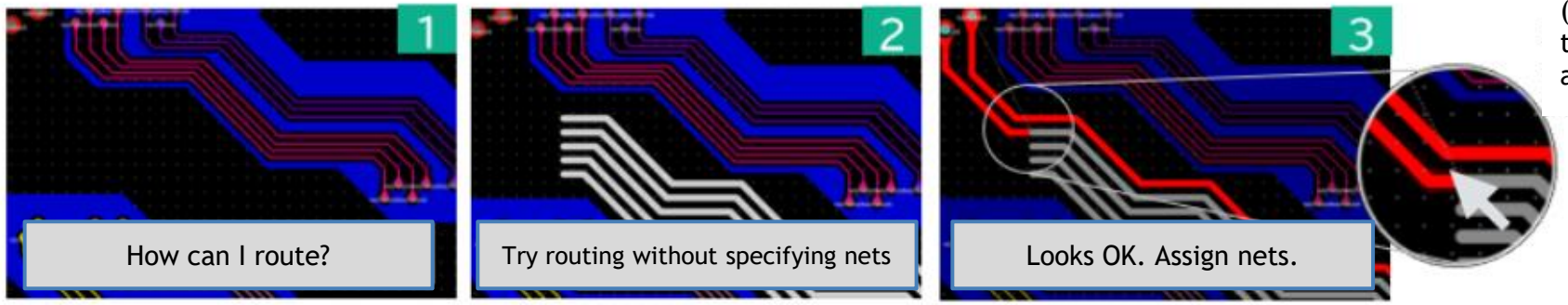
2

PCB

Enhanced Routing Capabilities

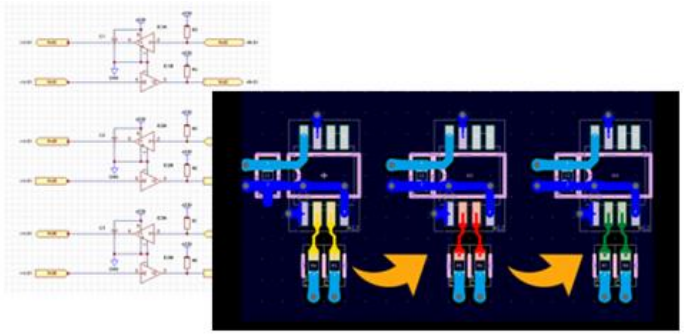
- The routing capabilities have been enhanced to allow you to create routes without nets.
- This is useful when you plan for high-density PCB routing or you copy and paste the same pattern repeatedly. In addition, this enables you to convert line objects imported from DXF files into routing objects.

Plan for Routing



If you connect a route with a net (red) to a route without a net (gray), the net that the red route has will be assigned to the gray route.

Reuse the Same Pattern



If you copy the pattern on the left block (Net: VCC1) and paste it on the right block, the nets of the pattern will be automatically changed in accordance with the nets of component pins in the right block.

Click here for more information, including examples of using routes without nets

Quadcept Blog

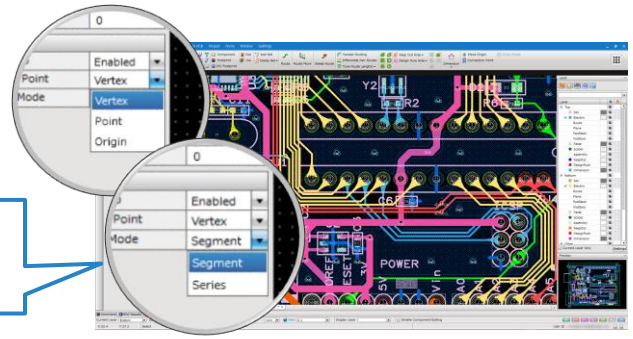
PCB

Enhanced Move / Copy / Cut Modes

Innovative EDA Solutions Quadcept

- The same options as the “Copy Continuously” mode that was added in V10.3 are now available in the Move, Copy, and Cut modes. This makes it easier to repeat operations.

The Same Options as the “Copy Continuously” Mode Now Available in the Move, Copy, and Cut Modes

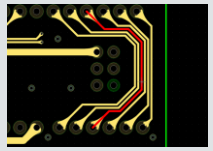


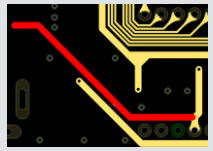
Specify options in the Property window

Segment

A single object

Series



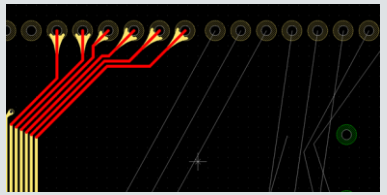


A series of routing objects, i.e., all contiguous routes until a via, pad or junction is encountered.

* You can choose a reference point when you select a single object.

Vertex : Sets a reference point to the vertex of the object.
(Useful when selecting a polygon.)
Point : Sets a reference point to the location you click.
(The point will not be on the Grid.)
Origin : Sets a reference point to the origin of the object.

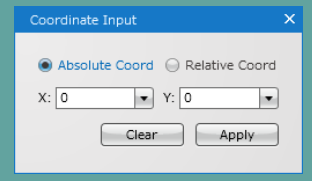
①Drag-select the object(s) you want to manipulate. (You can add/remove objects from the selection with the Ctrl/Shift key.)



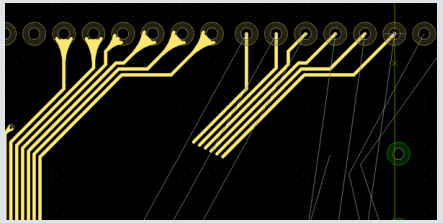
②Select a reference point. (You can also specify it by inputting X and Y coordinates.)

POINT!

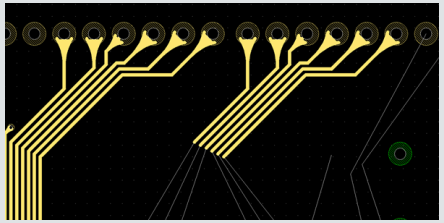
To input the X and Y coordinates, choose the Edit >> Coordinates Input command.



③The copied content will appear floating on the cursor. Move it to your desired location.



④Click to paste it.



* The copied content can be rotated/flipped while in each mode.
* You can see the current cursor location at the status bar while moving the copied content.
* To manipulate other objects continuously, right-click to select the Next command.
* If you want to specify a reference point again after the ① step, you can do that by right-clicking to select the Pushback command.

Online Manual

PCB

Added Ability to Generate Planar Transformer Routes

Innovative EDA Solutions Quadcept

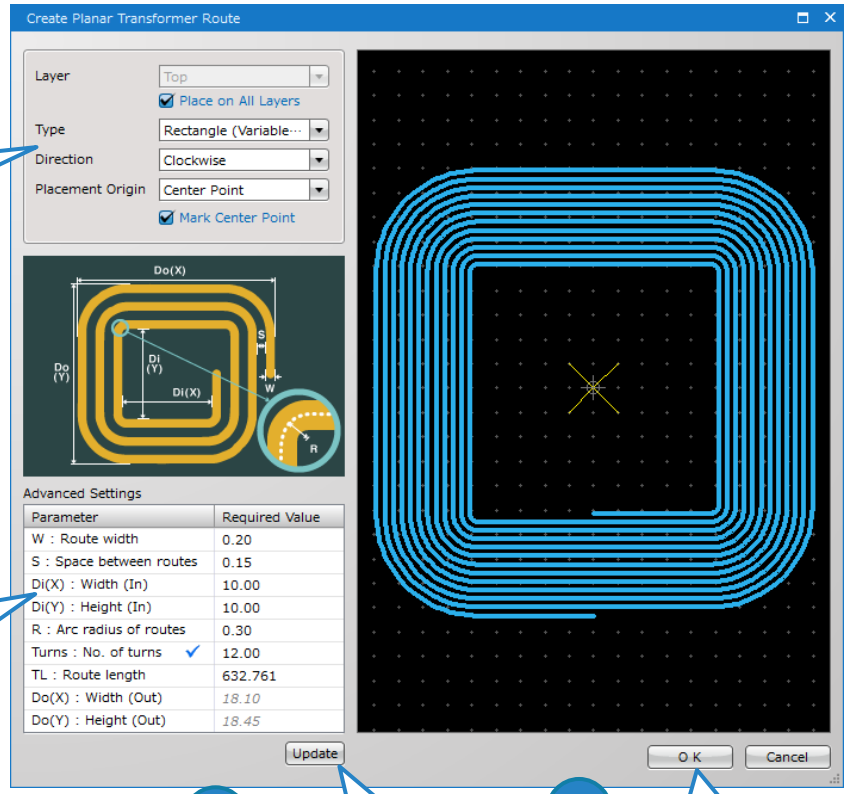
- 4 types of planar transformers can now be generated easily and quickly using the dedicated wizard.
- Just setting parameters will allow you to generate planar transformer routes on your PCB design.

Purposes of planar transformer routes

- Use as a coil
- Use as an antenna
- Use to generate heat etc.

Automatic Planar Transformer Generation

[Menu] Create PCB >> Create Planar Transformer Route



1 Configure basic settings

2 Configure advanced settings

3 Update preview

4 Generate

Type	Rectangle (Variable Corner Radius)	Rectangle (Equal Corner Radius)	Rectangle	Circle
Sample				

Parameters	Description
W : Route width	Route width
S : Space between routes	Clearance between routes
Di (X) : Width (In)	Inner diameter (X-axis)
Di (Y) : Height (In)	Inner diameter (Y-axis)
R : Arc radius of routes	Arc radius of corners
Turns : Number of turns	Number of turns
TL : Route length	Total route length
Do (X) : Width (Out)	Outer diameter (X-axis)
Do (Y) : Height (Out)	Outer diameter (Y-axis)

* The planar transformer routes will be generated without nets.

* The routes will be generated either the number of turns or the route length. The last input takes precedence. A check mark will be displayed for the one that is applied.

Online Manual

PCB

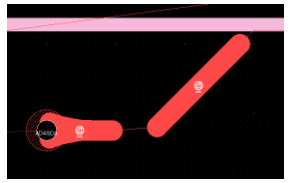
Added DRC Item for Checking Floating Routes

Innovative EDA Solutions Quadcept

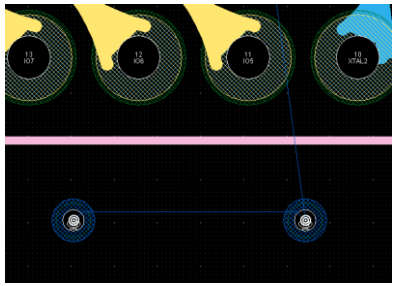
- A new DRC item has been added to allow you to check floating routes.
- You can now find routes, vias, pads and planes which are not connected to any component pads.

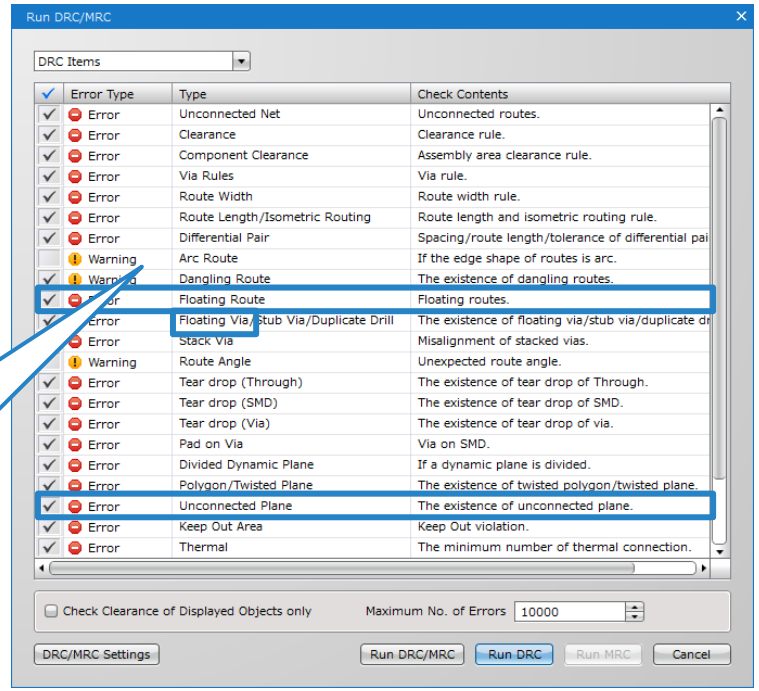
Floating Route

This will detect floating routes.
* Vias, pads, planes that are placed alone will not be detected by this check.



Floating Via





Unconnected Plane

Online Manual

PCB

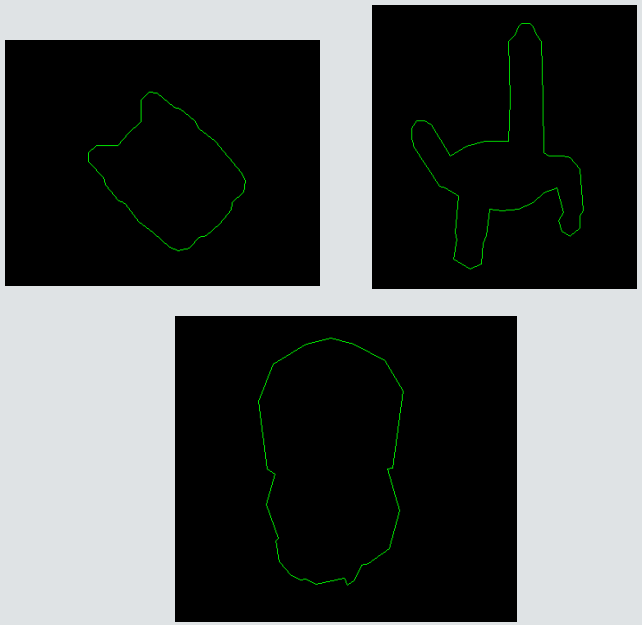
Enhanced DXF File Importer to Import Spline Curves

Innovative EDA Solutions Quadcept

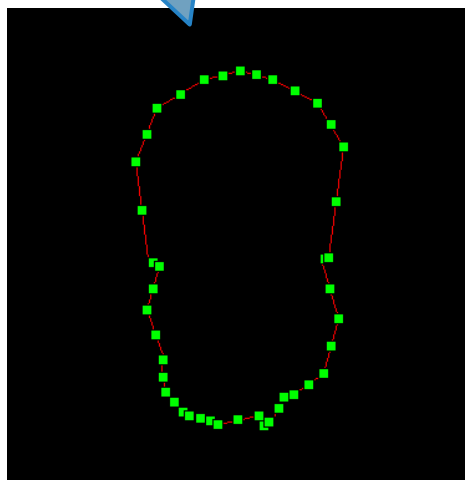
- Spline curves (smooth curves) can now be imported as line segments.
- This now makes it possible to import DXF files that contain complex shapes, such as board outlines, drawn with mechanical CAD systems.

Spline Curves Now Importable

DXF with complex shapes, including spline curves, drawn in mechanical CAD systems

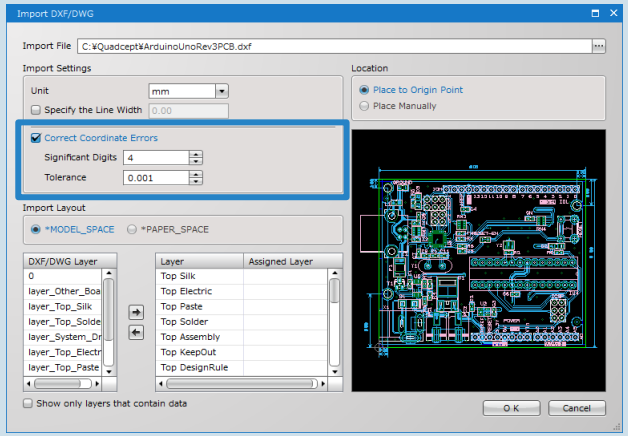


Now importable as line segments



Enhanced the ability to correct coordinate errors for DXF file import

Due to the coordinate errors caused during DXF file import, sometimes it was not possible to perform Spread Selecting or Convert to Filled Object for the imported lines. We have greatly enhanced this capability and now it is much easier to generate objects including board outlines and keep out areas.



Online Manual

PCB

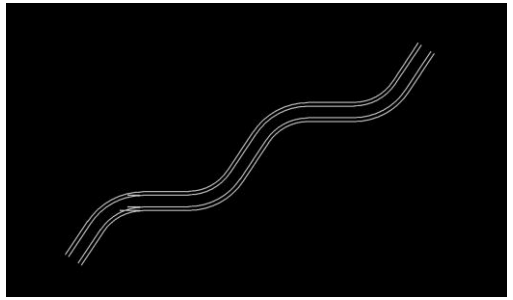
Added Ability to Convert Lines/Arcs into Routes

Innovative EDA Solutions Quadcept

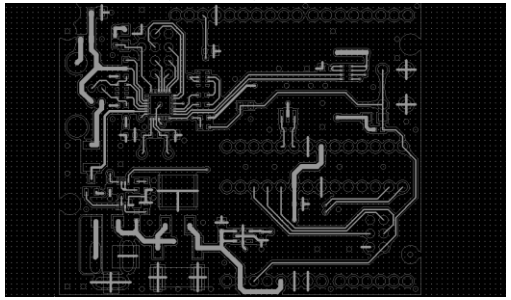
- Line and arc objects can now be converted into route objects.
- This allows users to convert complex shapes imported from DXF files or design data input from Gerber files into routing objects.
- Converting these objects into routes will allow users to define nets or use the Route Interpolation capability.

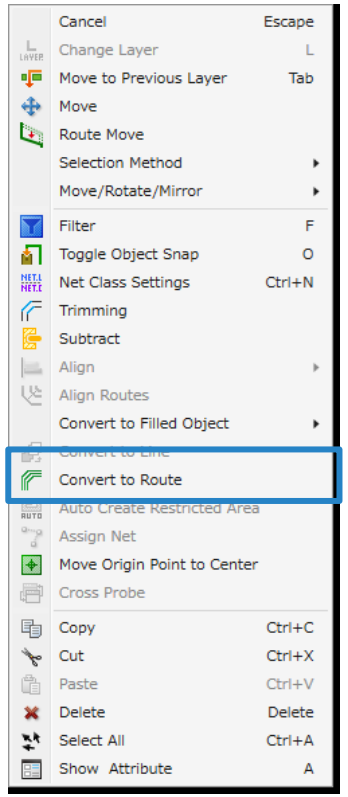
Use Lines as Routes

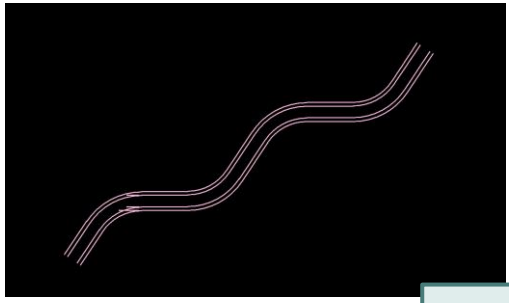
Lines / Arcs Imported from DXF files

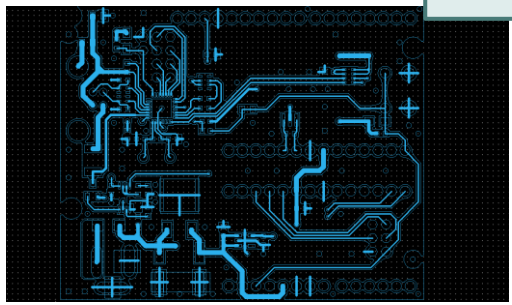


Gerber Data (Routes)





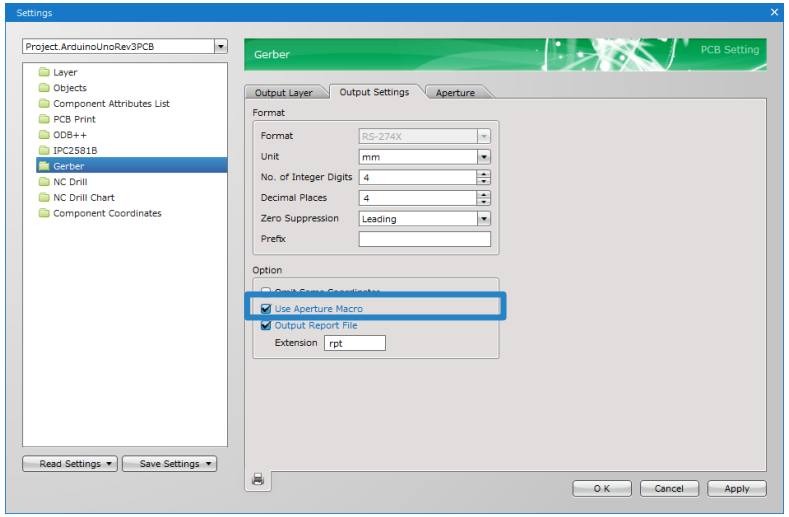




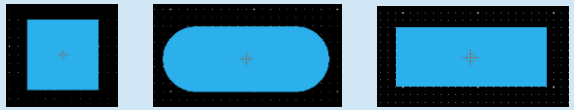
Available as Route Objects

Added Option to Enable/Disable Aperture Macro for Gerber Files

- The option has been added to use not the AM (Aperture Macro) parameter but G-codes (G36, G37) when exporting Gerber files in the extended Gerber format (RS-274X).
- If you are using an older board manufacturing machine or software, and the reader does not support AM (Aperture Macro) and cannot read the data, please uncheck the box and output the Gerber.



* Aperture macros are used for pads (squares, rectangles, and ovals).



Enable Aperture Macro

```
G71*
G90*
G04 Quadcept GERBER*
%MOMM*%
%FSLAX44Y44*%
%AMT10*0 Rectangle
45*21,1,0.8,1,0,0,45*%
%ADD10T10*%
G54D10*
X-544000Y-71000D03*
M02*
```

The data size will be smaller.

However, it may not be readable by older board manufacturing machines or software.

Disable Aperture Macro (G-Code)

```
G71*
G90*
G04 Quadcept GERBER*
%MOMM*%
%FSLAX44Y44*%
G75*
G36*
G01X-550364Y-70293D02*
G01X-543293Y-77364D01*
G01X-537636Y-71707D01*
G01X-544707Y-64636D01*
G01X-550364Y-7023D01*
G37*
M02*
```

The data size will be larger.

However, it can be read by older board manufacturing machines and software.