Printed Circuit Board Tutorial

Covers:

Defining Billet
Using shape boolean tool to combine shapes
Importing Gerber files
Grouping shapes together
Creating an offset path
Creating Offset Outside and Drill machining plans

This tutorial show how QuickCAM 2D Design can be used to manufacture a printed circuit board from a Gerber (RS247X) file.
Although there are no particular PCB designing tools, the software can be used to make minor modifications to an existing Gerber design.
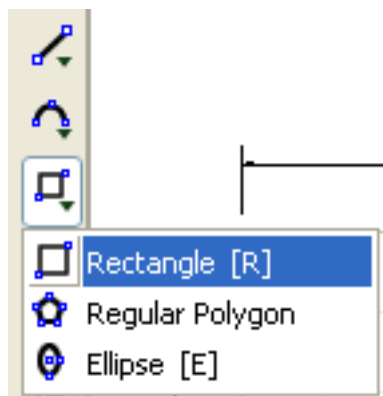
Define working area (billet)

Define the size of the billet needed to create the PCB - for this example, the Gerber used is about 40mm square, so set the billet slightly bigger at 50 by 50.

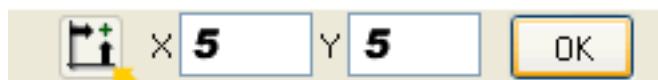This can be done as the software starts, or by clicking the resize material icon:

Create the PCB shape for cutting out

If the PCB needs to be fitted into a housing, there are various tools to help create the correct shape. For this example we know that the PCB must fit within a 45mm tube.

Select rectangle tool from the toolbar, or press **R** on the keyboard
To draw the rectangle at an exact position, use the co-ordinate entry boxes at the top of the design screen.

Absolute co-ordinates

Make sure the Absolute / Relative button is showing the absolute mode icon. Enter X5 Y5 into the co-ordinates and press OK
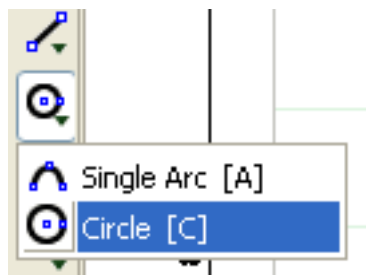
The system now wants to know the opposite corner of the rectangle, again use the co-ordinate entry boxes to specify a relative position to the last one entered.

Relative coordinates

Here, enter a relative size of X40 Y40. Pressing OK will now create a rectangle with 10,10 as the bottom left corner and 45,45

Now create the 45mm diameter circle which we can use to clip the rectangle:
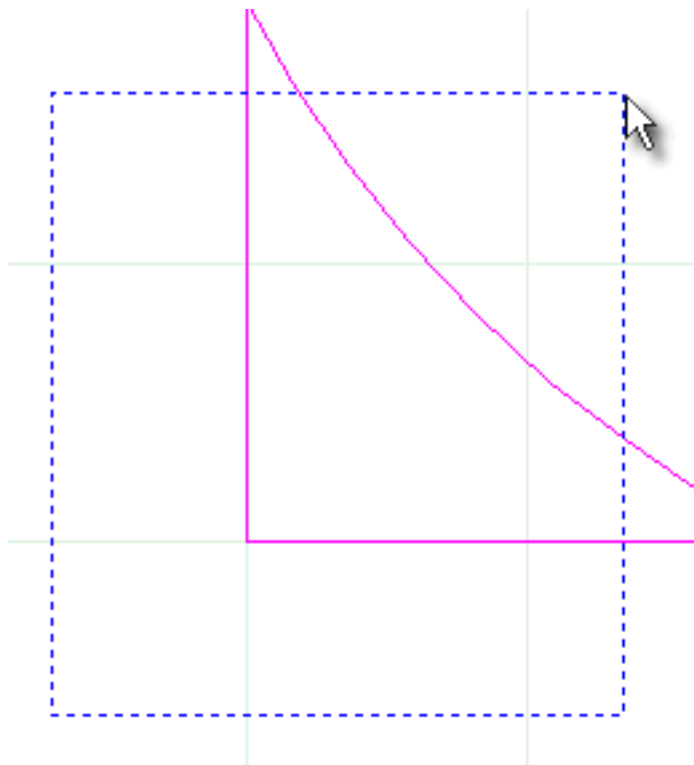
Single Arc [A]
Circle [C]

Select the circle tool from the toolbar, or press **C** on the keyboard

Enter X25 Y25 into the absolute coordinate edit boxes and press OK to define the centre of the circle

Change to relative mode and enter X22.5 Y0 and press OK to define the circles radius as 22.5 (45mm diameter)

Now select both the rectangle and the circle,

Move the mouse pointer to the left, bottom corner of the items we wish to duplicate
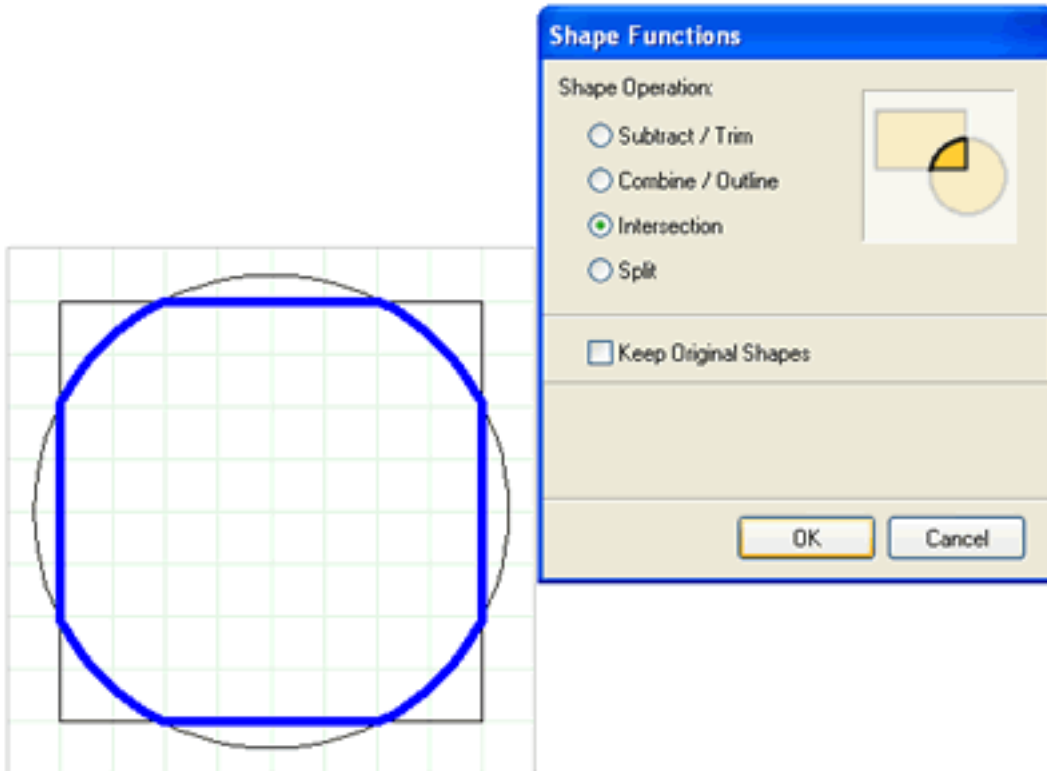
Press and hold the left mouse button

Now drag, whilst holding left mouse button down, the mouse to the top right corner of the design (as shown above) and let go of the mouse button
Alternatively, hold down SHIFT and click on each shape in turn

Now both items are selected, the Combine Shape tool will be active

NB, you must have more than one shape selected to use this tool

Shape Functions

Shape Operation:

- ○ Subtract / Trim
- ○ Combine / Outline
- ● Intersection
- ○ Split

☐ Keep Original Shapes

OK   Cancel

Choose the Intersection operation and you will see the result on the drawing screen in thick blue lines
Experiment with the other options to see how else this powerful tool can be used
The shape combine tool only works on closed shapes that have an area
- single lines will not be included in any boolean operations
- shapes such as open arcs will be automatically closed

Import the Gerber PCB design

For this tutorial, we are going to use a Gerber file which is installed with QuickCAM 2D Design into \My Documents\Denford\Gerber Files
The file is called **LightSwtch.gb0**

File   Edit   Draw   Gener

- New...
- Open...   Ctrl+O
- Reopen   ▶
- Save   Ctrl+S
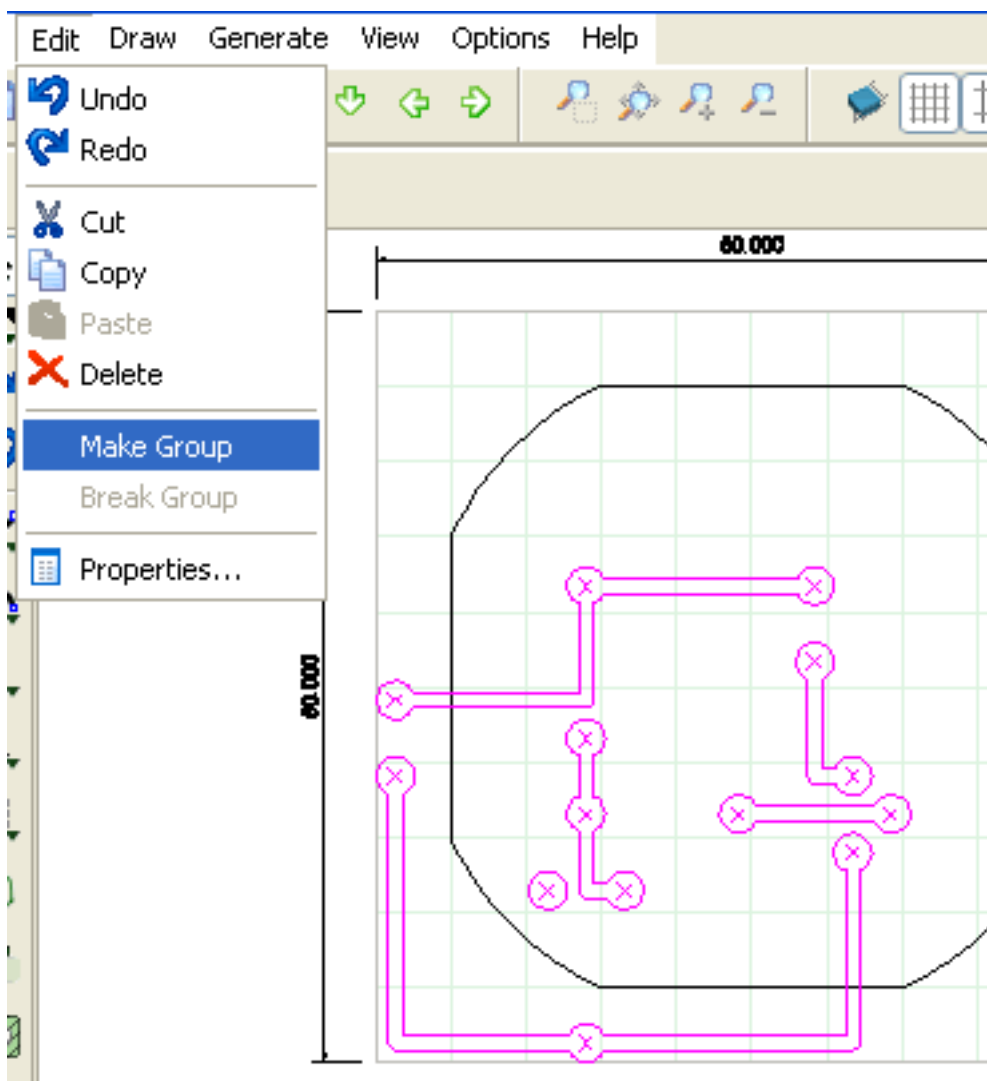- Save As...
- **Import...**
- Export...
- Print...
- Exit

From the File menu, select Import
From the file import dialog, choose **Gerber PCB (*.gb*)** as the file type
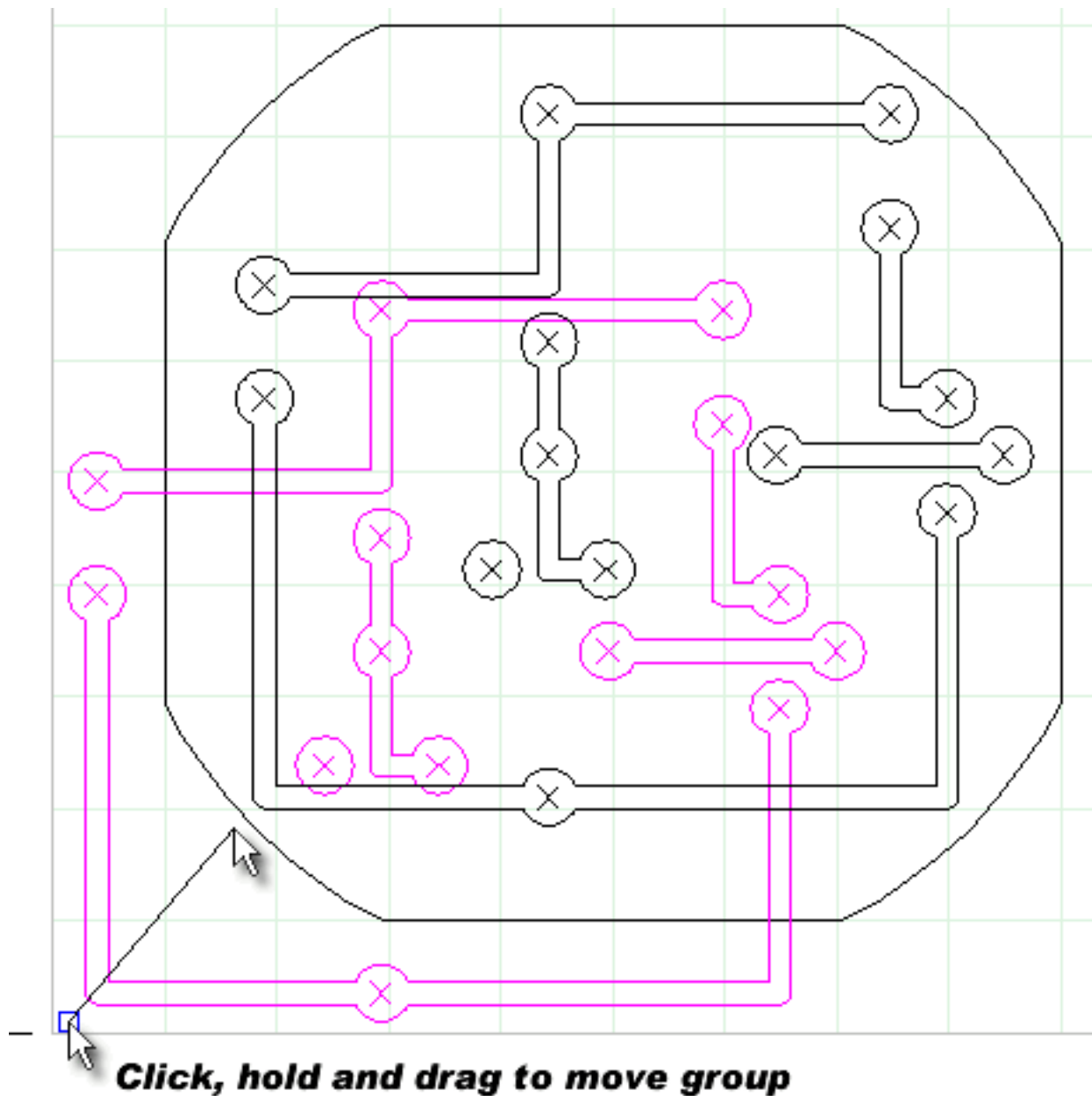Then browse your PC to find the LightSwitch file

When the PCB appears in the drawing, it is a good idea to group the tracks together to make life easier later on

Edit   Draw   Generate   View   Options   Help

- Undo
- Redo
- Cut
- Copy
- Paste
- Delete
- **Make Group**
- Break Group
- Properties...

60.000

60.000

With all the tracks selected, select the Make Group option from the Edit menu
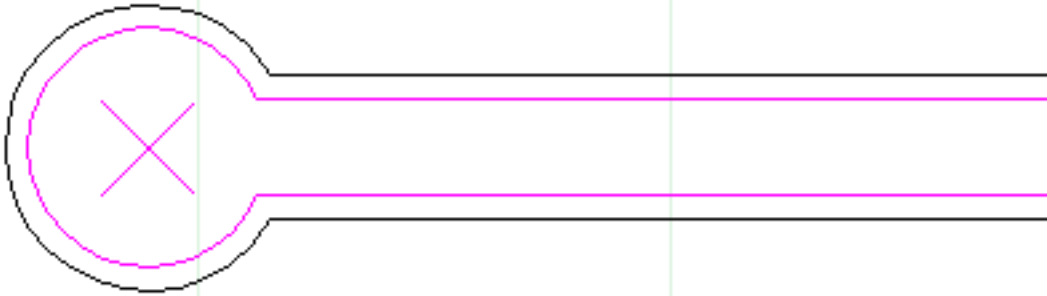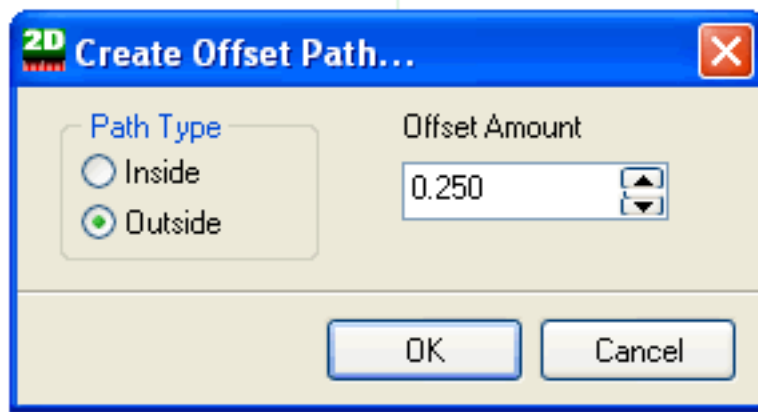When you now click on any of the circuit tracks, the whole design will be selected
The one grip in the lower left corner of the group will allow the tracks to be positioned as one
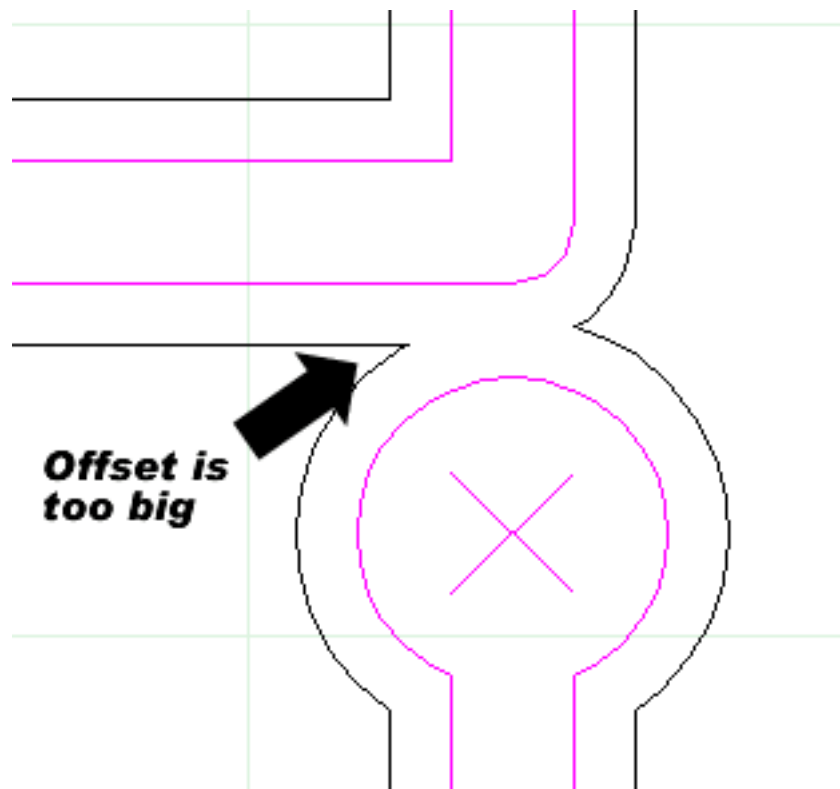
Click, hold and drag to move group

Create cutter path

There are two ways in which a cutter path can be created to machine isolation gaps around the tracks
The first is to create an offset path from the design screen - that way a follow path can be used in the CAM wizard and the exact diameter of the engraving tool need not be known
The other is to simply define a suitably small cutter (eg, 0.5mm diameter) and use it to create an offset toolpath within the CAM wizard

To create an offset path in the design, select the tracks group and choose the **Create Offset Path** tool from the toolbar [icon] or the Draw menu

Create Offset Path...

Path Type
- ○ Inside
- ● Outside

Offset Amount

0.250

OK   Cancel

If your cutter produced roughly 0.5mm wide cuts, then create and offset path of 0.25mm on the outside of the shapes

Offset is too big

Look out for tracks that are too close together - the offset path will not be created where it would mean cutting into another track

Alternatively, just run the CAM wizard to create a CNC program
For this example we have setup a 0.5mm engraving tool and material settings (feed / speed) for soft wood which should be fine for single layer copper -clad board and an 1/8" (3.175mm) slot mill for cutting out the PCB.
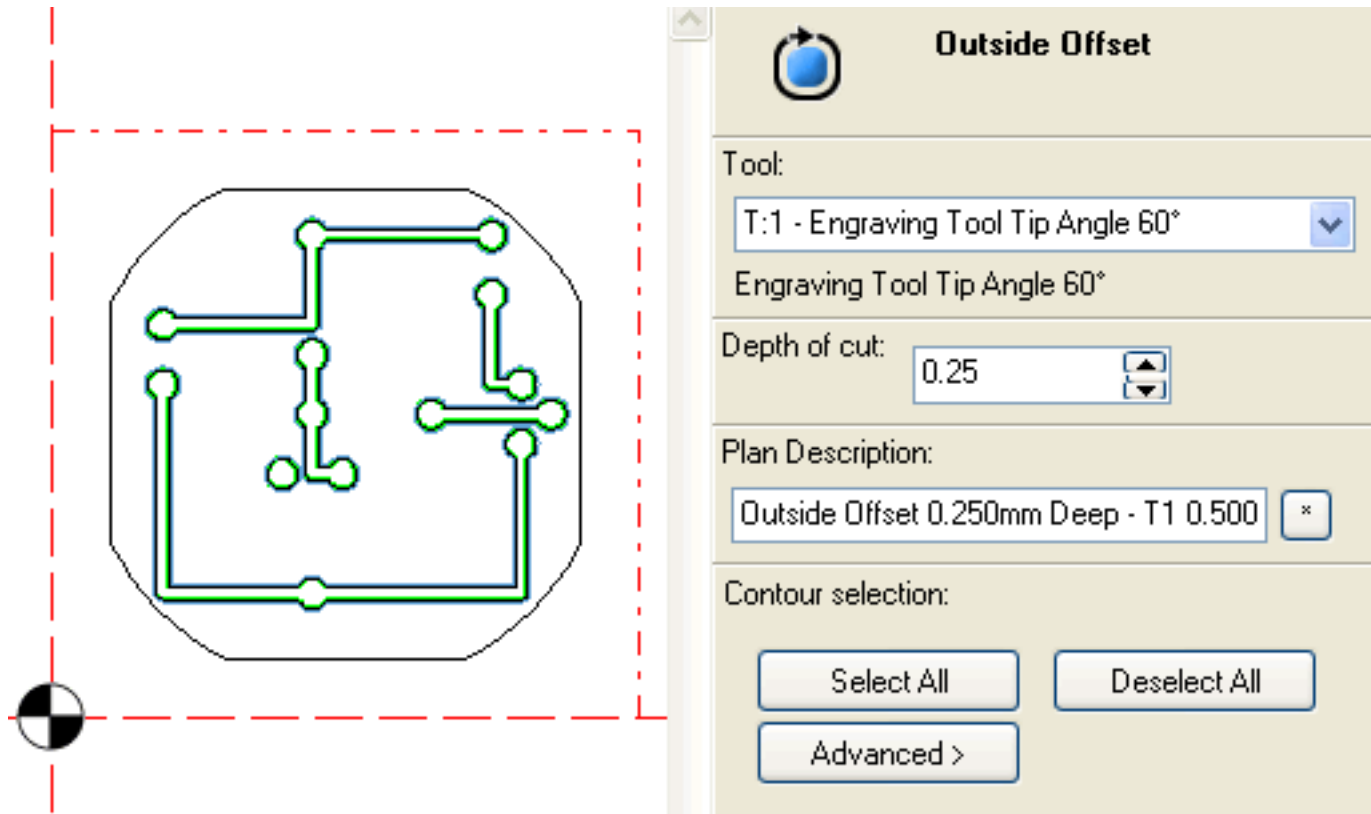Setting up tools and materials is covered in another tutorial.

Click the toolbar button: or select **CAM Wizard** from the Generate menu

Choose the correct material as this will affect the speed, feed and tool step-down depth in the final CNC program
Click Next >

Choose the machining plan **Outside Offset**
Make sure you select the correct tool as this effects the distance of the tracks' offset
Set a cut depth of about 0.25mm
Select the tracks by clicking each one in turn, or drag the selection box across all tracks
Now press OK

We now want to have the CNC spot-drill the pad centres (if you had a small drill defined, it would also be possible to drill all the way through)
Select the machining plan **Drill**
Keep the engraving tool selected
Keep cut depth to 0.25
Click Select All in order to choose all the points for spot-drilling
The points are created when the Gerber file is imported. The system generates points at the centre of every Gerber FLASH pad operation. It is possible that too many or too few points are created depending upon the file
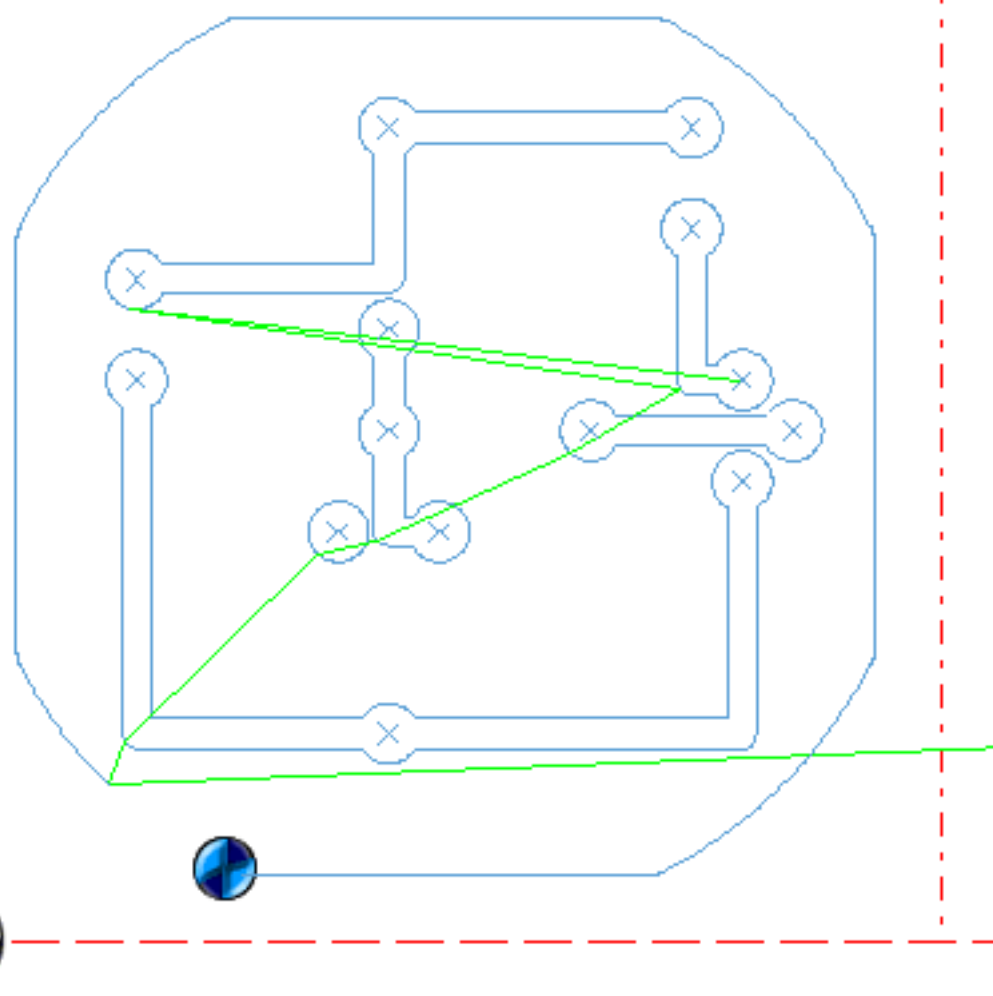
Next, select another **Outside Offset** plan
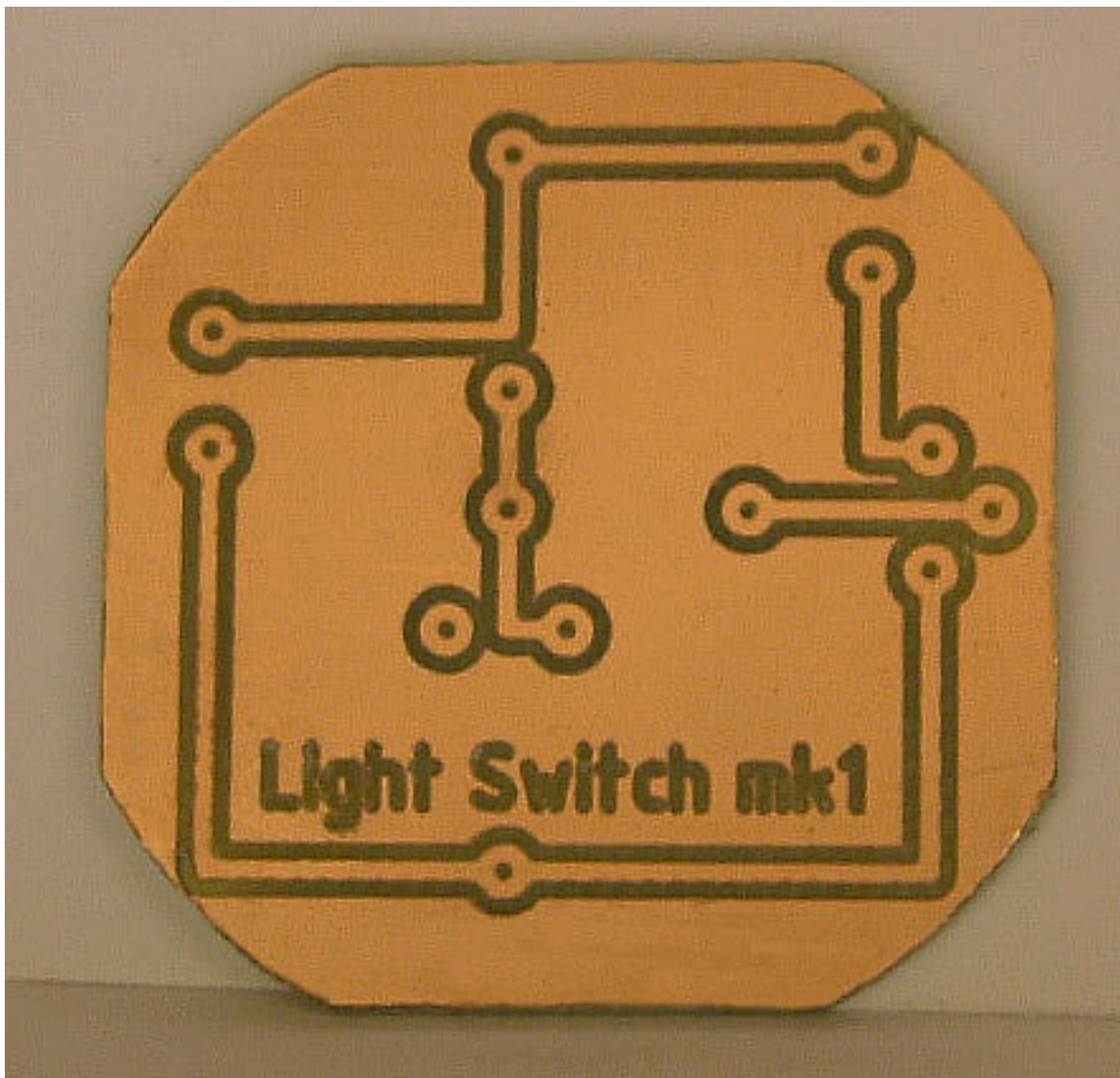Select the 3.175mm slot mill
Set a cut depth of 1.8mm (just enough to break through the board)
Click on the PCB shape and press OK

Click Next > simulate and post out the CNC program

The simulation in progress

The finished PCB
Try and add the single line text yourself (see the Key Fob tutorial for more details on how to do this)