Circuit PRO CAM Side Basic Operations

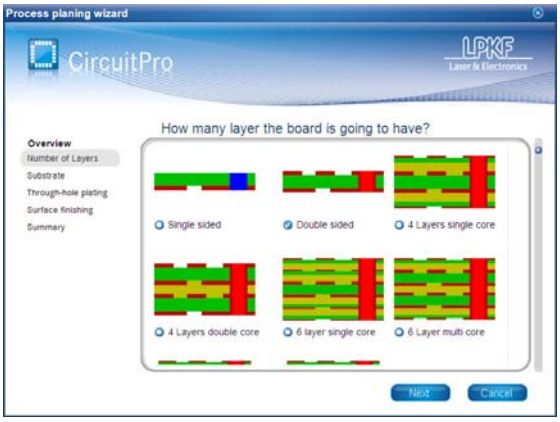
1. Launch Circuit Pro

   a. Double click on the Circuit Pro icon

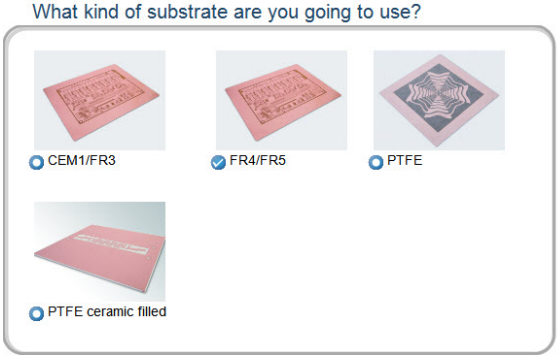
2. Execute Process Planning Wizard.

   a. Press the process planning wizard icon  , or go to the wizard menu and select process planning wizard.

   b. Select the amount of copper layers you are going to use.



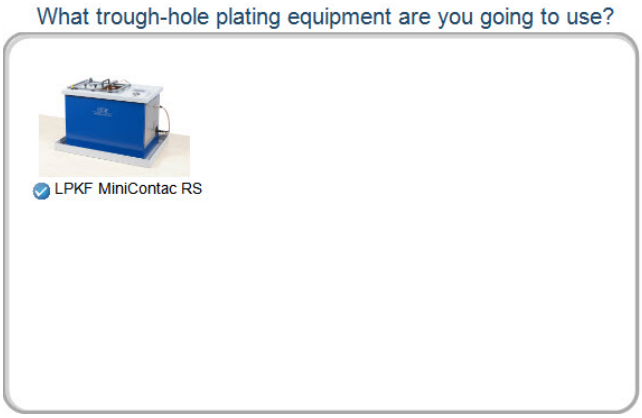
   c. Identify what material you are using.



Note: Currently the only settings available are for FR-4 material. The data for other materials will be included in later revisions for Circuit Pro

Author: LPKF/Chris Ware

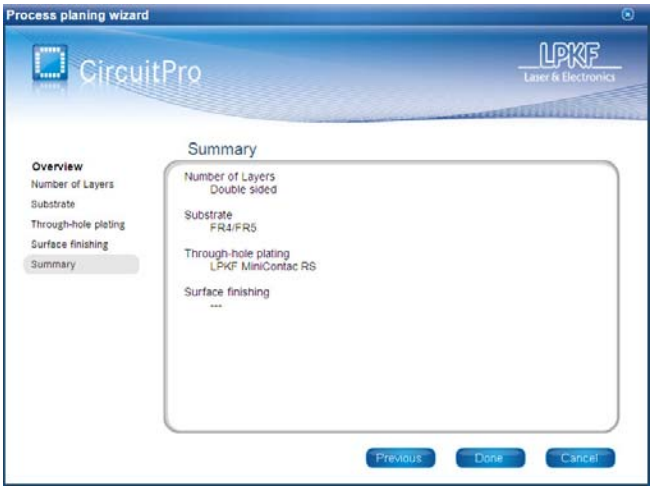
d. Identify which through hole plating method will be used.



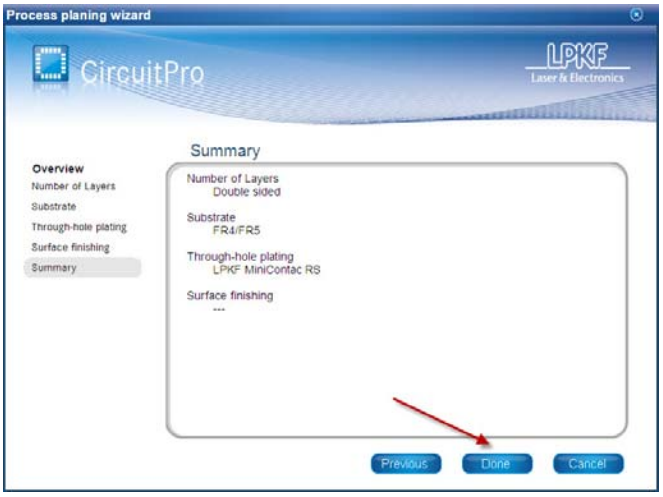
Note: The options will be presented based on what you told the system during the initial installation

e. Identify if you are using any surface finishing (Silkscreen or Soldermask)

f. Verify the configuration.



g. Press done.



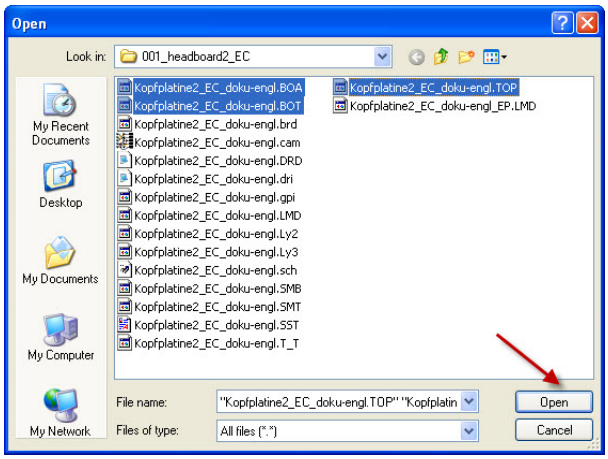
3. Import Gerber files

a. Press the import icon  , or go to the file menu, then import.

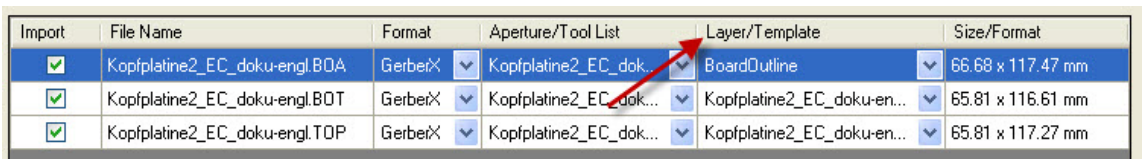
b. Find the location that has your Gerber files.

c. Single click on the top layer. Hold the control key and single click on the board outline and bottom layers.

d. Press open.



e. Change the layers/template column to the desired layer you want each file to import to. (.BOT is bottom layer, .BOA is board outline, .TOP is top layer)



f. Press ok.

4. Import Drill file

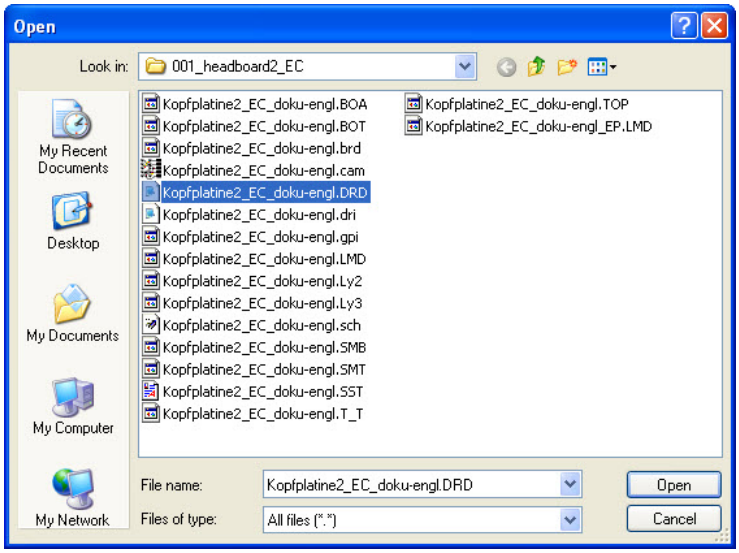
Note: This step can be completed at the same time as importing the Gerber layers.

a. Press the import icon  , or go to the file menu, then import.

b. Find the location that has your Gerber files.

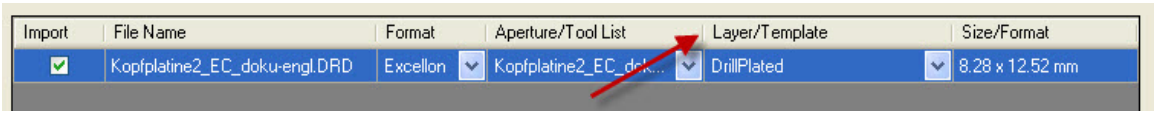
c. Single click on your NC-Drill file.

Note: In this document, the DRD file is the Excellon drill file. This may also be a text file, (.txt. extension) depending on your layout package.



d. Press open.

e. Change the laser/template column to drill plated or drill unplated.

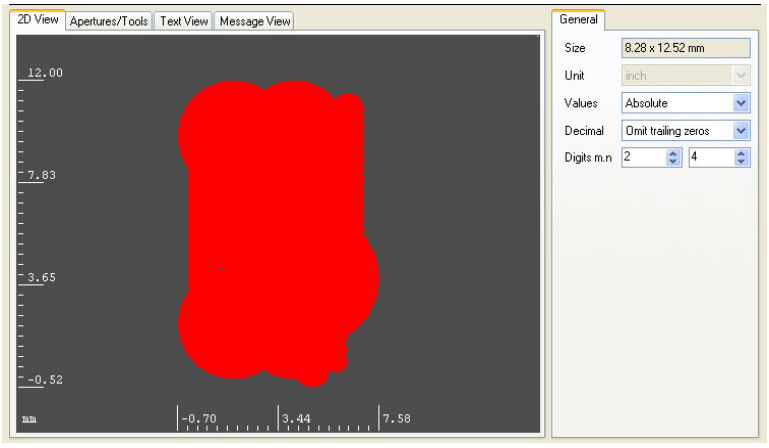


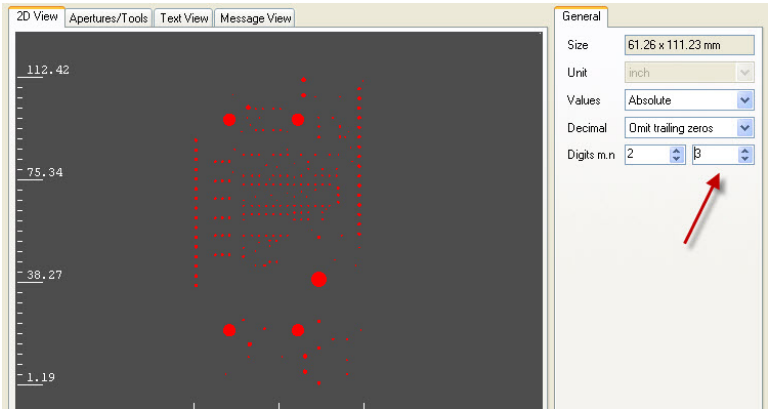
Note: This is determined if you are using a through-hole plating system. If so, use drill plated. If not, use drill unplated. This will also affect when the holes are drilled. Drill plated is drilled on the bottom side of the board. Drill unplated is drilled from the top side of the board. If this si a single sided board, use drill unplated.

f. In the graphic display, the correct view of the drills, you should see the different sized apertures and the correct location of your drills.

g. If the display is correct press open.

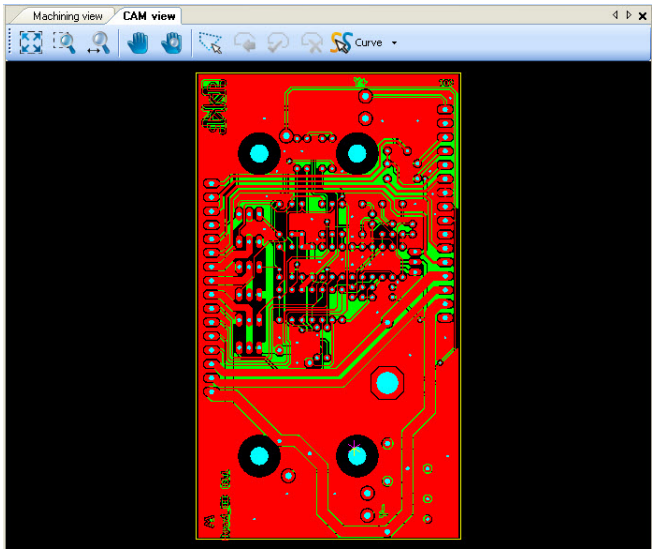
h. If the graphic display is not correct, change the N diget from 4 to 3. This signifies how many zeros the software uses. This setting is normally either 2,4 or 2.3.





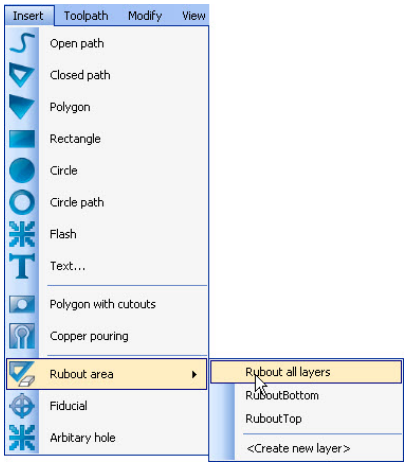
i. Press open.

j. Your design should be now displayed on screen.



5. Establish rubout boundary. (If removing excess copper is desired)

   a. Go to the insert menu, rubout area, rubout all layer.

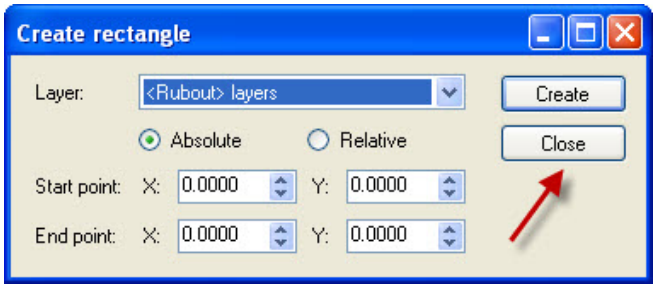


   b. Move the create rectangle window off to the side.

   c. Single click on one corner of the board, or desired location.

   d. Single click on the opposite corner of the board, or desired location

   e. Press the close button.

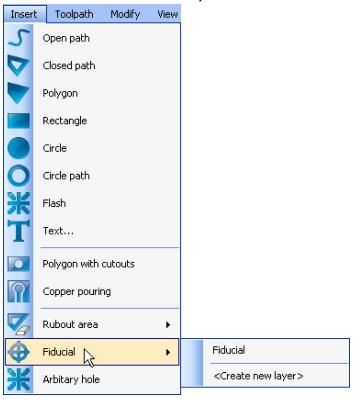


Note: You should see a rectangle outside of your board, or positions you selected. Circuit Pro will create your boundary on the rubout top and rubout bottom layer

Note: If you wish to create a rectangle by a specific size, you may use the coordinate system in the create rectangle window.

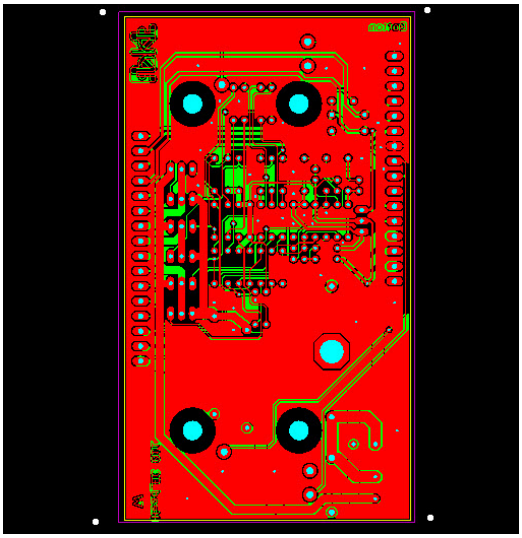
6. Insert Fiducials

   a. Go to the insert menu, select fiducial, then fiducial.



   b. Move the create circle window off to the side.

c. Left click where you want to place each fiducial hole. Best results are just off each corner of the board.



a. Press the X on the create circle window.

7. Create the tool pathways

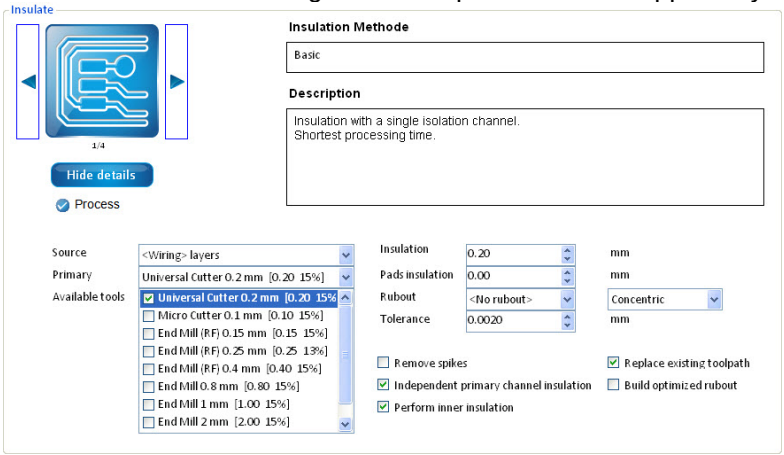
a. Press the technology dialogue icon  , or go to the toolpath menu and select technology dialogue.

b. Select insulation type.

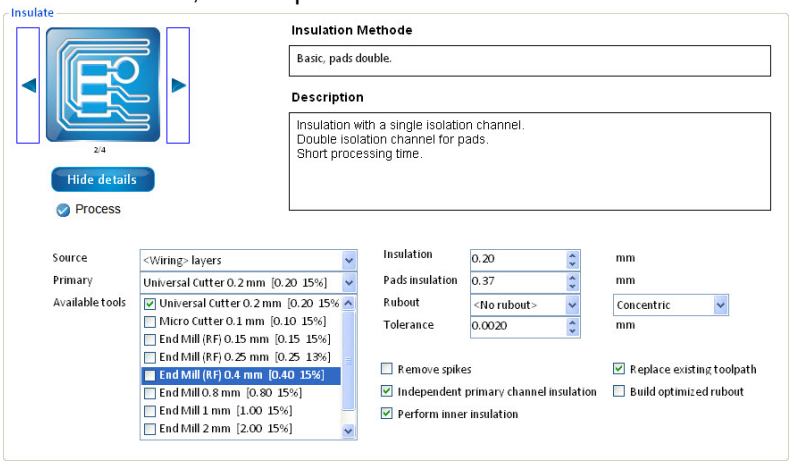
i. In the insulation process section, press the show details icon.

ii. Press the left arrow key on the screen to select which insulation type you desire.

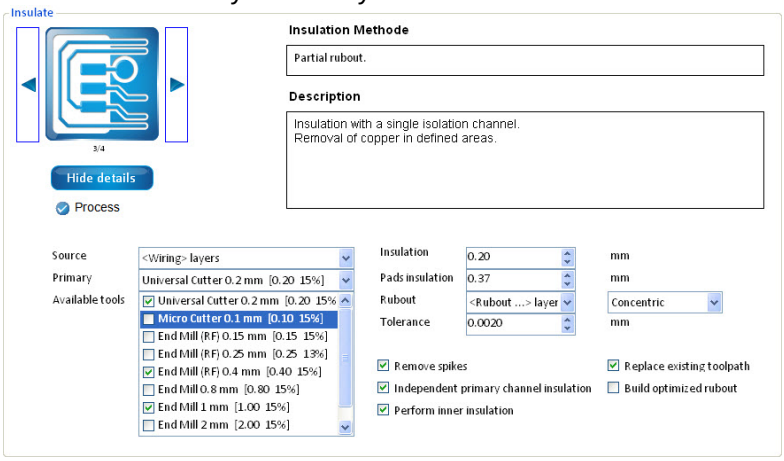
iii. Basic insulation is isolating the traced pads from the copper only.



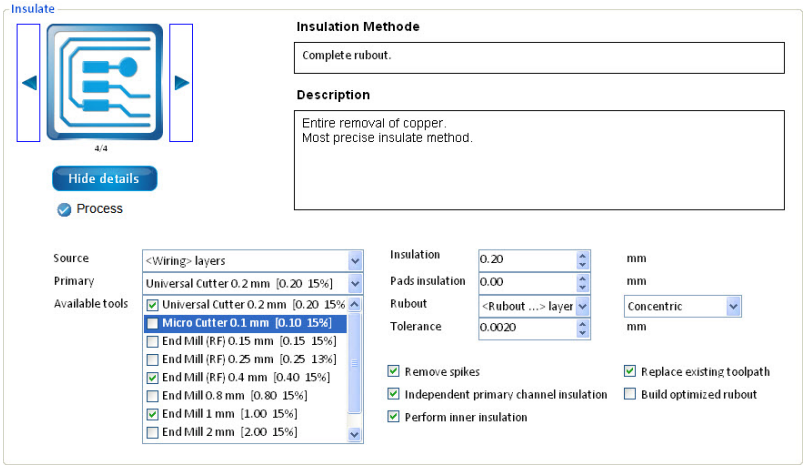
iv. Basic Insulation, double pads isolates the traces and double isolates the pads.



v. Partial rubout will isolate the traces and pads and remove any excess copper within the rubout boundary inside of your board.



vi. Full rubout will isolate the traces and pads, along with removing the excess copper from the entire board.



vii. Select the milling tools that you wish to use in the available tools drop down menu by checking/unchecking the tool name.
Note: The best combination of tools are: Universal Cutter 0.2mm, the 0.4mm End Mill and the 1.0mm End mill. For RF boards, use the 0.25mm End Mill instead of the Universal Cutter. If your designs have smaller spacing requirements than 8 mils, use the 0.15mm (6 mil) End Mill, the 0.1mm (4 mil) End mill or the 0.1mm (4 mil) Micro Cutter. If milling an RF board, change the primary tool

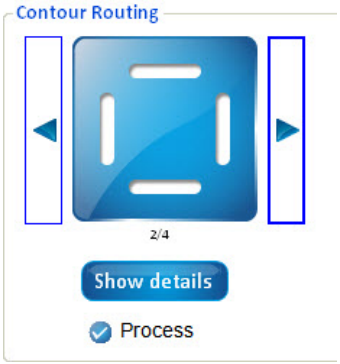
viii. Press the hide details button.

c. Select the contour routing type.

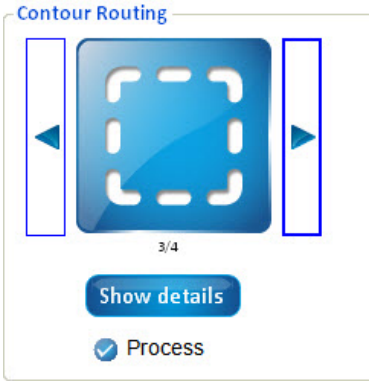
i. In the contour routing section, press the left arrow to select what option you would like to cut the board out.

1. Four edges. This will route along the outside of your board leaving tabs in the center of each edge of the board.

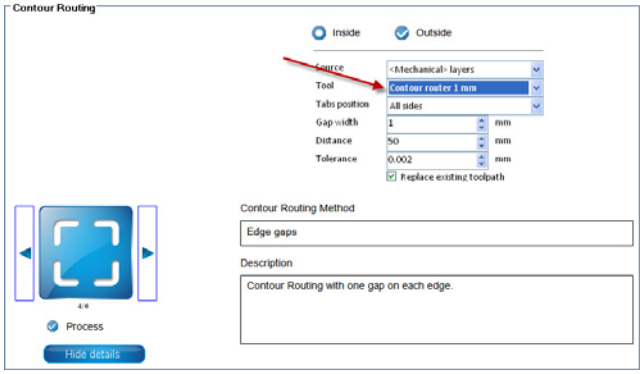
2. Four corners. This will route along the outside of your board leaving tabs in the corners.



3. Equidistant tabs. This will route along the outside of your board leaving tabs at equal distance that you specify.



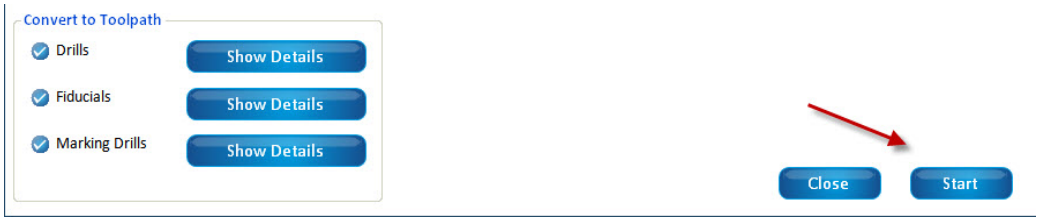
4. None. This will route along the outside of the board. No breakout tabs will be used.
5. If desired, change the tool used during the contour routing.
    a. In the used tool dropdown menu, change the 2.0mm contour router to the 1.0mm contour router.



Note: If your design is larger than 2 x 3 inches, the 2.0mm contour router is recommended. If your design is smaller than 2 x 3 inches, the 1.0mm contour router is recommended.

Note: The 1.0mm contour router is also recommended if the board outline is not a traditional rectangle. This helps minimize the risk of a rounded corner.

d. Press the start button.



d. The software will now create all tool paths and identify all which drills shall be used. When finished, you will get a report of which tools are used.