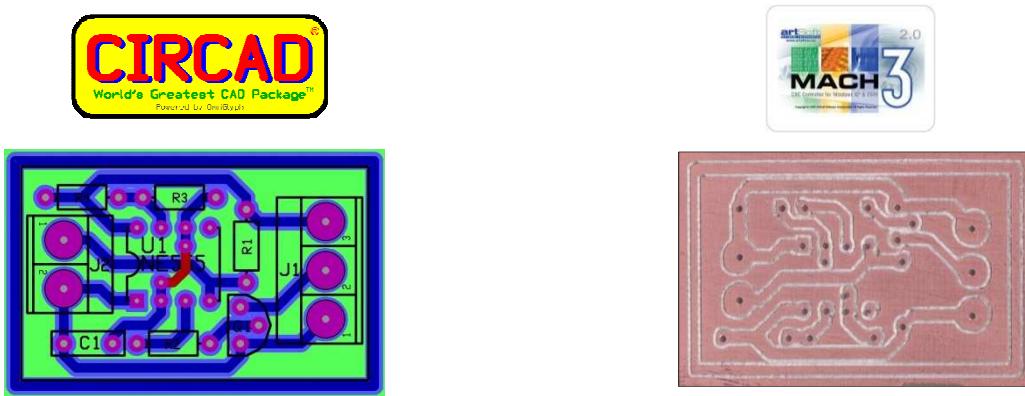


CirMach - Circad® to Mach3® Converter

Circad® / T-Tech® file to Mach3® file converter

Last revision: October 31, 2011



1. Introduction

This program converts files generated by Circad® (<http://www.holophase.com>) in HPGL format to be used by T-Tech® (<http://www.t-tech.com>) numerically controlled machines (CNC), using *G-Code* and *M-Code* commands interpreted by the program Mach3® (<http://www.machsupport.com>), allowing the production of printed circuit boards (PCBs) through the following steps: mechanical contouring of the electrical lines (engraving), elimination of copper areas (cleaning), cutting, drilling and, possibly, making of the panel of the PCB.

Circad® is available in a free trial version which lets you design, without limitation, the schematic and PCB of any electronic circuit and lacks, compared to the commercial version, only the export functions in special formats (eg. Gerber®, AutoCad®, etc.).

To learn how to use Circad®, I refer you to my course published by the (no longer printed) “Elettronica Flash” Italian magazine in 2004 in five episodes from February to June, now available for free (**only in Italian**) on the magazine's website (<http://www.elettronicaflash.it> [not always active]) and on my website (<http://www.tattik.it>) where the English speaking reader can also download the “Circad Lessons” in English.

The Circad® course refers to version 4.x of the program, available for download from the manufacturer (<http://www.holophase.com/dleval4.htm>) or from my site. Circad® is currently available in the version 5.x (called OmniGlyph ®) which is not supported by the program CirMach, and not by my Circad® course and has a file format that is incompatible with version 4.x.

2. How to make T-Tech® files

Before continuing, please read carefully the section "Circad® and CNC machines," of the fifth lesson of my Circad Course® (only in Italian) in order to familiarize yourself with the "Track Isolation" option and the concepts of contouring of the tracks and other concepts alike.

The making of T-Tech® files begins by opening any printed circuit created with Circad®. In this manual I will refer to the "CirMach_DemoPCB.pcb" example file, which is available on my website, if necessary, allow you to perform the operations herein described.

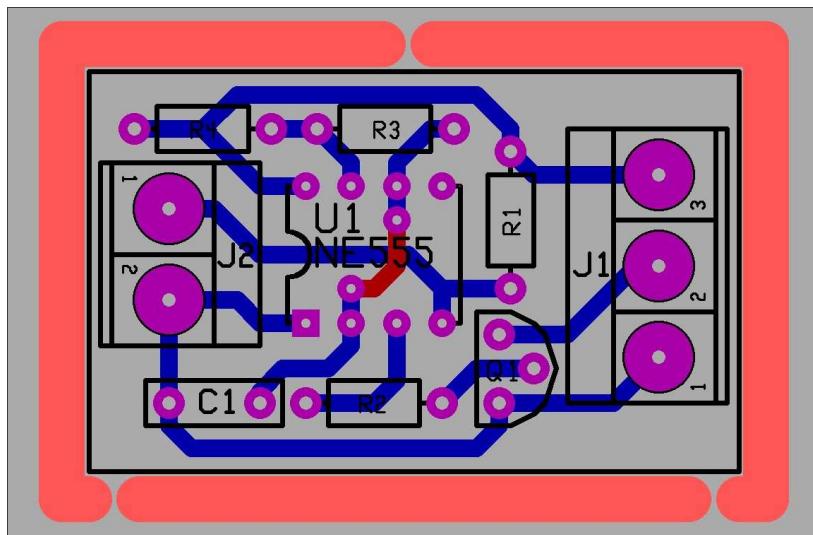


Figure 1 - The example "CirMach_DemoPCB.pcb" file opened by Circad®

The PCB contains all the elements needed to make them with a CNC:

- The thick **orange** lines that enclose the PCB represent the areas to be machined for cutting the contour of PCB itself. **The line thickness corresponds to the diameter of the CNC tool that will be used**, and the breaks in the boundary prevent the PCB from falling off the plate while it is being machined. When the process ends, you'll be able to remove the PCB by cutting the small areas that keep it bound to the rest of the plate. *If you don't want to cut out the PCB, you can omit those lines. Similar lines can be placed to create additional milling, for example to separate the PCB's high and low voltage circuit sections.*
- In **purple** you can see the pads of the components, which will be contoured during the engraving phase. Only these items will then be drilled during the drilling phase.
- In **blue** there are the traces that connect the components and that will be also contoured during the engraving phase.
- In **red** you can see a jumper, built as an electrical connection over the component side of the PCB, which will be ignored during the making of the T-Tech® files.
- In **black** there are the silk section of the PCB, which will be ignored when the T-Tech® files are created.

After opening the “*CirMach_DemoPCB.pcb*” file, you need to **move** the board as close as possible to the (“= inches) [**X=0.000” | Y=0.000”**] coordinates, corresponding to the lower-left corner of the yellow box that surrounds the PCB.

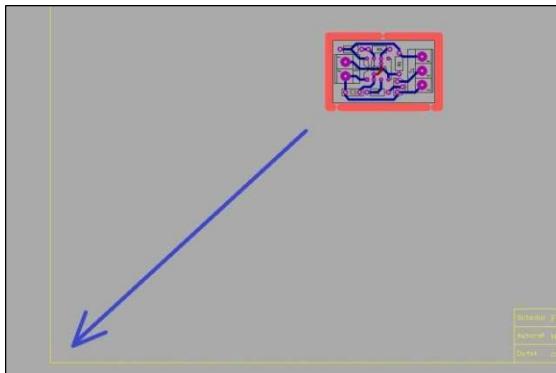


Figure 2 - PCB start position

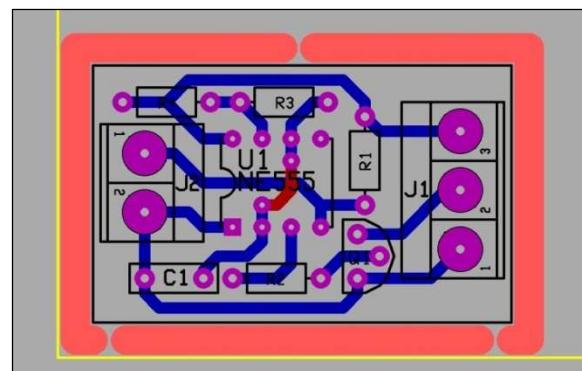


Figure 3 - The PCB after the move operation

You can change from absolute coordinates mode, required for the “move” operation, to the relative coordinates mode (look at the asterisk near to the Y coordinate) and vice versa, using cyclically the **[SP]** keyboard command, as seen on my “*Circad® Course - Lesson 1*”.

To make the “move” operation use the **[BG]** keyboard shortcut, and select a box that contains **all** of the PCB, including the **orange** lines, choosing as a “*source reference points*” the lower-left corner of the box; then with the **[BM]** keyboard command move the PCB in a position as close as possible to the final target coordinates [**X = 0,000” | Y = 0,000”**].

So, the two yellow lines, which originate from the point [**X=0.000” | Y=0.000”**], mark the work area that will be affected by CNC milling process.

Moving the PCB to this position ensures that, once Mach3® is started and set the absolute machine coordinate are set to [**X=0.000” | Y=0.000” (| Z=0.000”)**], the machining of the PCB will start exactly from that point, symmetrically with respect to the axis Y.

Before creating guidelines for engraving, cleaning and contouring the PCB, you need to know the specifications of the engraving mill bit used: the **shaft diameter (D)**, the **cut angle (A)** and the **cutting radius (R = t / 2)**.

For the best quality engraving use bit with a small cutting angle (10° -20°) and a small cutting radius (0.1-0.2 mm), but also drills with high cutting angle and radius can be used with excellent results.

Everything depends on the machining quality process required and on the characteristics of the CNC used to do this work.

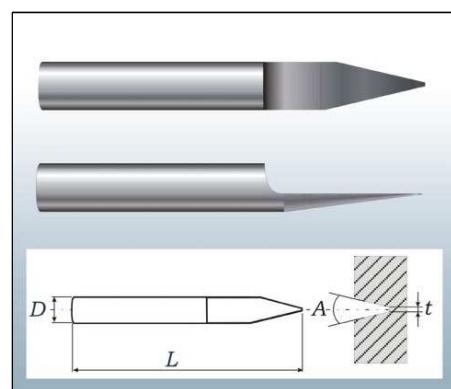


Figure 3 - Engraving mill bit

Note that a small cut angle limits the removal of traces of copper, while an equally small cutting radius increases the definition of the tracks, allowing you to cut out correctly even when the distance between a pair of them is very small.

| R (mm) | F (inches) | F (mils) |
|--------|------------|----------|
| 0,1 | 0,004 | 3,937 |
| 0,2 | 0,008 | 7,874 |
| 0,3 | 0,012 | 11,811 |
| 0,4 | 0,016 | 15,748 |
| 0,5 | 0,020 | 19,685 |
| 0,6 | 0,024 | 23,622 |
| 0,7 | 0,028 | 27,559 |
| 0,8 | 0,031 | 31,496 |
| 0,9 | 0,035 | 35,433 |
| 1,0 | 0,039 | 39,370 |

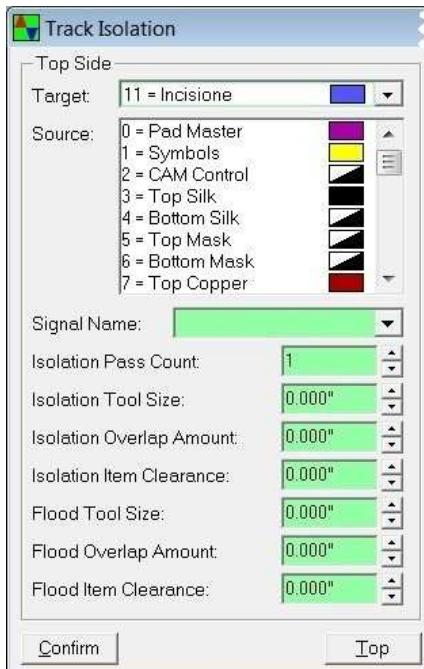
Multiply the value in millimeters of the cutting radius (R) by two and divide it by 25,4. This will give you a value, in inches, called F, which represents the minimum thickness of the contour path of the printed circuit board traces, and the minimum distance between two tracks that can be obtained using that mill bit.

For example, using a drill bit with **R = 0,1 mm** you will have **F = 0,004"**.

So, with the PCBs box still selected, use the keyboard command [BT] to access the configuration screen of the “*Track Isolation*” option, where you will configure only the “*Top Side*” section.

Table 1 - Conversion of R in F

This option, with the following configuration parameters must be used **twice**: once for the **engraving** phase (“*Incisione*” parameters column), and another for **cleaning** phase (“*Pulizia*” parameters column), while the data for contouring and drilling the PCB will be extracted later with the “*Isolation Output*” option.



| Parameters | Phase | |
|---------------------------------|-------------------------------------|-------------------------------------|
| | Incisione | Pulizia |
| <i>Target</i> | 11 = Incisione | 12 = Pulizia |
| <i>Source(*)</i> | 0 = Pad Master 8 = Bottom Copper | 0 = Pad Master 8 = Bottom Copper |
| <i>Signal Name</i> | leave blank | leave blank |
| <i>Isolation Pass Count</i> | 1 | 1 |
| <i>Isolation Tool Size</i> | F (eg. 0.004") | 0.000" |
| <i>Isolation Overlap Amount</i> | 0.000" | F/2 (eg. 0.002") |
| <i>Isolation Item Clearance</i> | 0.000" | 0.000" |
| <i>Flood Tool Size</i> | 0.000" | F (eg. 0.004") |
| <i>Flood Overlap Amount</i> | 0.000" | 0.000" |
| <i>Flood Item Clearance</i> | 0.000" | F/2 (eg. 0.002") |
| | ↓ Set and press “Top” | ↓ Set and press “Top” |

Figure 5 - The “Track Isolation” option

(*) You need to select the layers by clicking on their name, so they will go alternately from the selected state (**black** background) and not selected state (**white** background).

The result on the PCB will be that visible in **Figures 6 and 7**.

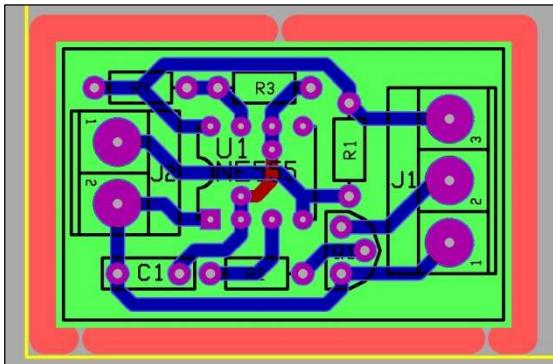


Figure 6 - Effect of the “Track Isolation” option



Figure 7 - Detailed view of engraving and cleaning tracks

So, on the “*Incisione*” layer will be created lines and arcs (in light blue on **Figures 6 and 7**) that surround all the items on the “*Bottom Copper*” layer (copper component side, in blue on **Figures 6 and 7**) and on the “*Pad Master*” layer (components pad, in purple on **Figures 6 and 7**). These lines and arcs will constitute the contour path to be followed by the CNC in the making of the printed circuit board.

The green area (**Figures 6 and 7**) consists by thin horizontal lines (**Figure 8**) that fill the entire area selected before using the “*Track Isolation*” option, without colliding with the elements on the layers above.

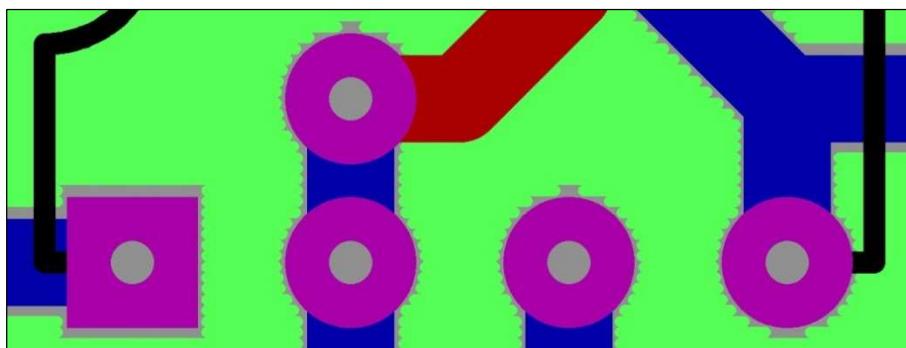


Figure 8 - Detailed view of horizontal lines used to clean the PCB

Note: the cleaning phase of the PCB (full engraving of the green area - **Figure 6, 7 and 8**), requires a large amount of time, because the CNC will cover every single area bounded by green lines on the layer “*Pulizia*”.

This cleaning step is not strictly necessary, if you do not use SMD components or when the mill bit used during engraving (light blue lines in **Figures 6 and 7**) allows you to make a good electrical insulation between the tracks and adjacent areas of copper.

There is an **alternative and faster method**, which lets you track a progressive series of contour lines of the printed circuit board traces, in order to reduce the final size of the residual copper engraved, while still providing a proper electrical isolation between the different connections.

To obtain these series of contour lines, without using a large diameter mill bit, that could damage the areas of the PCB in which the isolation required between different areas of copper is very small, set the option “*Track Isolation*” **once** with the options visible below, in order to achieve a single “*Incisione + Pulizia*” (engraving and cleaning) phase, instead of creating two distinct phases (copper engraving of the contour tracks and cleaning of rest areas)

These settings will create a series of 10 parallel paths of isolation (“*Isolation Pass Count*” parameter), gradually more distant from the track to be engraved out. These lines should be positioned to overlap partially with the previous ones (“*Isolation Overlap Amount*” parameter), in order to ensure the absence of residues of copper. These parameters are of course changeable depending on the type of work required and the characteristics of the mill bit used.

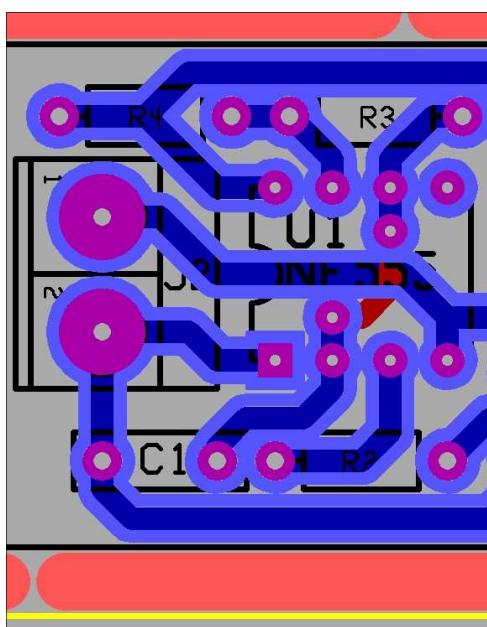


Figure 9 - Alternative method to clean the PCB

| Parameters | Phase Incisione+Pulizia |
|---------------------------------|-------------------------------------|
| <i>Target</i> | 11 = Incisione |
| <i>Source</i> | 0 = Pad Master 8 = Bottom Copper |
| <i>Signal Name</i> | leave blank |
| <i>Isolation Pass Count</i> | 10 |
| <i>Isolation Tool Size</i> | F (eg. 0.004") |
| <i>Isolation Overlap Amount</i> | F/2 (eg. 0.002") |
| <i>Isolation Item Clearance</i> | 0.000" |
| <i>Flood Tool Size</i> | 0.000" |
| <i>Flood Overlap Amount</i> | 0.000" |
| <i>Flood Item Clearance</i> | 0.000" |
| | ↓ Set and press “Top” |

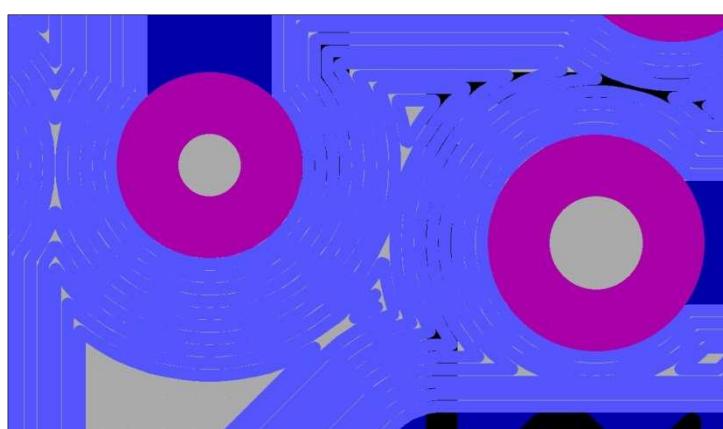


Figure 10 - Detailed view of the isolation paths (not overlapped)

Even if the visual effect is only one thick isolation path (**Figure 9**), this one is made by 10 overlapped paths.

By setting the parameter “*Isolation Overlap Amount*” to **zero**, you can see all the paths the CNC will follow during the single “*Incisione + Pulizia*” phase (**Figure 10**).

After using the “*Track Isolation*” option and getting the desired result, use the **[BE]** keyboard shortcut to clear the circuit board selection. Then, still with the **[BG]** keyboard shortcut, select it again, but this time also **including the orange lines** that surround the PCB.

Now you can create the T-Tech® file that will be converted later with CirMach, so use the option “*Output Isolation*” of the “*Block*” menu to enter the configuration screen, that you will set as shown in **Figure 11**.

For faster access to the files, I recommend you let the “.plt” files to be stored in the default folder “**C:\CAM**” which, if not existing, will be created automatically when you press the button “*Confirm*” on the screen option.

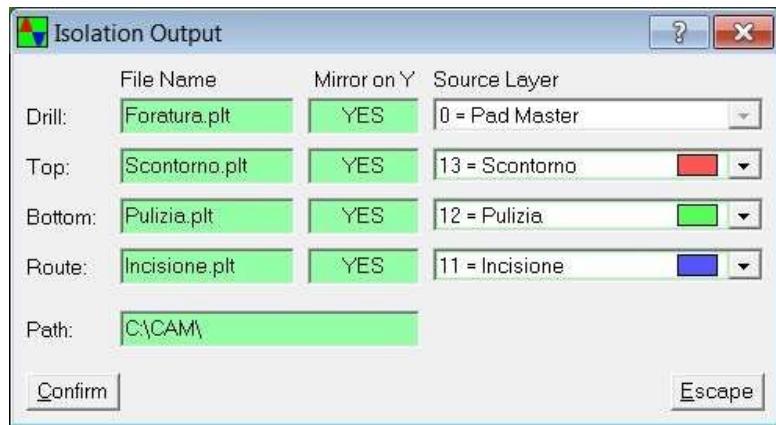


Figure 11 - Configuration of the “Isolation Output” option

Caution: for all the files the “*Mirror on Y*” field must be set to “YES”, because the PCB has to be worked on its copper side, which is specular to the one shown on video (seen from the component side).

To avoid errors during the construction of double-sided printed circuit boards with traditional components and/or SMDs components, you must be careful choosing which type of work is required from the CNC machine and, consequently, choosing the setting “*Mirror on Y*” fields.

For double-sided circuits, the “*Track Isolation*” and “*Isolation Output*” options will be used multiple times to create contour and cleaning paths, over the respective layers, and the related T-Tech® file.

3. How to convert T-Tech® file in Mach3® file

Files created with the “Isolation Output” option contain only **X** and **Y** coordinates of the points where the CNC machine will be moved, so they need the **Z** coordinate values that the CNC will reach during the different phases of processing, along with the CNC’s speed of movement.

CirMach can work with millimeters (mm), thousandths of an inch (mils) and inches (or “) units. The fields values can be entered manually in the field boxes and/or changed by the arrows next to the controls. When the values are entered manually, they will be always recalculated and eventually rounded up to 1 mils, which is the minimum value interpretable by Mach3® (**1 mils = 0,001 inch = 0,0254 mm**).

The screen default values are those typically used for perform mechanical works on a standard PCB with a thickness of 1.6 mm. Any changed value will be automatically stored on program exit and restored on the next reboot.

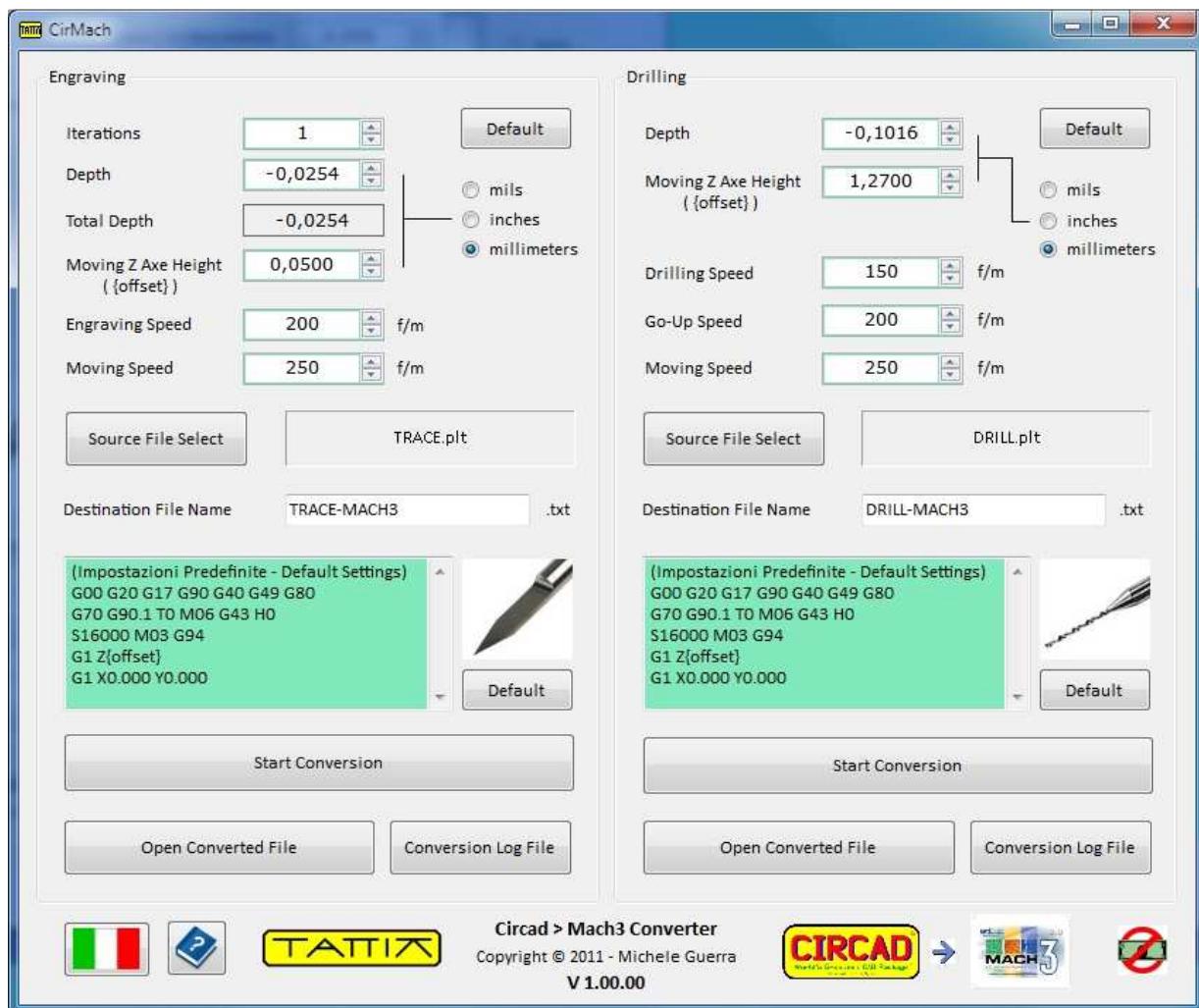


Figure 12 – Main screen of CirMach

3.a. Engraving phase (*Isolation, cleaning, PCB cutting*)

In the “*Depth*” field of the “*Engraving*” section, set the depth (negative height) that the CNC machine will reach during the engraving phase of the printed circuit board. Consider that the position [Z=0.000”] means that the mill bit mounted on the CNC is exactly in contact with the PCB.

To avoid partial engraving due to various factors (curvature of the PCB, not perfect flatness of surface, etc.), you can specify how many times the engraving loop will be repeated. At each subsequent iteration the mill bit of the CNC will be lowered by the initial value of depth.

The “*Moving Z Axe Height*” field, represents the height of the CNC mill bit during the switch operation between the different areas to be milled. During the conversion, the value of this parameter will also replace the words “{offset}” in the box below, which contains specific (and editable) settings for Mach3®.

The “*Engraving Speed*” and “*Moving Speed*” fields represent, in “*feed per minute*” the base speed (100%) at which the instructions will be sent to the CNC by Mach3®. The value of these parameters depends on the type of media used and the mill bit specifications.

These speeds can be changed later (from -90% to +300%) by Mach3®, acting within the “*FRO*” controls (“*Feed Rate Output*”), even dynamically during processing.

Choose the file to convert by pressing the “*Source File Select*” button. After the selection, the program automatically assign a name for the converted file, which can be manually changed editing the “*Destination File Name*” field. The extension of the file can't be modified and will be always “.txt”.

If you want to run the three different phases of engraving, cleaning and contouring the PCB (“*Incisione*”, “*Pulizia*” and “*Scontorno*”) you must load and convert three related files (“*Incisione.plt*”, “*Pulizia.plt*” e “*Scontorno.plt*”) previously generated with Circad® using the “*Isolation Output*” option.

To convert the format T-Tech® file in a Mach3® file, press the “*Start Conversion*” button. Once the conversion is finished and no errors appears, press the button “*Open Converted File*” to open the converted file and, if you want, include special settings required by Mach3® program.

On other hand, pressing the “*Conversion Log File*” button will display the source list of T-Tech® commands and those converted in Mach3® format. This file is for information only and although it can be changed, it will not be used by Mach3® for milling.

3.b. Drilling phase

The “*Drilling*” section must be set and used in the same way as see for the “*Engraving*” section. The settings are slightly different because in this case it is necessary to control the speeds at which the drilling bit will move during drilling phase and after it. You must use the “*Foratura.plt*” file, previously generated by Circad® with the “*Isolation Output*” option, as the source file to be converted.

4. Using the file with Mach3®

This guide doesn't look in detail at the configuration and use of the Mach3® program, which is deeply covered, even on the web, by excellent guides and manuals.

The files converted by CirMach can be immediately used with Mach3® and include the change cutter M-code ("M06 - Tool Change"). So, during operation, processing will stop each time the file requires to change the CNC milling, and the spindle will be temporarily switched off to allow you to replace the drill itself.

The number of the tool that you will be asked to use reflects the thickness of lines and arcs of isolation paths, or the holes diameters on the PCB. So, for example, when Mach3® prompts you to use the tool number 20, it means that you will use the one with a diameter of **20 mils**.

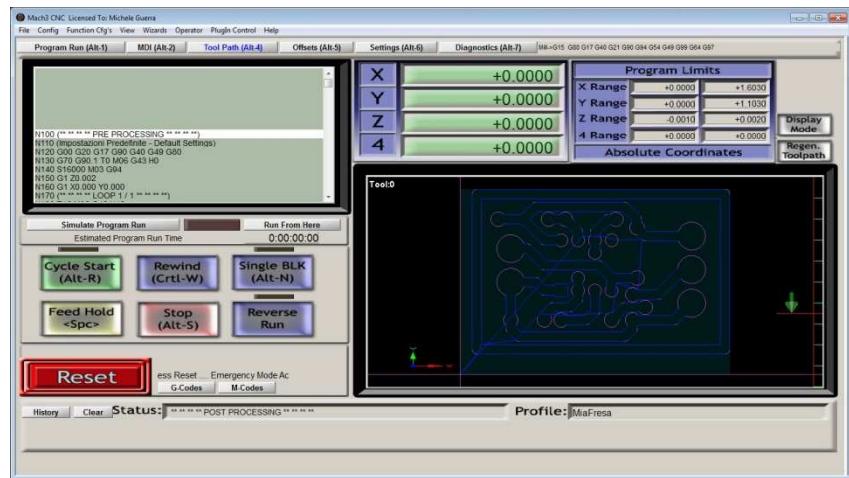


Figure 13 - The "Tool Path" screen of Mach3®

| Tool (mils) | Ø (mm) |
|-------------|--------|-------------|--------|-------------|--------|-------------|--------|-------------|--------|-------------|--------|-------------|--------|
| 1 | 0,0254 | 15 | 0,381 | 65 | 1,651 | 115 | 2,921 | 165 | 4,191 | 215 | 5,461 | | |
| 2 | 0,0508 | 20 | 0,508 | 70 | 1,778 | 120 | 3,048 | 170 | 4,318 | 220 | 5,588 | | |
| 3 | 0,0762 | 25 | 0,635 | 75 | 1,905 | 125 | 3,175 | 175 | 4,445 | 225 | 5,715 | | |
| 4 | 0,1016 | 30 | 0,762 | 80 | 2,032 | 130 | 3,302 | 180 | 4,572 | 230 | 5,842 | | |
| 5 | 0,1270 | 35 | 0,889 | 85 | 2,159 | 135 | 3,429 | 185 | 4,699 | 235 | 5,969 | | |
| 6 | 0,1524 | 40 | 1,016 | 90 | 2,286 | 140 | 3,556 | 190 | 4,826 | 240 | 6,096 | | |
| 7 | 0,1778 | 45 | 1,143 | 95 | 2,413 | 145 | 3,683 | 195 | 4,953 | 245 | 6,223 | | |
| 8 | 0,2032 | 50 | 1,270 | 100 | 2,540 | 150 | 3,810 | 200 | 5,080 | 250 | 6,350 | | |
| 9 | 0,2286 | 55 | 1,397 | 105 | 2,667 | 155 | 3,937 | 205 | 5,207 | 255 | 6,477 | | |
| 10 | 0,2540 | 60 | 1,524 | 110 | 2,794 | 160 | 4,064 | 210 | 5,334 | >255 | (*) | | |

Table 2 - Number of the tool and drill bit diameter in mm (highlighted: most near common value in mm)

(*) Caution: because the tool number can be only between 0 and 255, CirMach will tell you to always use the tool 255 also for holes over 255 mils. In this case, the user will choose which is the right tool to be mounted on the CNC.

Mach3® allows you to configure each tool for a specific “offset” (working height), but to simplify things, once the mill bit is inserted in the CNC, using commands to manually move the CNC (keyboard or codes “G01” on the command bar of the box “MDI”), move the mill bit itself to slightly touch the PCB until you reach the [X | Y] coordinates where the CNC will start working.

Now press the “Zero X”, “Zero Y” and “Zero Z” buttons in the “Program Run” box, to set the absolute coordinates of the machine at [X=0.000 | Y=0.000 | Z=0.000] and starts the CNC.

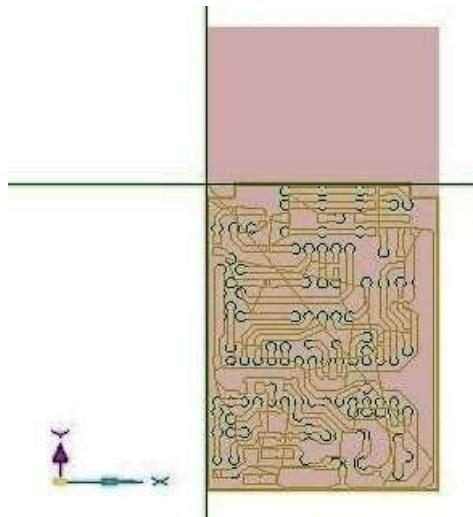


Figure 14 - PCB tool-paths on Mach3®
 (changed colors)

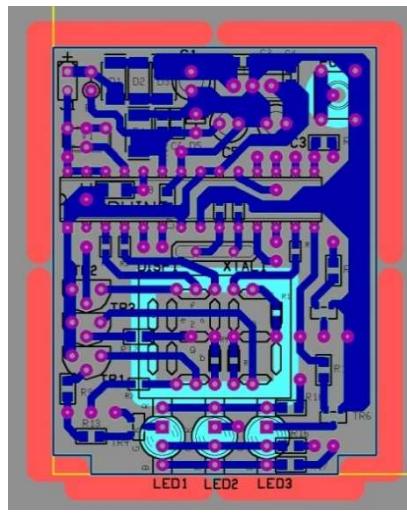


Figure 15 - PCB with SMD components
 and light blue lines and circles to make
 the circuit panel

The symmetry of the Y-axis machining can be seen comparing the circuit as displayed in Circad® and the engraving path displayed under the “Tool Path” screen of Mach3® (**Figure 14**).

The PCB in **Figure 15** use an additional layer (light blue) to define the areas that will be milled during the making of the front panel of the circuit board.

In fact, to create a circuit panel, just as you did to contour the PCB, you need to place lines and/or arcs on a specific layer, using the same size of the CNC mill bit; then using only the “Isolation Output” option of Circad®, you can generate the files that will be converted by CirMach and, therefore, opened with Mach3® to make the panel.

Note: CirMach is provided free and “AS IS”, without any express or implied warranty regarding its operation. We accept no responsibility for any damage to property and/or injury resulting from use or misuse of the program. The program is covered by international Laws regarding protection of intellectual property, but can be freely distributed. The publication of this manual, by any means and in a partial form, is prohibited without the written permission of the author. All trademarks are the property of their respective owners. All rights reserved. Copyright © 2011.

Michele Guerra - michele.guerra72@gmail.com - <http://www.tattik.it>



Thanks to ArtSoft - Newfangled Solution LLC (USA) to permit me to include the Mach3® icon on CirMach screen and manual.

“There was never a genius without a little madness” ~ Seneca