

PCB Prototyping Software

PHCNC LSSE

v6.6.3

User's Manual

www.telenet-eu.com

© 2007-2026, TELENET LTD.

Aug 15, 2026

Table of Contents

1. PhCNC LSSE Basics	6
1.1 First installation of PhCNC LSSE	7
1.2 Upgrade, Repair and Removal of PhCNC LSSE	8
1.3 PhCNC LSSE Software License Activation	9
1.3.1 Online activation	10
1.3.2 Offline activation	11
1.3.3 License persistence and reinstallation	12
1.4 Machine parameters' setup	12
2. CAM mode	14
2.1 Gerber/Drill Import	14
2.1.1 Window for Gerber/Drill files import	14
2.1.2 Gerber/Drill files import	15
2.1.3 "Gerber View" window	20
2.1.4 Profiles for automated Gerber & Excellon import	21
2.1.5 Gerber RS 274-D files import	24
2.1.6 Gerber files for Stencils Import	27
2.1.7 Convert Gerber to Drill	31
2.1.8 Info layers import	32
2.1.9 Gerber editor	33
2.1.10 Single layer PCB import	34
2.2 PhCNC LSSE Project Files	36
2.3 Buttons and fields used in CAM mode	36
2.4 Tools	41
2.4.1 "Insulate" Tools	43
2.4.2 "Cut" Tool	47
2.4.3 "Rubout" Tools	48
2.4.4 "Drill" Tool Table	55
2.4.5 "Stencil" Tool (Pro version only)	56
2.4.6 Project Tools	56
2.5 Object selection mode	58
2.6 Move Objects	59
2.6.1 Moving objects using the mouse	59
2.6.2 Move using "Rotate & Move Selected Objects ..."	61
2.6.3 Move using "Move To ..."	61
2.7 Clone (Copy) objects	62
2.8 Rotate objects	63
2.9 Group/ungroup objects	64
2.10 Delete objects	65
2.11 Rubout – excessive copper removal	65
2.12 Editor for the mechanical layer	68
2.12.1 Contour cutting selections	68
2.12.2 Contours selection in the mechanical layer	69
2.12.3 Contours generation with "PhCNC LSSE Printer"	69
2.12.4 Generation of contours with ("Insert Text")	72
2.13 Design Rules Violation (DRV) detector	73

2.14 Registration Pins and Thumbtacks	76
2.14.1 Thumbtacks.....	76
2.14.2 Registration Pins	76
2.15 Insulating copper rubout.....	79
2.16 ODB++ import.....	81
2.16.1 Automatic layer selection	81
2.16.2 Hierarchical panel processing	81
2.16.3 Advantages of ODB++ import.....	81
2.17 DXF import	82
2.17.1 Selecting DXF layers for "Cut" operation	83
2.17.2 Selecting DXF layers for Drill & Countersink.....	83
2.17.3 Selecting DXF layers for milling (MILL).....	84
2.18 Holes and pads insertion	86
2.18.1 Holes insertion.....	86
2.18.2 Through-hole pads insertion	86
2.18.3 Insert Target	87
2.19 Vector Text insertion function.....	87
2.20 Breakout Tabs.....	90
2.20.1 Breakout tabs insertion.....	91
2.20.2 Breakout tabs removal.....	93
2.20.3 Breakout tab width setting.....	93
2.20.4 Breakout tabs complete removal.	93
2.21 Units of measurement [mm]/[in]	93
2.22 UNDO/REDO	94
2.23 Forced Insulation.....	96
2.24 Remove spikes (IPD)	97
2.25 Measure	100
2.26 G-Code Export from PhCNC LSSE.....	101
2.27 Gerber Export	105
2.28 Print.....	107
2.29 Panelization	108
2.30 Marking Drill	111
2.31 Holes Selection / Modifications / Removal.....	113
2.32 QR Code.....	115
2.33 DataMatrix.....	116
2.34 Pick & Place Helper.....	118
3. Setup	121
3.1 Depth Options Setup	121
3.2 CNC process control settings.....	122
3.3 G-Code post-processor and machine compatibility settings	123
3.3.1 Tool change commands.....	123
3.3.2 Arc format.....	124
3.3.3 Modal G-Code output	124
3.3.4 G-Code units and line numbering	125
3.3.5 G-Code coordinate offset.....	125
3.3.6 CNC simulator options.....	126
3.3.7 G-Code header and footer	126

3.4 Display settings.....	127
3.5 Multithreading & PhCNC LSSE64.....	129
4. CNC mode	130
4.1 Manual controls.....	131
4.1.1 Spindle/camera positioning using Jog buttons or mouse.....	131
4.1.2 Home X & Y	132
4.1.3 Home Z.....	132
4.1.4 "X=0", "Y=0, "Z=0" Buttons	132
4.1.5 Spindle & Vacuum control.....	132
4.1.6 Feed Rate Override.....	133
4.1.7 Emergency Stop	133
4.1.8 Pause	133
4.1.9 Park.....	133
4.2 Program execution in CNC mode	133
4.2.1 Control buttons for G-Code programs execution	138
4.2.2 Drawing control buttons in CNC mode	139
4.2.3 G-Code editor menu.....	142
4.3. Machining Window	142
4.4 Regions Selection	145
5. In case of further questions.....	146

1. PhCNC LSSE Basics

This manual applies to both PhCNC LSSE and PhCNC LSSE HD. The two products provide the same CAM functionality and differ only in their licensing method.

For simplicity, unless explicitly stated otherwise, references in this manual to PhCNC or PhCNC LSSE apply equally to both PhCNC LSSE and PhCNC LSSE HD.

This manual explains how to use the different features of PhCNC LSSE. PhCNC LSSE is the result of almost 20 years of continuous development of the PhCNC software platform.

The licensing methods are as follows:

PhCNC LSSE requires a software license assigned to the computer on which the software is installed. A valid software license unlocks the production output functions.

PhCNC LSSE HD requires a valid USB hardware dongle (PhCNC LSSE.HD) connected to the computer. The dongle unlocks the production output functions.

Without a valid software license or connected hardware dongle, both products operate in Evaluation Mode. Evaluation Mode provides access to almost all CAM, project-editing, toolpath-generation and simulation functions, but disables the following production output operations:

G-code export;
Gerber and drill-file export;
Printing.

Projects can still be created, edited, saved and reopened in Evaluation Mode.

Software Versions and Windows Compatibility

PhCNC LSSE is available in both 32-bit and 64-bit versions. All versions are compatible with Windows User Account Control (UAC) and can be installed, updated and uninstalled through the standard Windows installation and application-management tools.

PhCNC LSSE 32-bit (x86) can be installed on Windows XP, Windows Vista, Windows 7, Windows 8.x, Windows 10 and Windows 11.

PhCNC64 LSSE 64-bit (x64) requires a 64-bit edition of Windows and can be installed on 64-bit Windows 7, Windows 8.x, Windows 10 and Windows 11.

PhCNC LSSE Licensing

Online licensing of PhCNC LSSE is available on computers running Windows 7 or later and requires an active Internet connection.

Offline licensing is available on all supported Windows versions. The license request can be generated on the PhCNC LSSE computer and processed using another Internet-connected computer.

PhCNC LSSE HD USB Dongle

PhCNC LSSE HD uses a USB hardware dongle instead of a computer-bound software license.

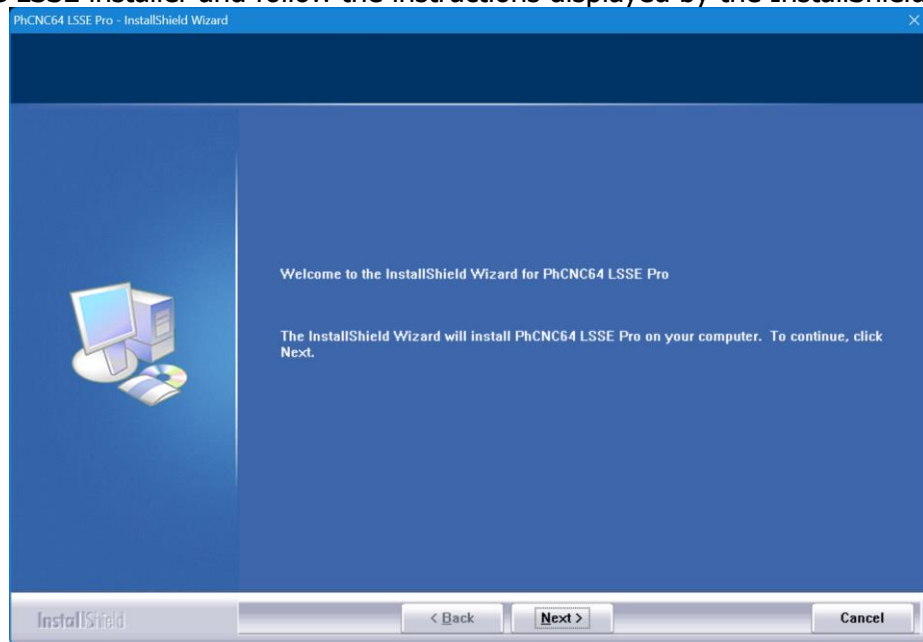
The required USB dongle drivers are installed automatically during the PhCNC LSSE HD installation. The drivers are digitally signed and comply with the Microsoft Windows driver-signing and security requirements supported by the corresponding Windows versions.

A valid USB dongle must remain connected to the computer to enable the production output functions. If the dongle is missing or not recognized, PhCNC LSSE HD operates in Evaluation Mode.

1.1 First installation of PhCNC LSSE

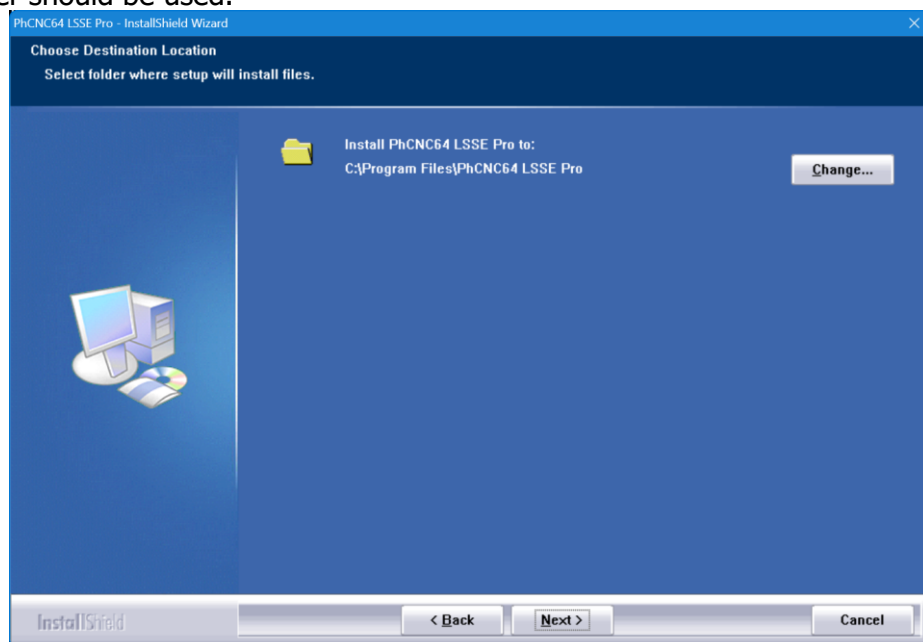
PhCNC LSSE is installed in the same way as a standard Windows desktop application. The screenshots below show the main steps of a first-time installation.

Start the PhCNC LSSE installer and follow the instructions displayed by the InstallShield Wizard.

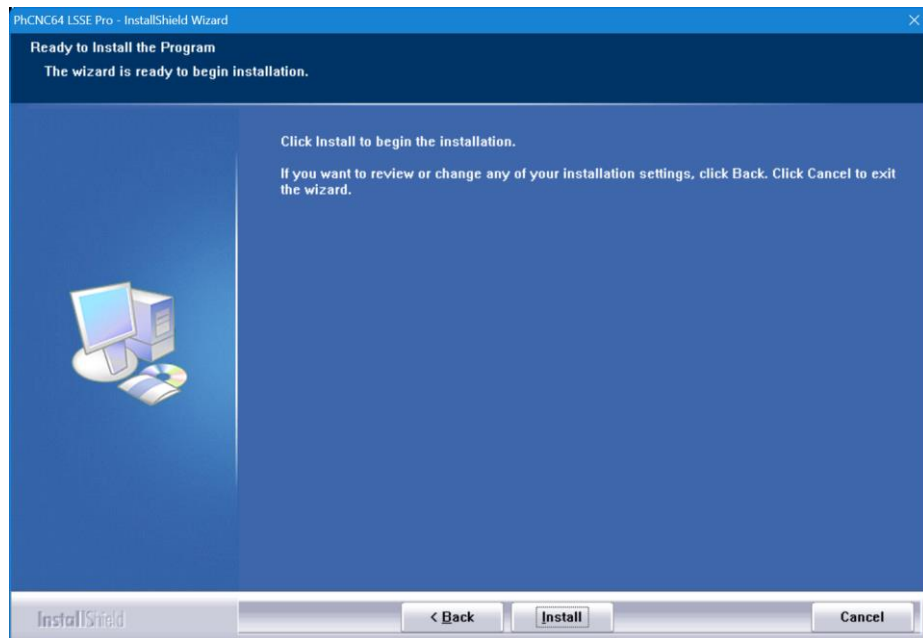


Read the End-User License Agreement and accept its terms to continue with the installation.

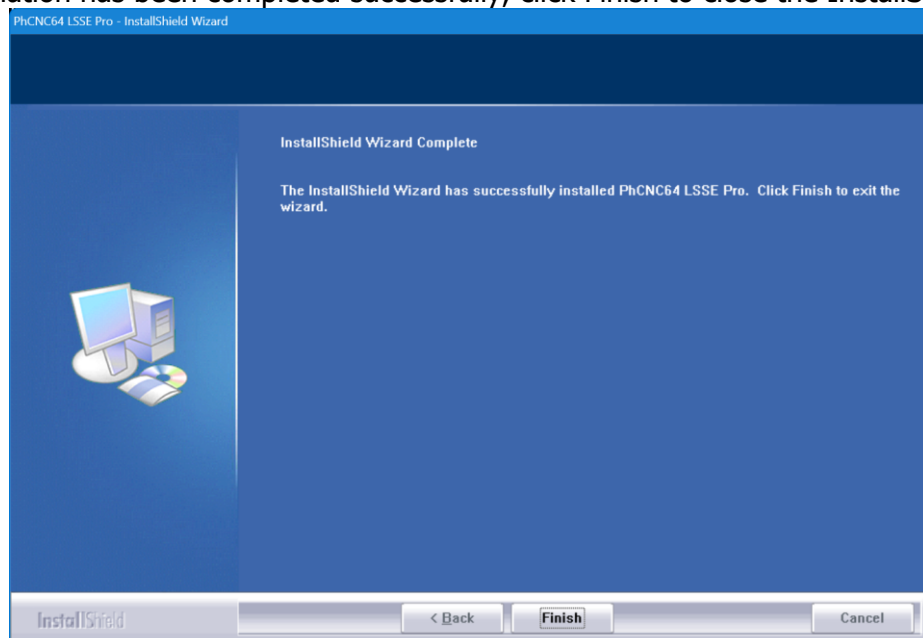
The installer allows the installation folder to be selected before installation begins. Normally, the default installation folder should be used.



Click Install to start the installation.



When the installation has been completed successfully, click Finish to close the InstallShield Wizard.

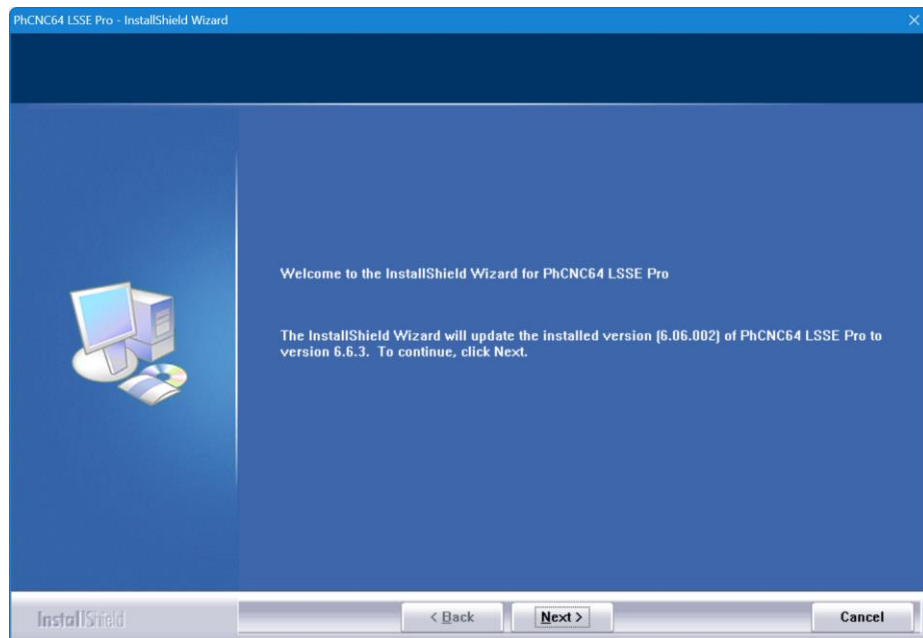


PhCNC LSSE is available in both 32-bit (x86) and 64-bit (x64) versions. PhCNC64 LSSE requires a 64-bit edition of Windows.

1.2 Upgrade, Repair and Removal of PhCNC LSSE

If an earlier version of PhCNC LSSE is already installed, starting a newer installer automatically starts the upgrade procedure.

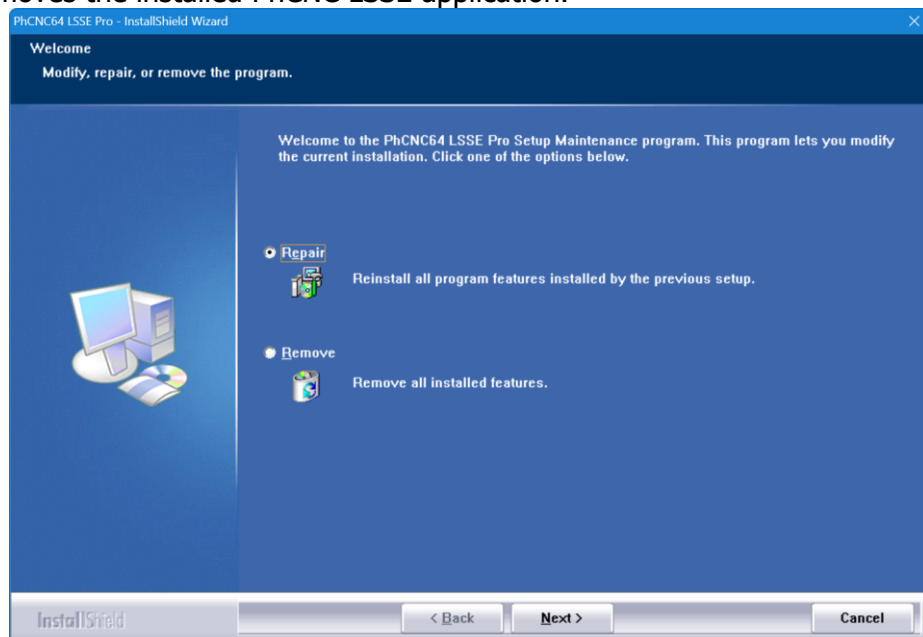
The installer displays the currently installed version and the version to which it will be upgraded. Follow the InstallShield Wizard instructions to complete the update.



Existing PhCNC LSSE settings and the installed software license are preserved during a normal software upgrade. No re-activation is required.

If the installer contains the same version as the currently installed version, the maintenance dialog provides the following options:

- Repair - reinstalls the program files of the existing installation.
- Remove - removes the installed PhCNC LSSE application.



If you need to install a version that is older than the currently installed version, first uninstall the newer version using the standard Windows application-management tools and then install the required older version.

Uninstalling PhCNC LSSE does not remove the installed software license. If PhCNC LSSE is later reinstalled on the same computer, the existing license is detected automatically and re-activation is not required.

1.3 PhCNC LSSE Software License Activation

This section applies to PhCNC LSSE, which uses a computer-bound software license. PhCNC LSSE HD uses a USB hardware dongle and does not require software-license activation.

A valid PhCNC LSSE software license is required to enable the production output functions, including G-code export, Gerber/Drill export and printing.

Without a valid software license, PhCNC LSSE operates in Evaluation Mode. CAM processing, project editing, toolpath generation, CNC simulation and other evaluation functions remain available.

A PhCNC LSSE Activation Key can be purchased online from:

<https://www.telenet-eu.com/PhCNCLSSE.php>

After a successful purchase, the Activation Key is provided immediately and a copy is also sent to the email address specified during the purchase.

The Activation Key is used to generate and install a software license for the computer on which PhCNC LSSE will be used.

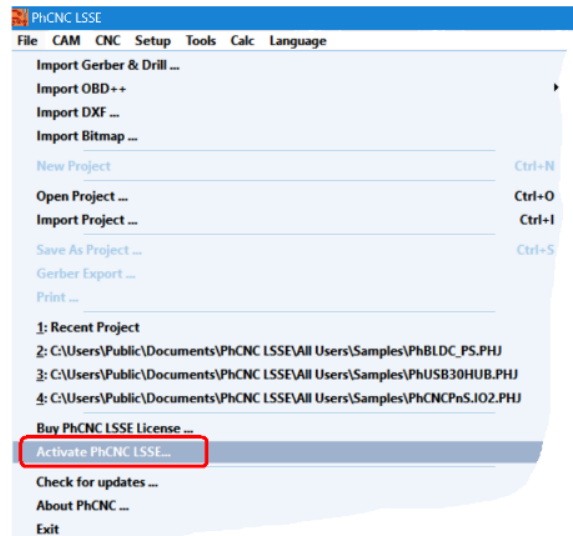
Important: Software-license activation is not supported on virtual machines. When PhCNC LSSE detects that it is running in a virtual machine, it operates in Evaluation Mode and the software-license activation options are not available.

1.3.1 Online activation

Online activation is the easiest way to install the PhCNC LSSE software license.

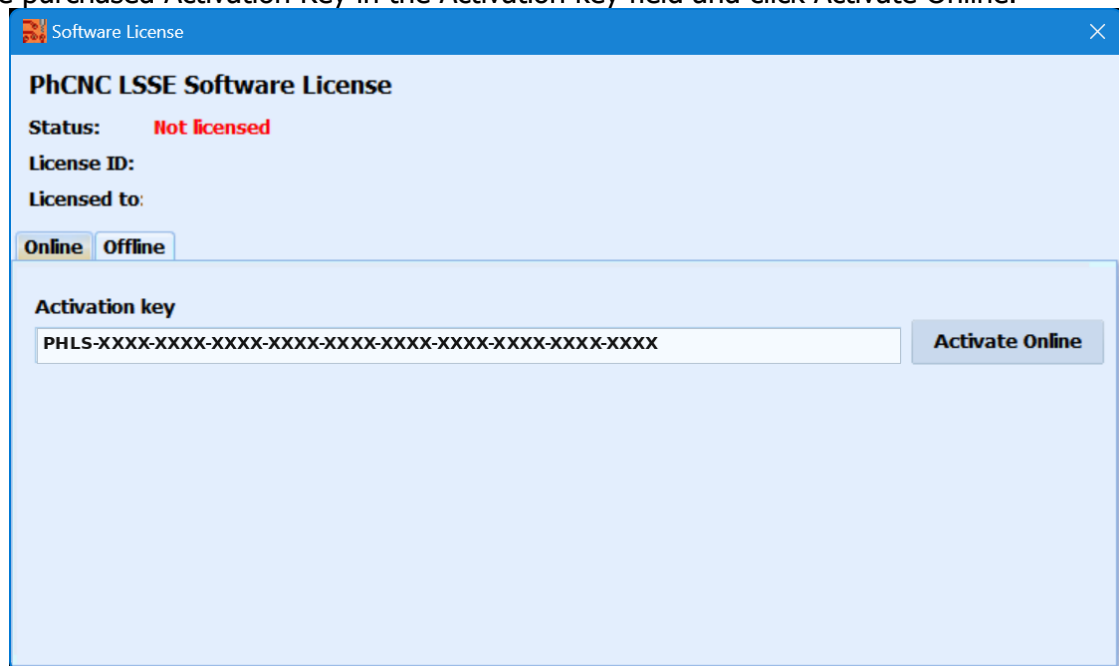
Online activation is available on computers running Windows 7 or later and requires an active Internet connection. PhCNC LSSE must be allowed to access the Internet and must not be blocked by Windows Firewall or other network-security software.

The File menu contains commands for purchasing a PhCNC LSSE license, activating the software and checking for software updates.



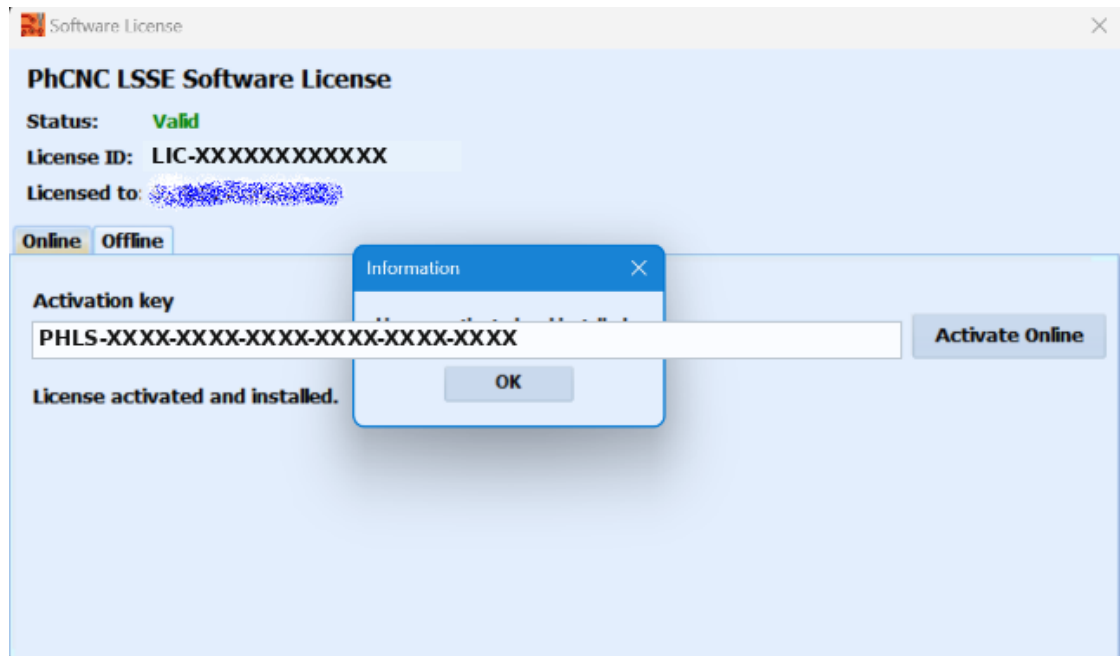
Select Activate PhCNC LSSE... to open the Software License window.

Enter the purchased Activation Key in the Activation key field and click Activate Online.



PhCNC LSSE sends the computer hardware information to the licensing server. If the Activation Key is valid, the corresponding software license is generated and installed automatically.

After successful activation, the Software License window displays the license status, License ID and licensed user information. The production output functions are then enabled.

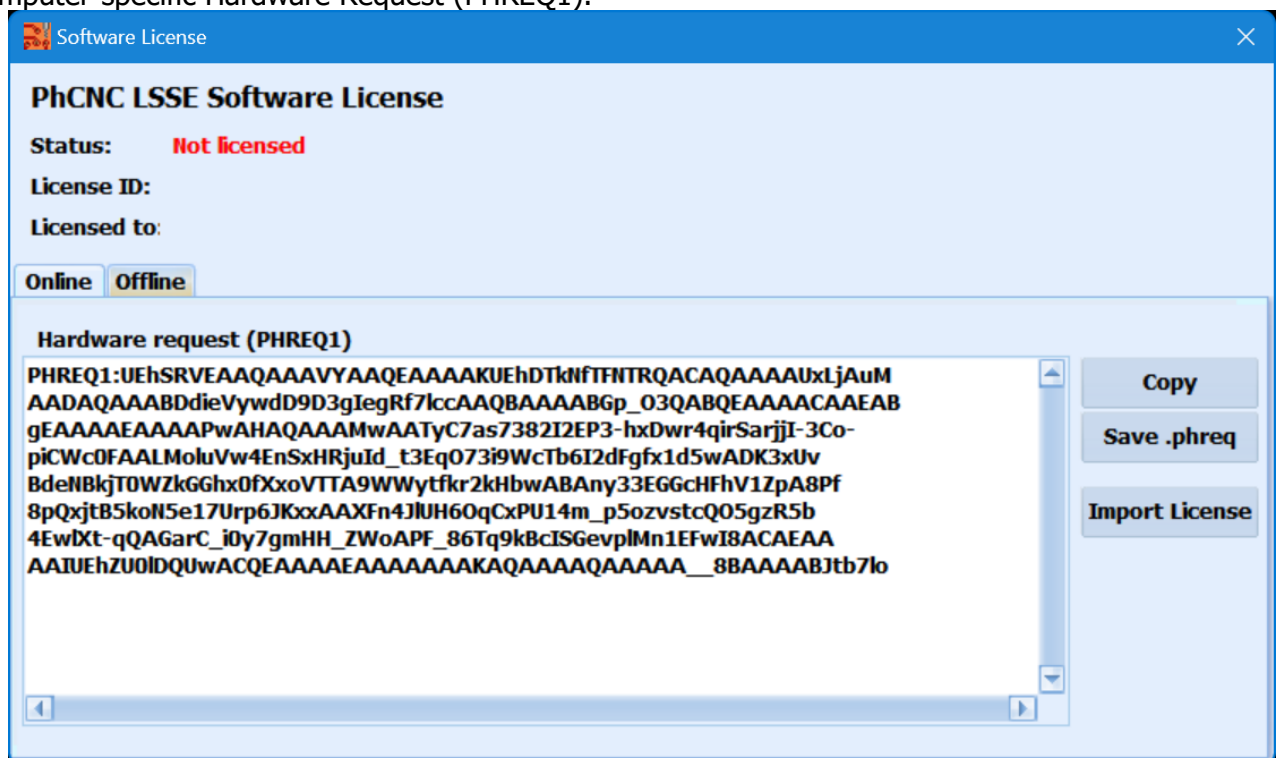


1.3.2 Offline activation

Offline activation can be used when the computer running PhCNC LSSE does not have Internet access. Offline activation is available on all supported Windows versions.

The activation request can be generated on the PhCNC LSSE computer and processed using another computer that has Internet access.

Open the PhCNC LSSE Software License window and select the Offline tab. The window displays the computer-specific Hardware Request (PHREQ1).



The Hardware Request can be copied to the clipboard using Copy or saved to a .phreq file using Save .phreq.

On an Internet-connected computer, open:

<https://telenet-eu.com/license/public/offline.php>

PhCNC LSSE Offline Activation

Activation key

Activation request

Activate

Enter the following information:

- the PhCNC LSSE Activation Key;
- the complete Hardware Request (PHREQ1) in the Activation request field.

Click Activate to generate and download the software license file. Transfer the downloaded license file to the PhCNC LSSE computer.

Open the Offline tab of the Software License window and click Import License. After the license has been imported successfully, PhCNC LSSE is licensed and the production output functions are enabled.

1.3.3 License persistence and reinstallation

The installed PhCNC LSSE software license is stored in:

C:\Users\Public\Documents\PhCNC LSSE\All Users

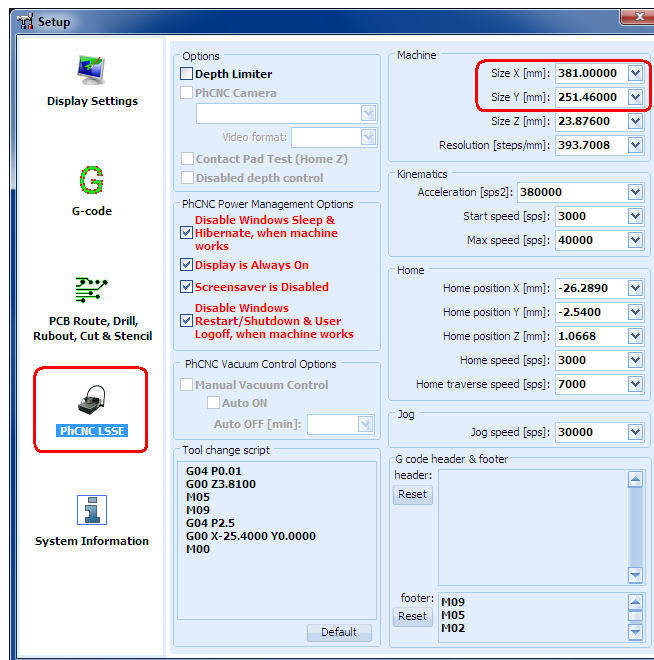
The license files stored in this folder are not removed or modified during a normal PhCNC LSSE upgrade, uninstall or reinstall.

Therefore, software re-activation is normally not required after upgrading, uninstalling or reinstalling PhCNC LSSE on the same computer.

If the license files are manually deleted or otherwise lost, the same Activation Key can be used again on the same computer to restore the license.

1.4 Machine parameters' setup

Before using PhCNC LSSE, it is strongly recommended that you enter the dimensions of the machine's working area. This can be done in the Setup window:

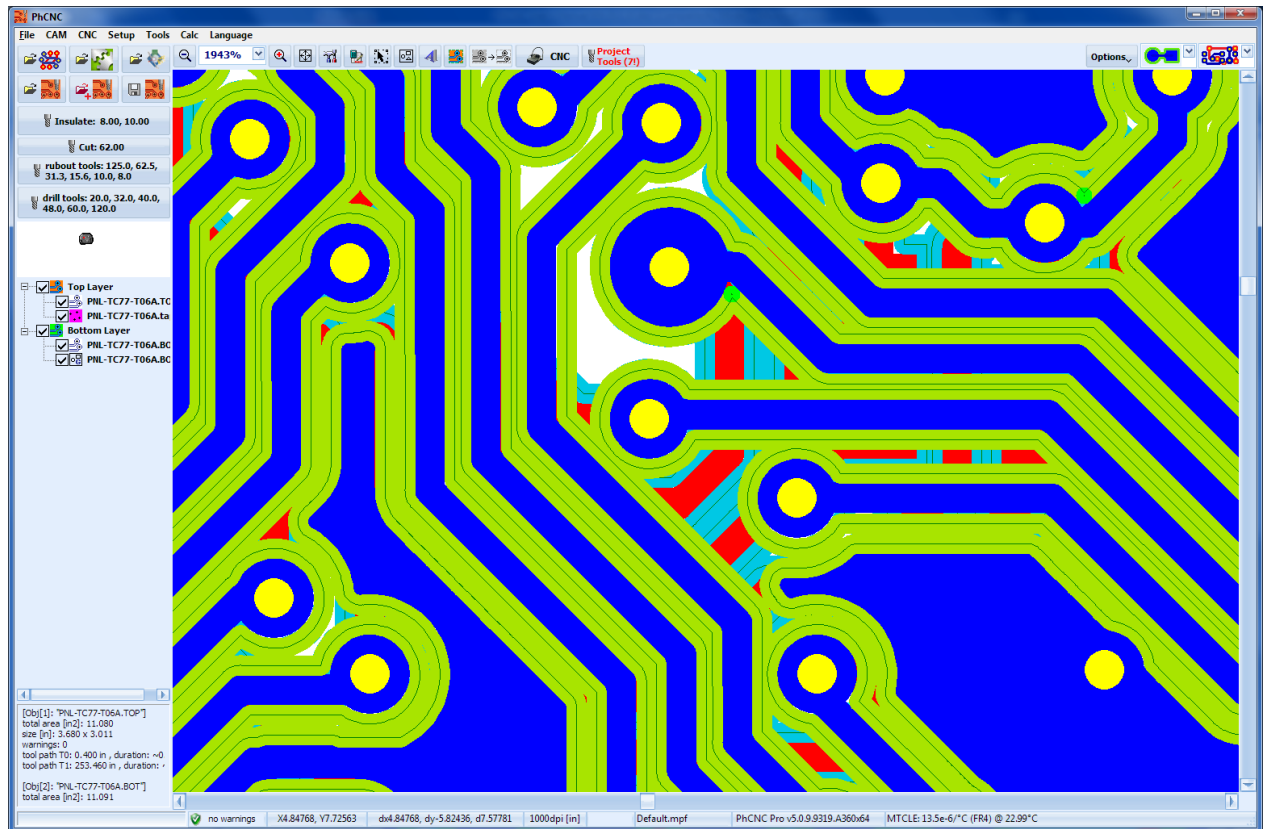


These parameters are particularly important when manufacturing double-layer PCBs.

The other parameters (Kinematics, Home, and Jog) are used to calculate the execution times of generated G-code programs. They are also used by the PhCNC LSSE CNC machine simulator.

2. CAM mode

When PhCNC LSSE starts, it opens in CAM mode. In this mode, you can load and edit CAM data in preparation for prototyping in CNC mode.



In CAM mode, you can perform the following operations:

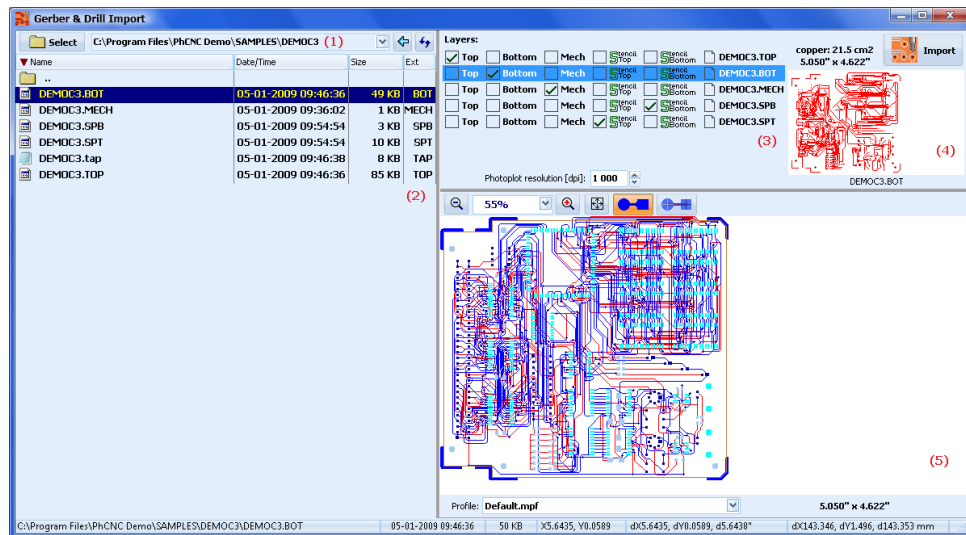
- Load one or more PCB designs
- Group more than one PCB in a single object
- Ungroup already grouped objects
- Clone (copy) objects or groups of objects
- Delete objects or groups of objects
- Select and deselect objects and groups
- Rotate selected objects or groups by an arbitrary angle
- Move selected objects or groups
- Visually check insulation tool paths
- Determine tools and regions for excessive copper rubout
- Select inner or outer tool diameter compensation for all cutting operations

2.1 Gerber/Drill Import

Current versions of PhCNC LSSE use a visual approach to importing Gerber and drill files. It is based on the PhCNC LSSE GerberView technology (button). 

2.1.1 Window for Gerber/Drill files import

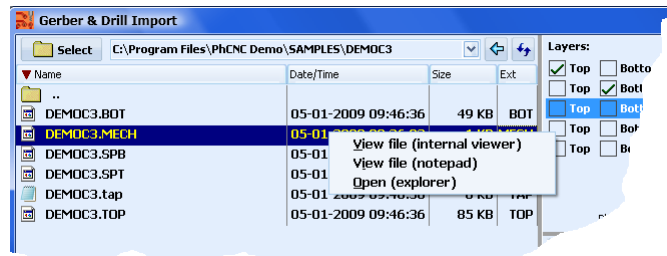
Shown below is the basic view of the Gerber/Drill file import window. Note the numbers in the red brackets. The meaning of each one is explained below.



- (1) – Currently selected folder
- (2) – List of current folder contents. It can be sorted out by any column in ascending or descending order (either numeric or alphabetical)
- (3) – List of files selected for import to PhCNC LSSE and the layer assigned to them.
- (4) – The graphics of currently selected Gerber/Drill file from the list (3)
- (5) – “Gerber View” window of all selected Gerber/Drill files from the list (3)

The selection of Gerber/Drill files is very simple. Folders can be selected with this button or by entering the path manually in (1) and pressing the ENTER key. In the left part of the screen (2) you will find the contents (files and folders) of the selected folder. Folders inside the selected folder can be selected by double clicking on the folder name. Returning to the upper folder can be done by double clicking on this button , or using the left arrow button .

In field (1) (drop down) are listed the last 20 used paths. In many cases that makes the selection even easier.



From the list of files (2) you can open files in different programs by right clicking on them.

The current window arrangement, such as columns size and position etc., will be saved, so the next time you open it they will be the same.

2.1.2 Gerber/Drill files import

To select a file simply click on it in the list (2). PhCNC LSSE analyzes the file content and if it is a valid Gerber/Drill file adds it to the “Layers” list (3).

The selected file has an initial status of “none” and will not be selected for import in PhCNC LSSE.

At this point you can see a small image of the file content in the right (4) and decide what status to select for this file. All selections except “none” will import the file into PhCNC LSSE.

The choices for Gerber files are “Top”, “Bottom”, “Mech” (means mechanical).

The choices for Drill files are “Top”, “Bottom”.

The Gerber file choices are the same for manufacturing solder paste stencils.

Files with status “none” will be removed from the list after the next file selection.

Note:

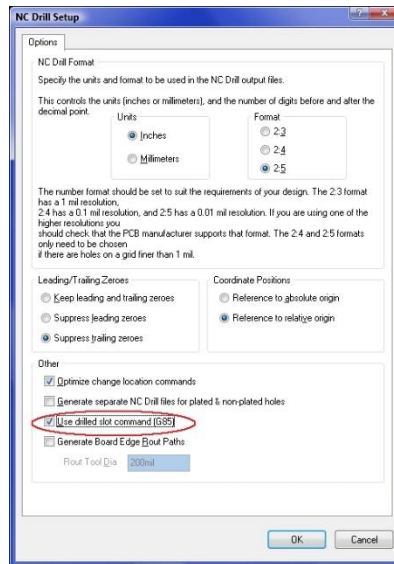
You can change the status/layer of selected files at any time.

It is possible to import more than one drill/drill-slot files (up to 8) and up to 8 mechanical layers.

Note:

Only drill-slot files containing G85 command are good for this import

Below is an example of the export of drill-slot files from Altium Designer CAD system.

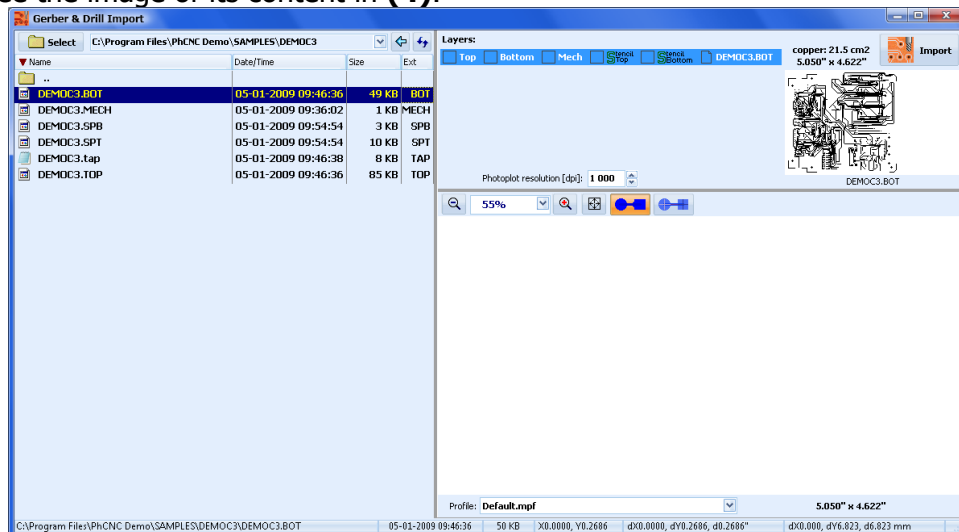


Important:

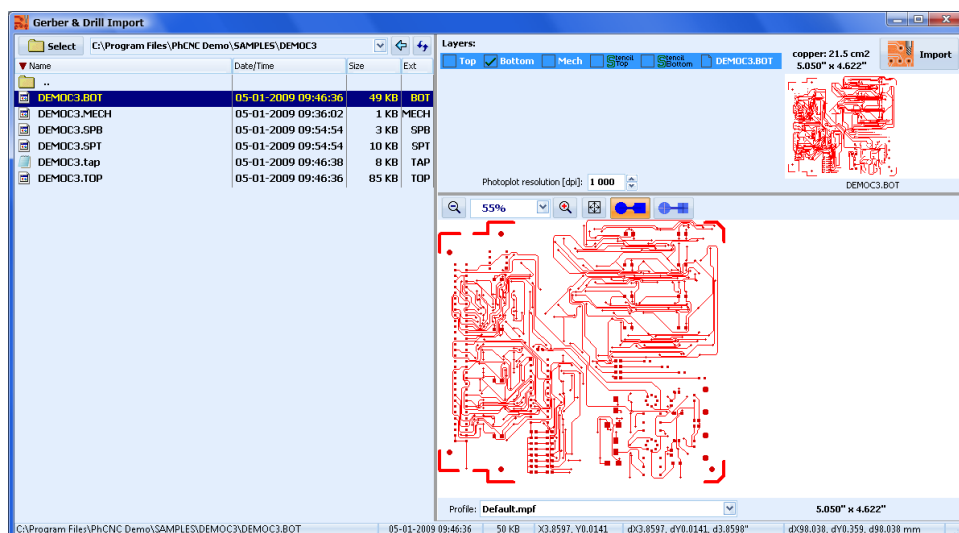
A router bit or drill bit designed for drill-slot operations will serve you better than traditional drill bits.

Here is an example of importing files:

We will select the first file from the list **(2)**. This file is a valid Gerber file and appears in the list **(3)** and you can see the image of its content in **(4)**.

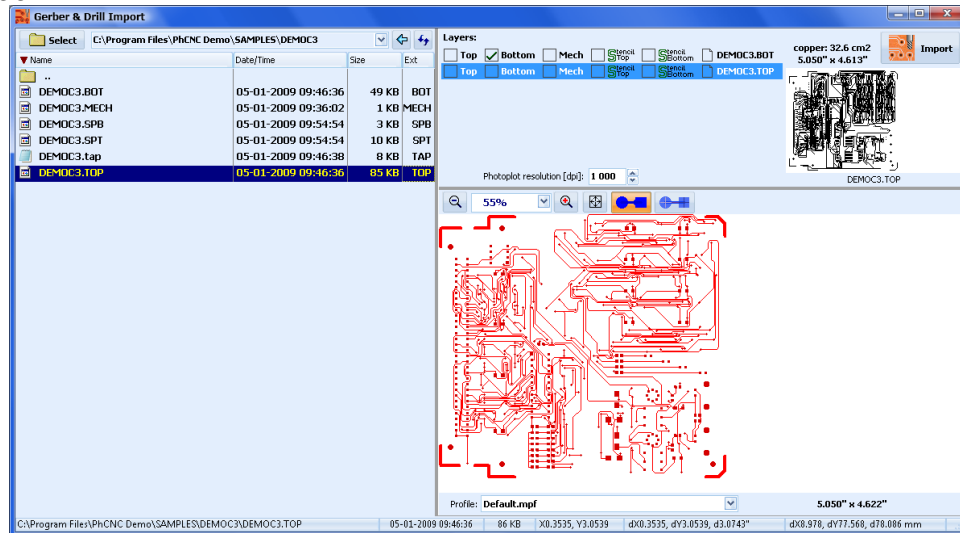


We will select "Bottom" as the status of this file.

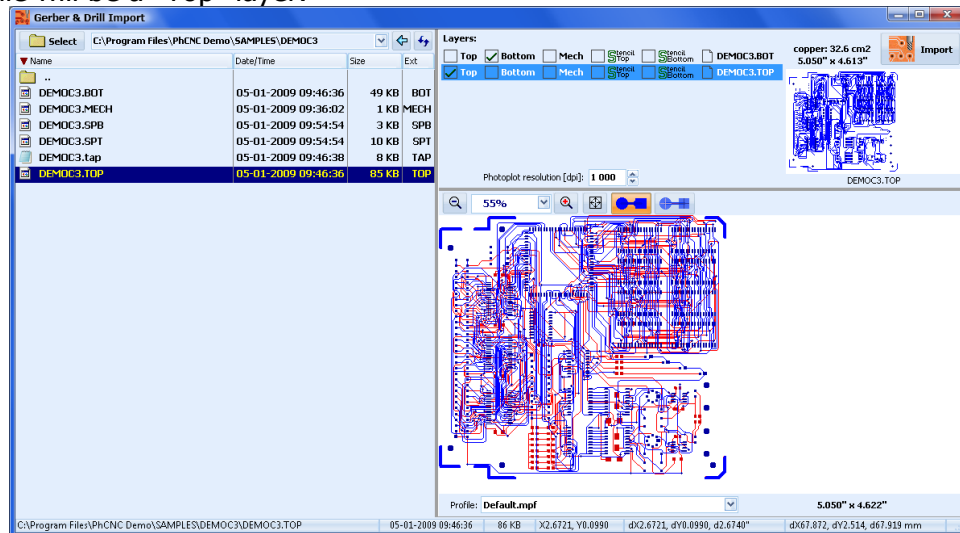


It will appear in the "Gerber View" window **(5)** with the color for bottom layer.

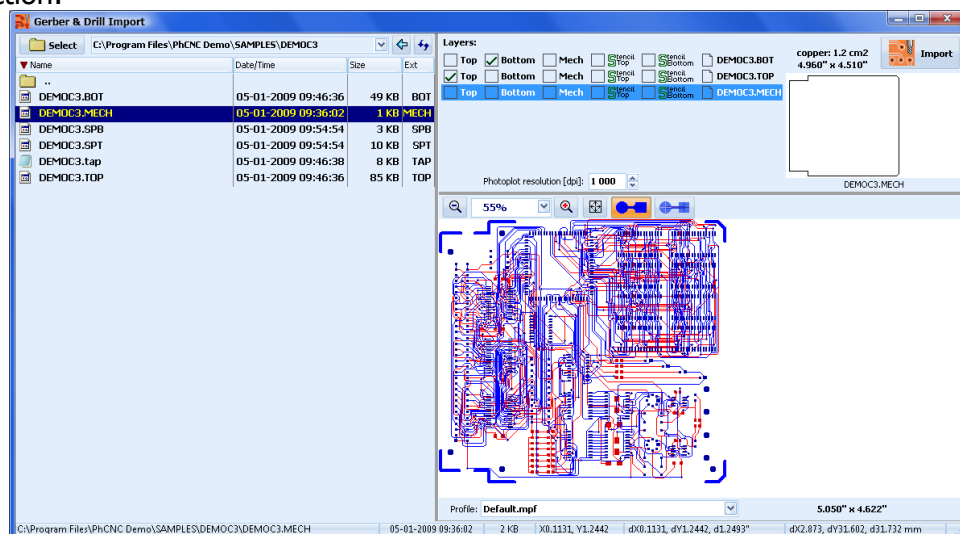
Next file selection:



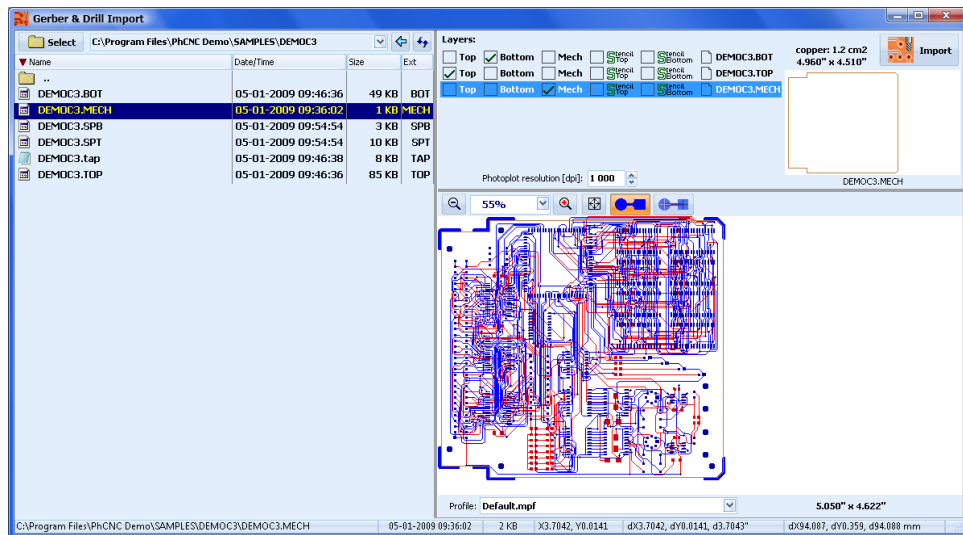
The selected file will be a "Top" layer:



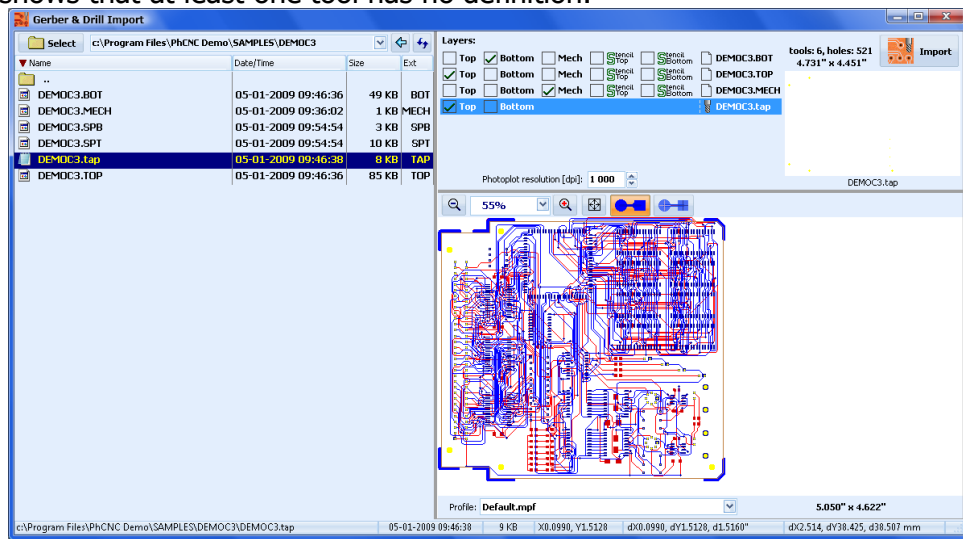
Third file selection:



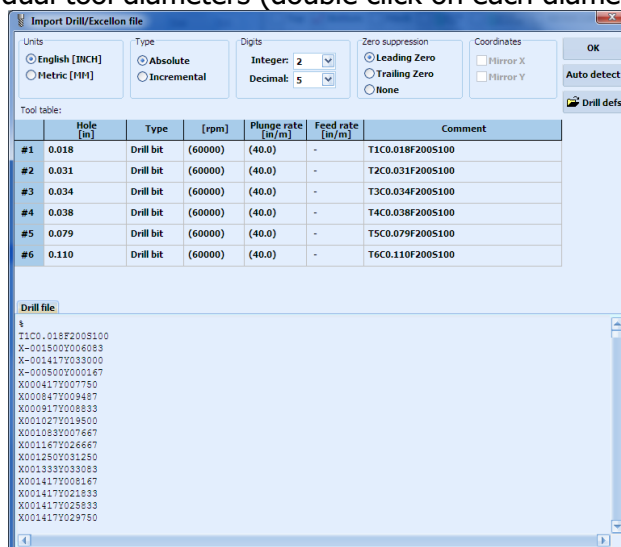
This file is a "Mech" (mechanical) layer file. It contains the internal cuttings and final cutout of the board.



The next selected file is recognized by PhCNC LSSE as a drill file. If it does not contain tool definitions, the symbol  shows that at least one tool has no definition.



After selecting a "Top" layer for drilling, PhCNC LSSE shows the table of the tools used in this file (shown below). Here you may make changes such as the coordinate format to Integer/Decimal, units of measurement, and individual tool diameters (double click on each diameter box).



Unfortunately, Excellon format does not contain Integer/Decimal information; therefore, it is advisable to look at the info file generated from your CAD system. The best way to avoid errors in this matter is to generate an Excellon2 drill file, if your CAD system has this option.

If it is necessary, the tool definitions can be entered manually or from a tool definitions file. To import from such a file, click on the  button.

Button **Auto detect** is designed to do an automatic recognition of the data format for Excellon drill files. In order to be used it needs at least one Gerber file to be selected in advance (Top, Bottom, Stencil Top or Stencil Bottom). This option analyses the coordinates of drill patterns and compares them with the coordinates of the Gerber's. This function allows you to automatically select all parameters of your drill file. It may not work in 1 of 1000 cases and will require a manual settings. In such a case you can study the "Excellon" format or contact us for help. In any case that there is suspicion about the file format, a careful examination (using high zoom) is recommended.

Important:

Parameter "Coordinates" ("Mirror X" and "Mirror Y") should be used ONLY if a mirror transformation of the drill coordinates is needed.

In all cases that the drill file(s) need manipulated we recommend the following sequence:

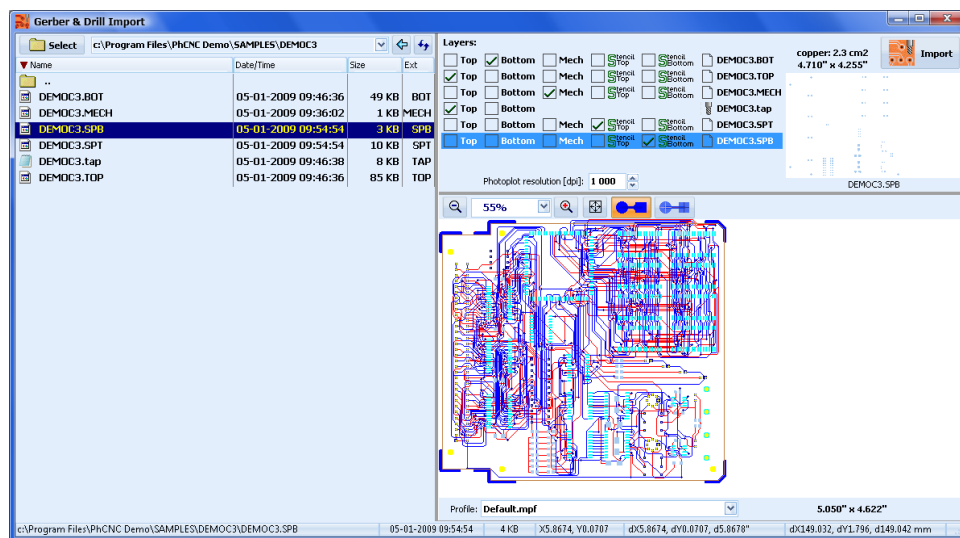
- Import your Gerber files completely (but without drill file(s))
- Start a new procedure "Import Gerber & drill"
- Import your drill file(s)
- Now you can select your drill file(s) as a separate object
- Make all transformations needed and align your drill files to the Gerbers
- Finally use high zoom for precise alignment (you will be able to match to 0.0001")

Important:

Contemporary CAD products usually export the drill file in Excellon2 format and it contains all the tools definition information.

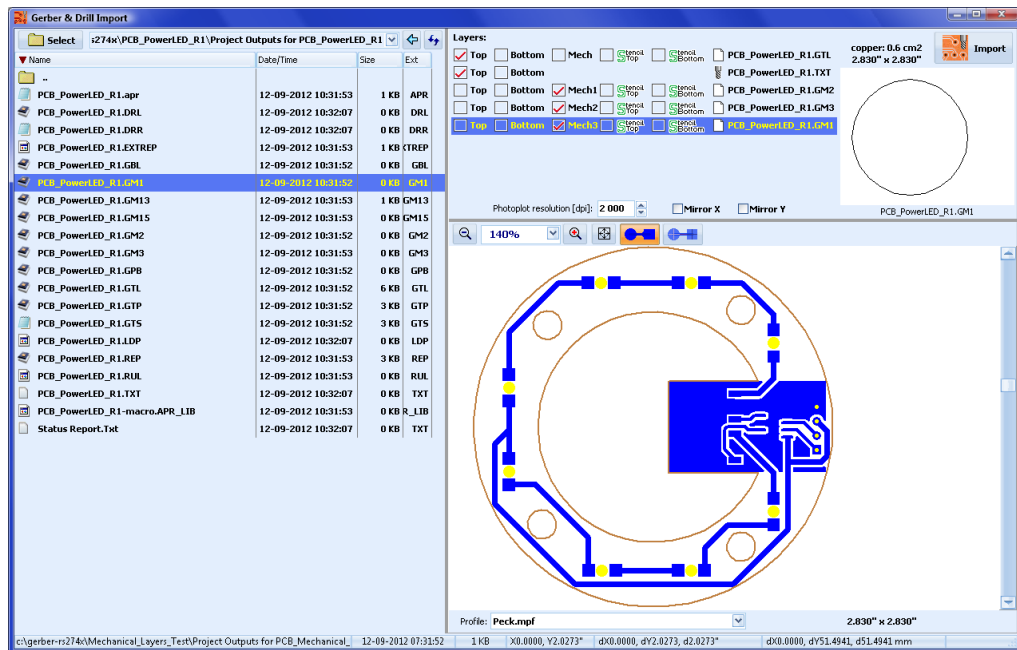
If for some reason the file format is not correct (like inch/metric, fractional digits, leading/trailing zeros suppression etc.) it will be very obvious. To correct the problem, use double click at file name and change the necessary settings.

In the example below all files selected are ready for import into PhCNC LSSE.



Note:

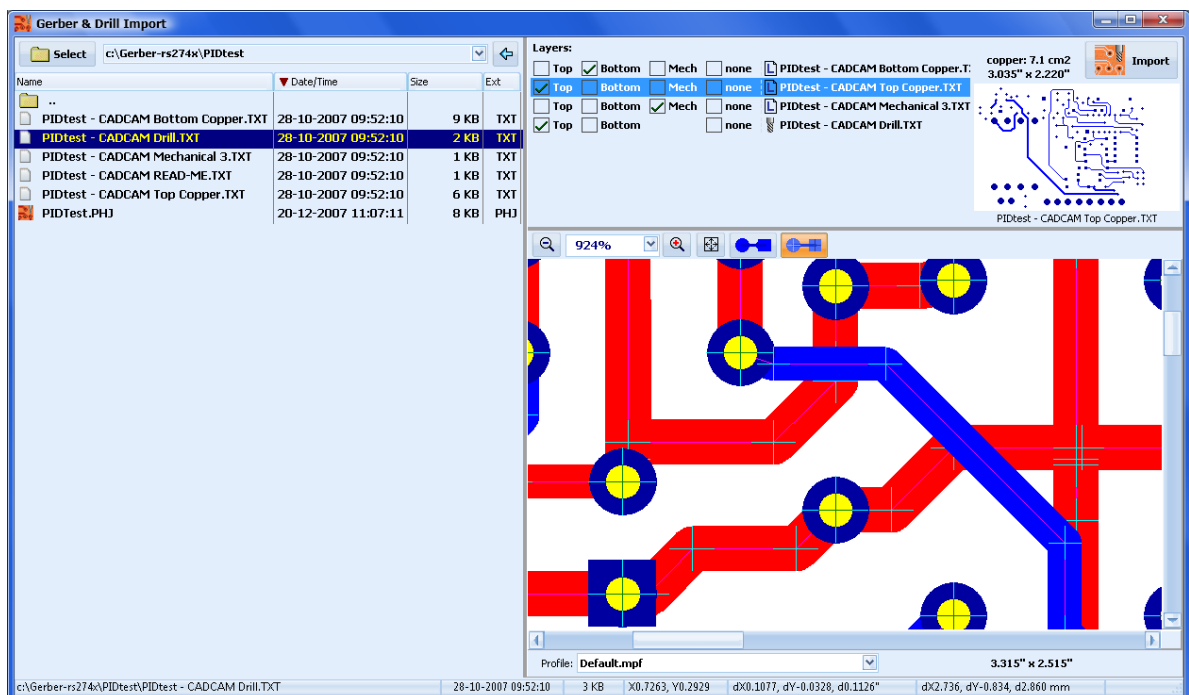
PhCNC LSSE allows the operator to import more than one mechanical layer (up to 8 layers are allowed). This is shown on the screen below. Each layer is imported from a separate Gerber file.



After the import of more than one mechanical layer, PhCNC LSSE generates a single file that contains all imported information. This file is named "Mechanical.GM"

2.1.3 "Gerber View" window

Now you have all the needed files in the "Gerber View" window (5). Here you can zoom in for more detail and perform measurements, the same way, as in CAM mode.



To set the reference point for a measurement press the SPACE BAR or make a right click. Move your mouse pointer to the new point and read the difference (dX and dY) in the status bar.

In the video clip below, the "Gerber & Drill files import" is demonstrated:

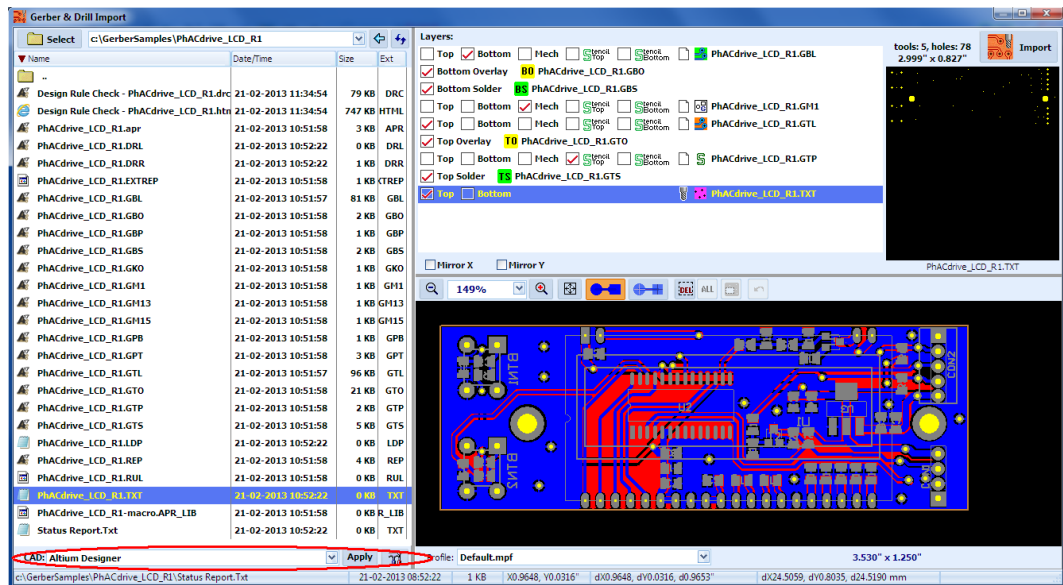
<http://youtu.be/JIx5mMC7Cqq>

2.1.4 Profiles for automated Gerber & Excellon import

PhCNC LSSE has ability to create profiles for automated import to the generated from the PCB CAD systems Gerber & Excellon files for the layers.

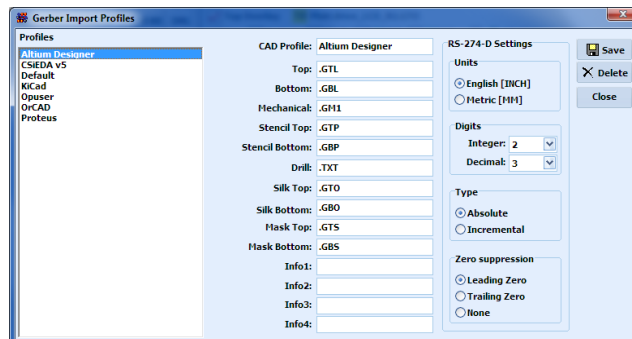
The predefined profiles are shown in the list below. When the user select the desired PCB CAD profile and presses the button "Apply", PhCNC LSSE will select automatically the files corresponding to the PCB layers (according to the selected CAM profile).

On the picture below are shown the selected Gerber & Excellon files generated from Altium Designer:



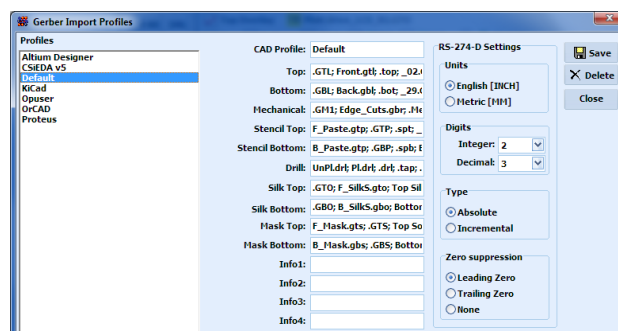
The predefined profile "Default" is a universal profile, and will work for most of the PCB CAD outputs (Gerber & Excellon).

The button  opens the "Gerber Import Profiles" window, allowing to the user to create/edit/delete the existing profiles.



Each profile contains information (the file name extension, or part of the file name) about the names of the files generated from the specified PCB CAD system.

PhCNC may support more than one extension for the file name for every layer. Typical example is the profile "Default":



The different file extensions must be separated with ";".

In the example, in the "Top" layer will be selected Gerber file which name ends with some of the next strings:

"GTL"

"Front.gtl"

".top"

"_02.GBR"

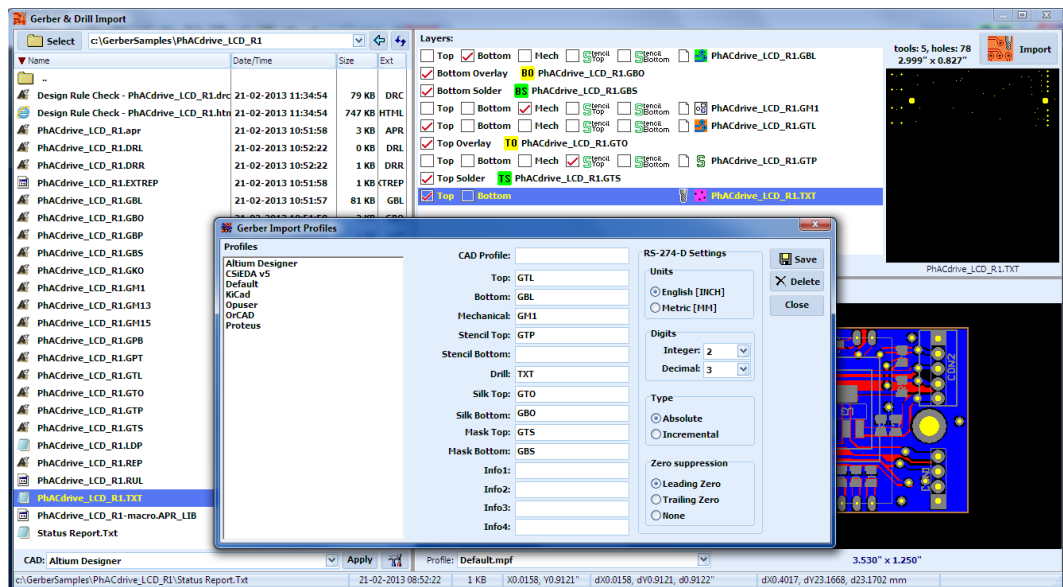
"top.gbr"

"Top Copper.TXT"

"Top Pattern.gbr"

The PCB CAD import profiles contains information about coordinates format of the RS-274-D gerber files (not used during import of the RS-274-X gerbers).

The easiest way to create profile for automated Gerber & Excellon import is the next:



- Select the Gerber & Excellon files for the layers (ready for import into PhCNC)

- Press the button 

Now PhCNC will fill the unique file names extensions (as is shown above) using the names of already selected files.

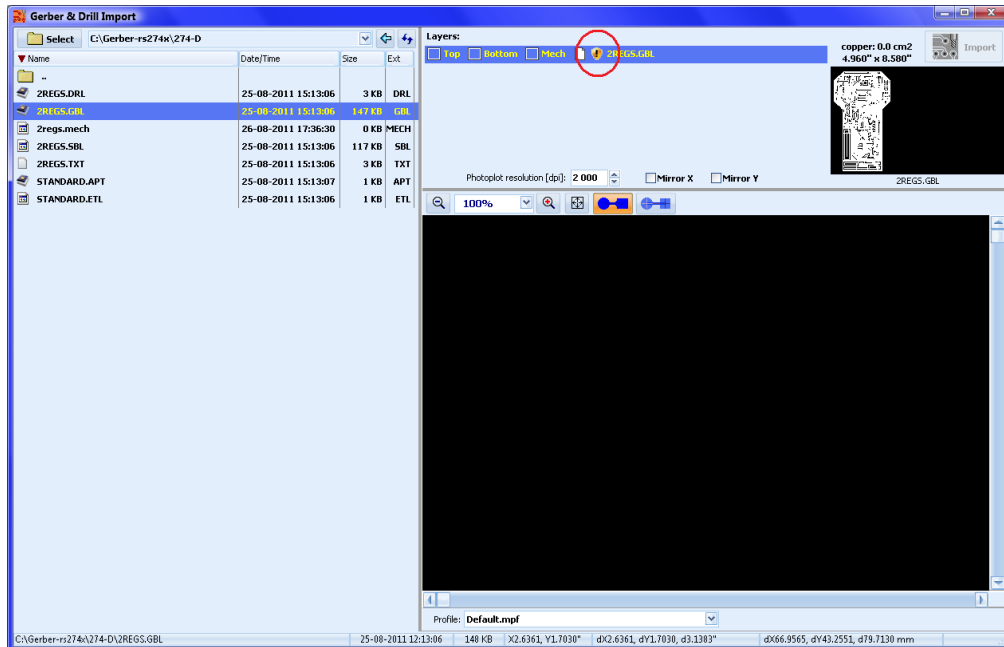
To save the profile, you have to enter the name for the profile and press the button "Save".

Video:

https://youtu.be/KQ6m_ytnq90

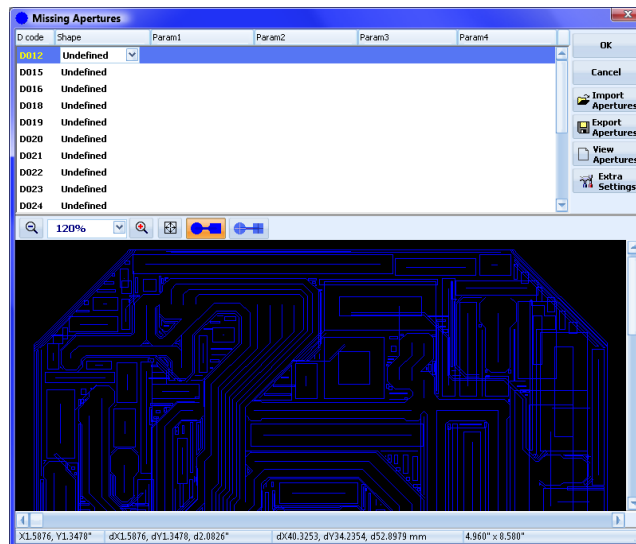
2.1.5 Gerber RS 274-D files import

Gerber files in RS 274-D format are considered obsolete. All PCB CAD systems generate Gerber files in RS 274-X (called Extended Gerber). Despite this we offer the possibility to import RS 274-D for the users of older CAD systems. When such a file is selected an attention sign will be displayed in front of the file name to alert the operator that the file does not contain the description of the used apertures (Gerber RS 274-D):

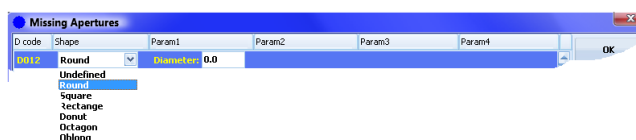


During the file selection an additional window will appear for apertures to be entered. Due to the fact that there is no standard format for aperture descriptions in RS 274-D they must be entered manually

2.1.5.1 Manual aperture entry



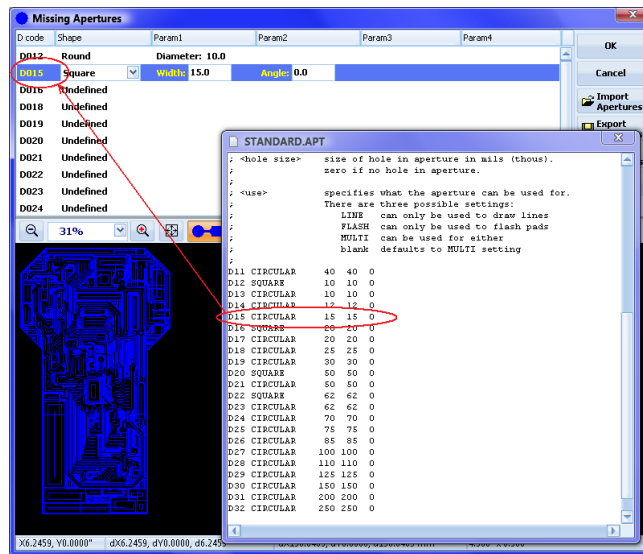
To enter/change parameters you have to click on it. This will open fields for its aperture type and parameters.



After the apertures and their parameters are entered, it is recommended to save them in a *.APT file for future use (another layer or another project) using the button "Export Apertures".

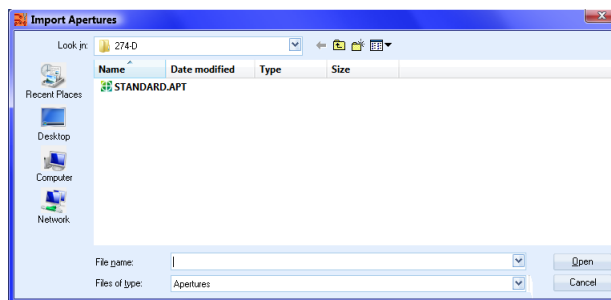
Important:

If the apertures are presented in a text file, you may use the "View Apertures" button. This will open an additional window that will make the entry process more convenient.

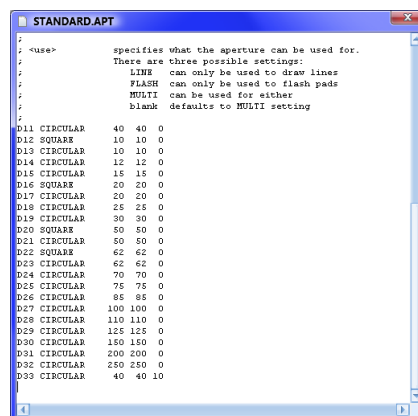


2.1.5.2 Aperture definitions import in PhCNC (file *.APT)

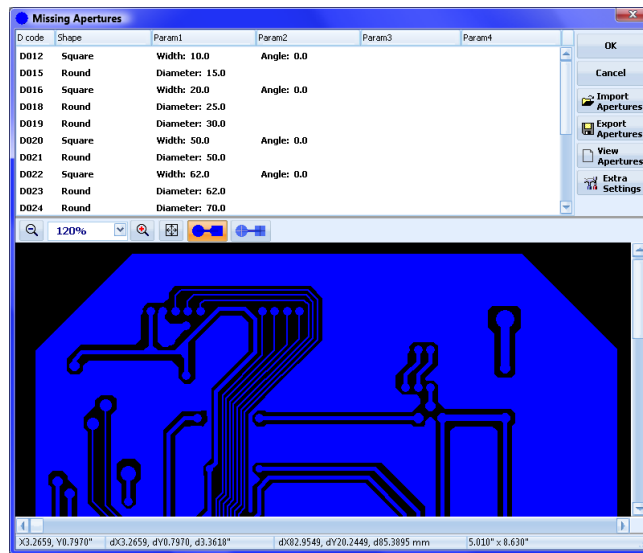
You can import apertures definitions in PhCNC when you use a RS 274-D Gerber format. Once the aperture file is generated (or supplied) you can enter it using "Import Apertures"



Shown below is the format of *.APT file that you can enter in PhCNC LSSE.

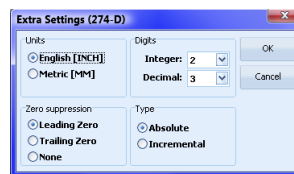


After the import of the apertures, it is highly recommended to review the graphics of the layer for missing or improperly entered apertures. After any changes in the apertures, PhCNC LSSE will redraw/display the changes in the layer.

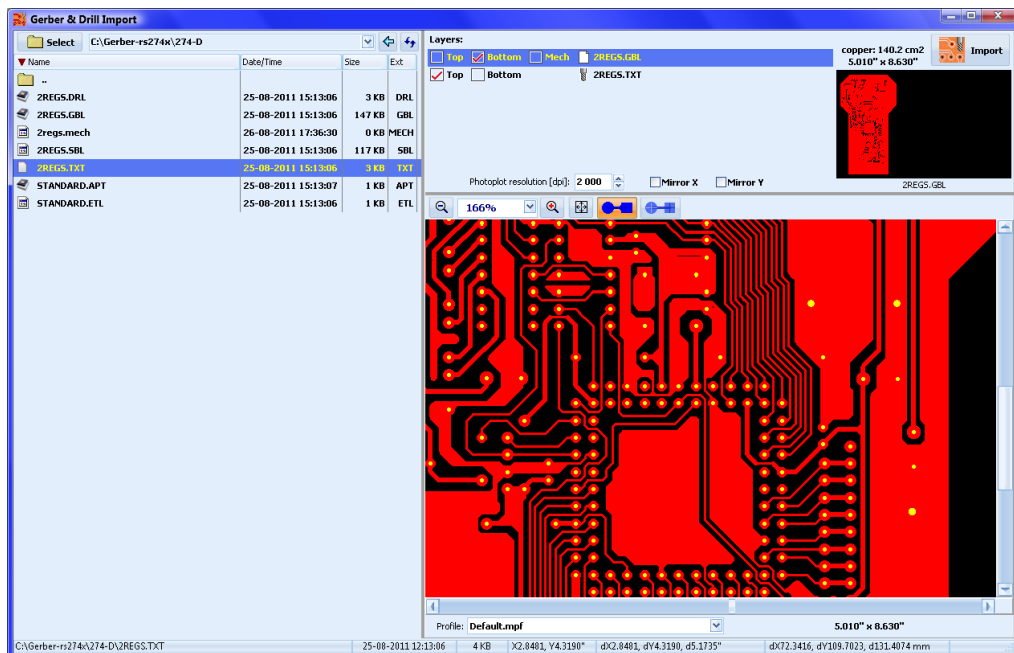


2.1.5.3 Extra Settings

Gerber RS 274-D does not contain the information about the coordinates format, units of measurement etc. They usually are sent to board manufacturers by separate letter or telephone conversations. Here they can be entered in PhCNC by using the "Extra Settings" button



Entering wrong parameters in this window will definitely create a wrong interpretation of the Gerber file and will be visible in the graphic window. It is important to carefully check the image and its size and make sure that it is what you intended to have. Shown below is the file after correct import:



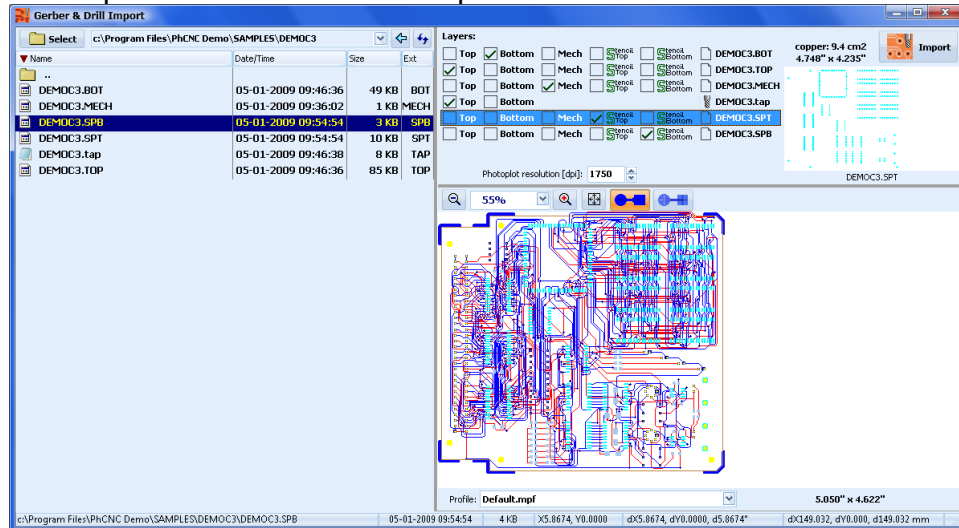
Important:

If the CAD system that you are using can export Gerber files in both RS 274-D and RS 274-X, you definitely have to use RS 274-X (Extended Gerber). This will save you time, efforts and reduce chances of errors.

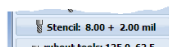
2.1.6 Gerber files for Stencils Import

PhCNC LSSE can import Gerber data layers designed to machine stencils for solder paste placement. Stencil layers can be imported together with the files for board(s) prototyping (recommended when cloning and reorientation of the board in PhCNC LSSE is needed) or separately. This import is similar to top and bottom Gerber files for boards. You may import top and bottom Stencil files ("Stencil bottom" is needed only for boards that are populated with components on both sides). "Stencil Top" and "Stencil Bottom" are virtual names, that reflect the stencil for board top side and the stencil for board bottom side, each made on separate work piece.

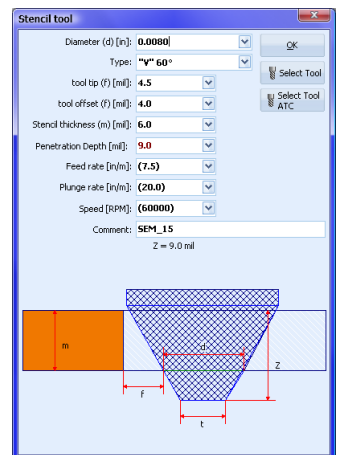
Shown below is a sample window with stencils import:



When stencil files are imported in the project, additional buttons will appear on the left side of CAM window shown below.



The "Stencil" button allows you to select a tool and set the parameters for it. There are two types of tools that can be used in this operation. "V" tip tools (the same used mostly for insulation of the traces) and stub end mills. "V" tip tools will allow you to achieve corner radiuses on square pads close to 3 mils without paying the price for a 6 mils stub end mill. Stub end mills will make the openings in your stencil with completely vertical walls. Using "V" tip tools the actual tool diameter is the track on the bottom side of your work piece. This requires a precise depth setting. Stub end mills have lower requirements for this setting. Tapered walls of the stencils made using "V" tools may require a bit bigger "safety zone" (tool offset), an internal compensation to the pad size (usually 2-5 mils). The operator will be able to set this too. All settings needed for the stencils tool appear with a click on the Stencil button. To the right is the stencil tool window.

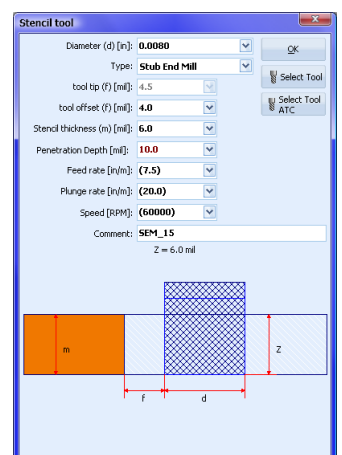


Where:

- 'd' is the tool track diameter calculated for the bottom side of your work piece.
- 't' is the tip diameter of your tool. In some cases, it is not published by distributor/manufacturer. Call the source and find this parameter. It is important to know it in order to have accurate depth limiter settings.
- 'f' is the internal compensation, technologically needed for the process of solder paste application. (usually 2-5 mils)
- 'm' is the thickness of your work piece.

Tool penetration depth is calculated automatically by PhCNC LSSE and shown in the tool change window.

When using a "Stub End Mill" the depth is only recommended (should be more than material thickness).



2.1.6.1 Stencil layer(s) visualization

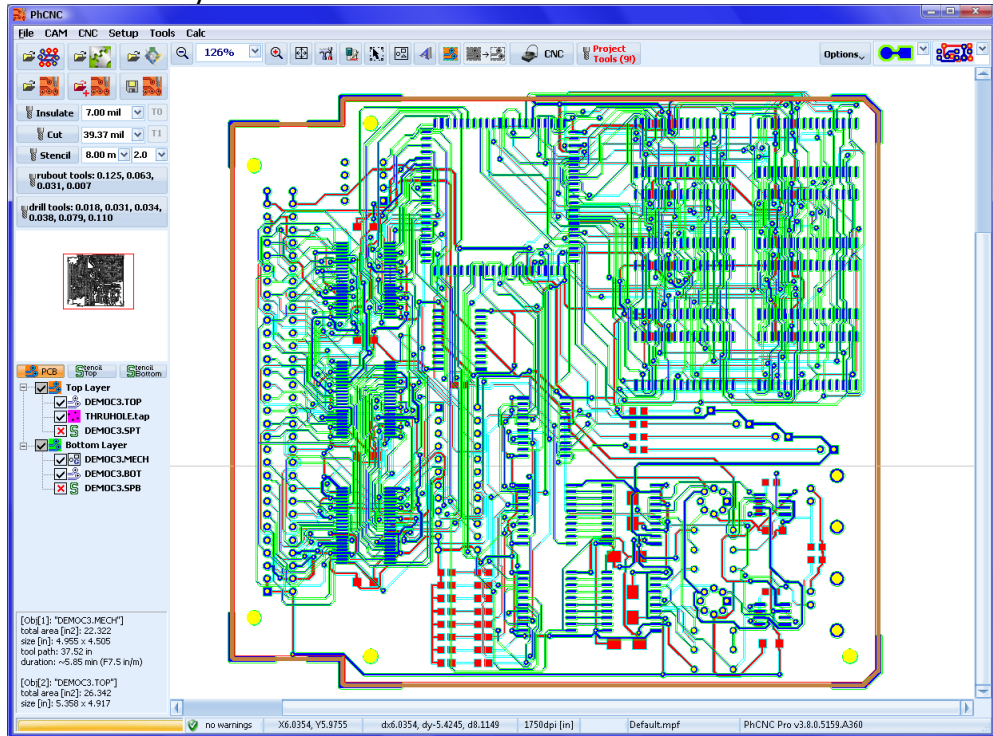
If a project containing stencil layer(s) has been imported, additional buttons that control the visualization of all layers will appear.



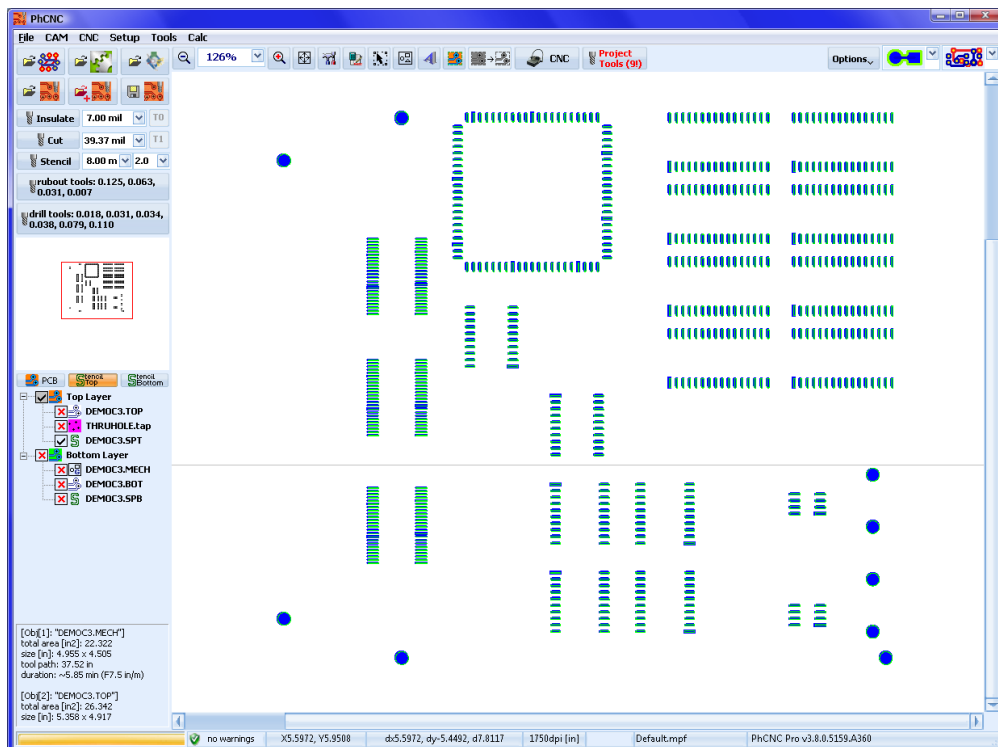
The "PCB" button, shows all files except the stencils

The "Stencil Top" & "Stencil Bottom" shows Top & Bottom stencil images

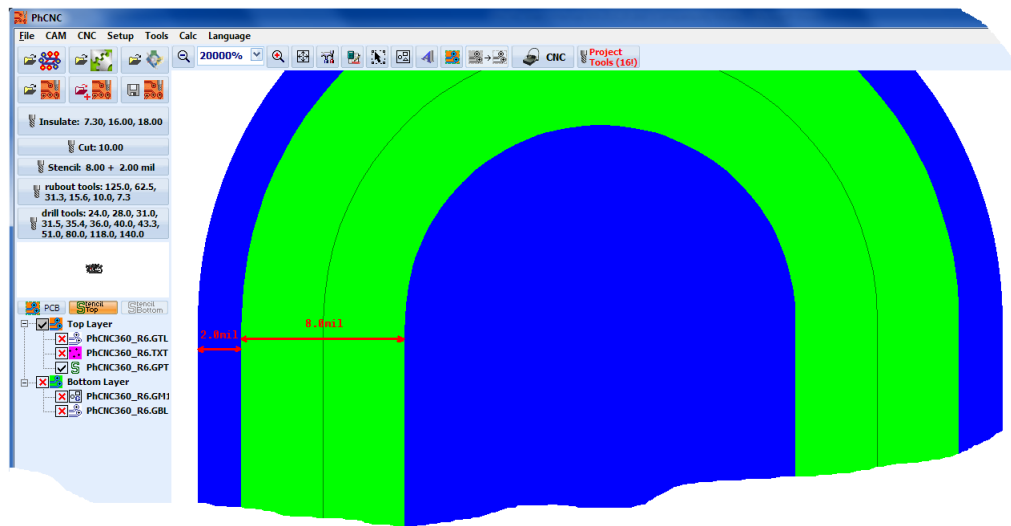
Shown below "PCB" files only:



Stencil Top:



Below is an example of tool path dimensions and "safety zone" compensation

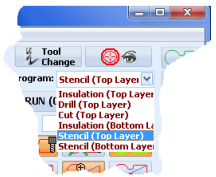


2.1.6.2 Stencils machining in CNC mode

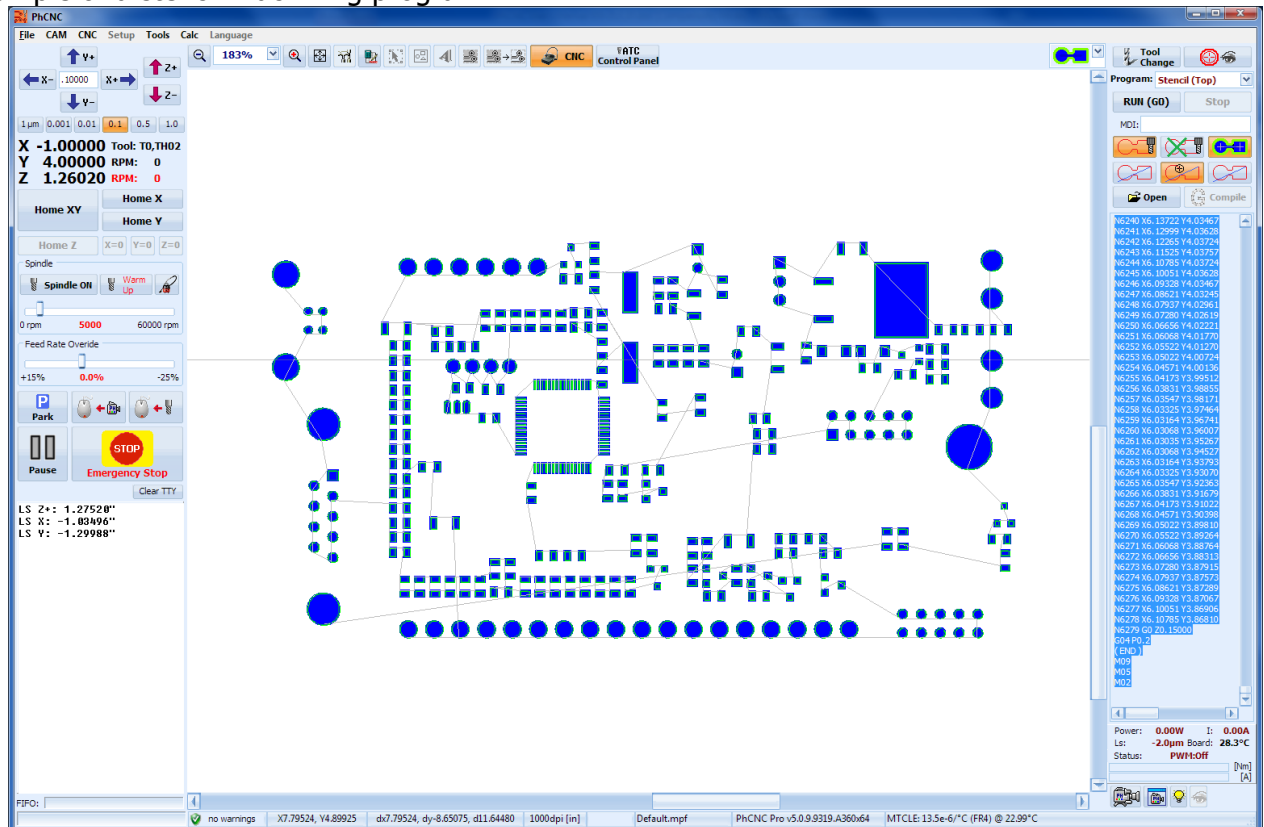
Entering the CNC mode the operator will see the programs for stencil machining:

Important:

PhCNC LSSE generates programs in CNC mode **ONLY FOR THE OBJECTS/LAYERS VISIBLE** in CAM mode.



Example of a stencil machining program:



Note:

When you machine a PCB, your "bottom" files are mirrored along the X axis. Stencil machining does not mirror the "Stencil Bottom" file. Here the "Bottom" means for the bottom side of the board and does not need to be mirrored. "Top" and "Bottom" stencils are separate work pieces.

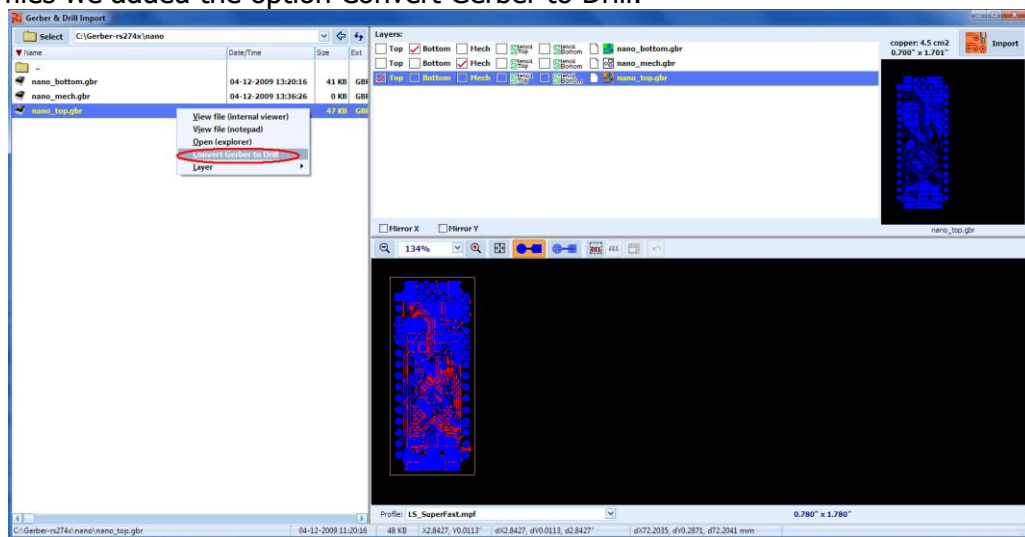
Important:

Due to significantly different properties of the materials used for stencils, there are separate settings for the machining in the "Setup" menu:

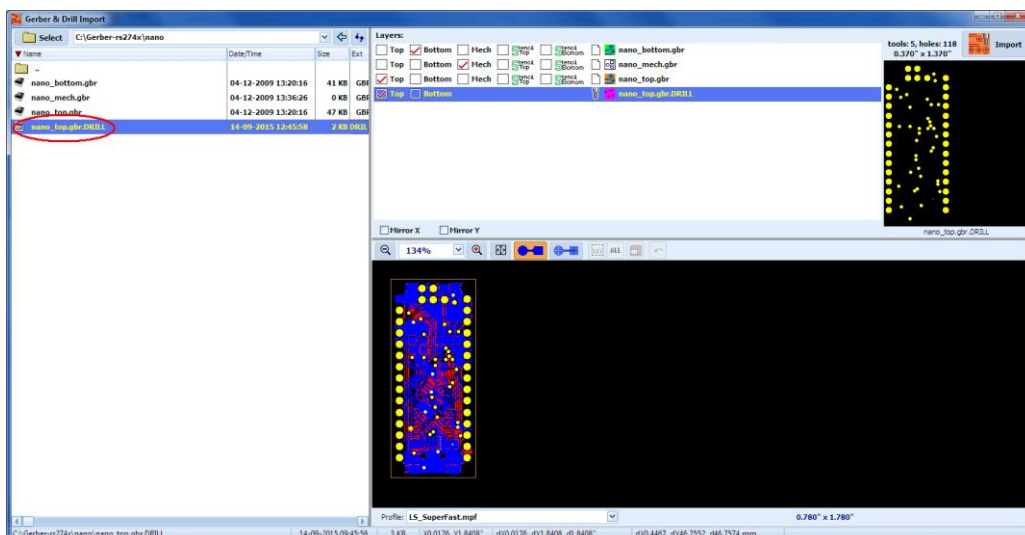


2.1.7 Convert Gerber to Drill

PhCNC LSSE allows drill files import in Excellon/Excellon2 formats. In rare cases it is possible that CAD system generates Gerber format (using round apertures for each hole). In order to be able to use these type of drill files we added the option Convert Gerber to Drill:



After execution of "Convert Gerber to Drill", you will find in the list of files a file with the same name as the Gerber file, but with an extension .DRILL. The new file then can be selected as a Drill file for the project.



IMPORTANT

In order "Convert Gerber to Drill" to function properly the folder should allow to write files.

If a drill file already exist "Convert Gerber to Drill" option is disabled.

"Convert Gerber to Drill" option is also disabled if the file selected is not a Gerber file.

Please make sure that you are using full screen video to be able to see all details.

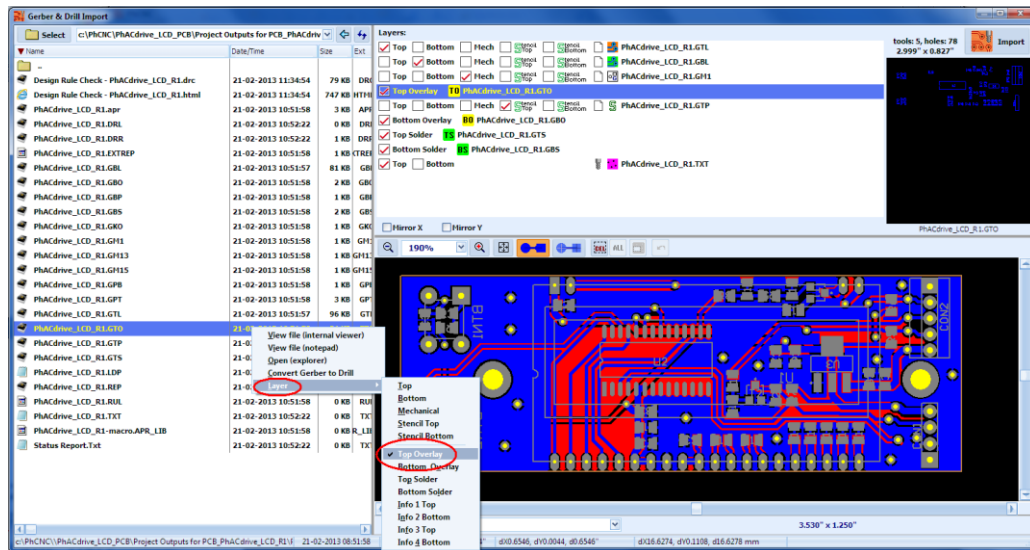
Convert Gerber to Drill - video:

<https://youtu.be/U6KZnUt10Yo>

2.1.8 Info layers import

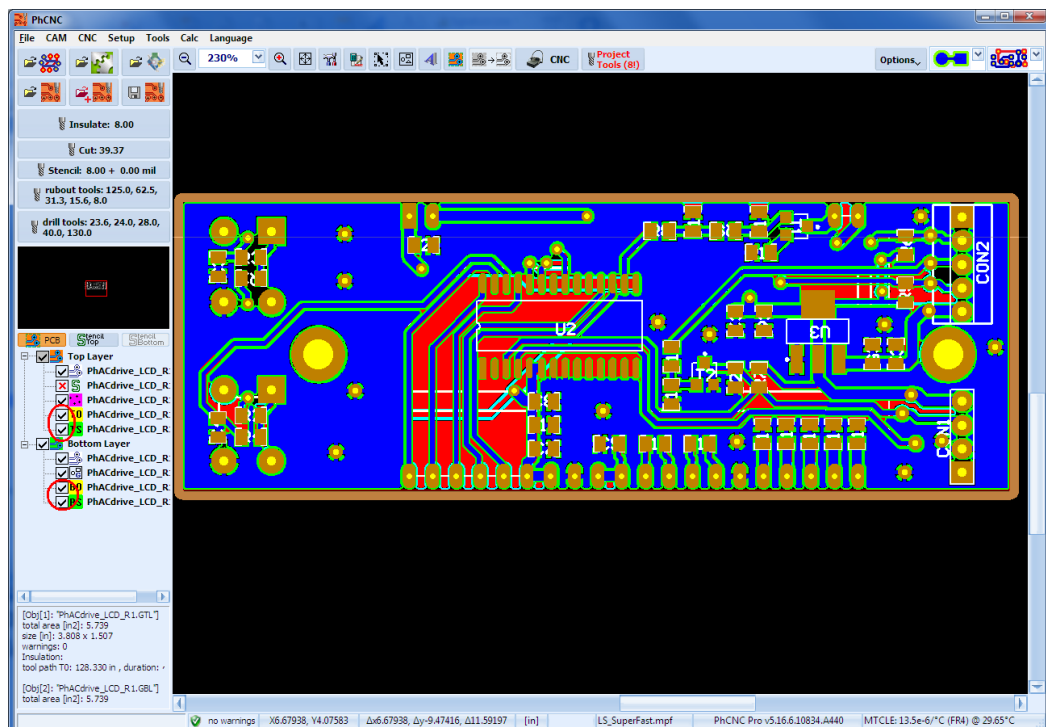
PhCNC LSSE now allows the import of additional informational Gerber files in addition to the files needed for machining such as: Top Silk, Bottom Silk, Top Mask, Bottom Mask, etc.

The additional files are not used for machining but offer convenience when a "Gerber Export" and "Print" are used.



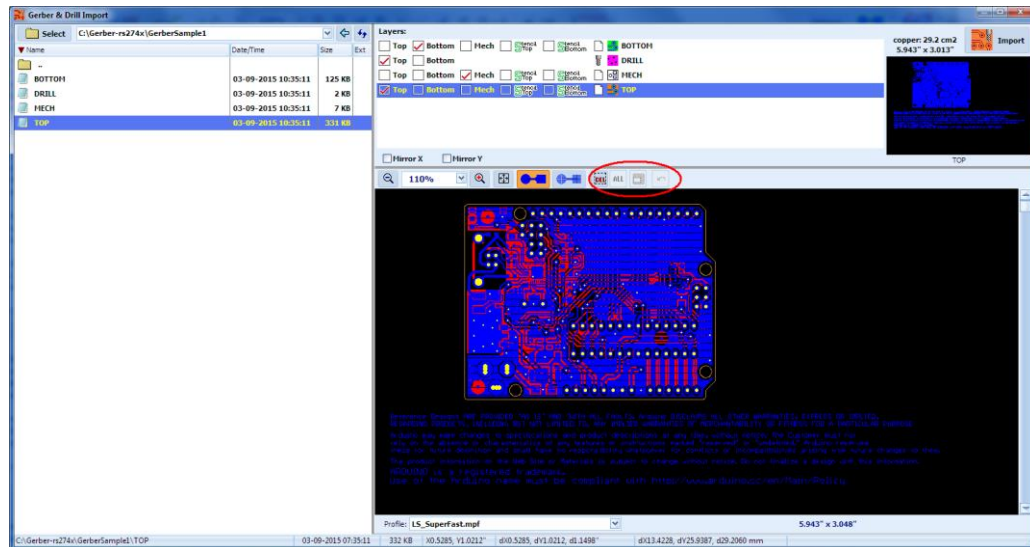
Layers for machining: "Top", "Bottom", "Drill", "Mechanical", "Stencil Top" (Top Paste), "Stencil Bottom" (Bottom Paste), also can be selected from popup menu.

After an import in PhCNC LSSE, info layers can be shown/hidden the same way as the layers for machining.

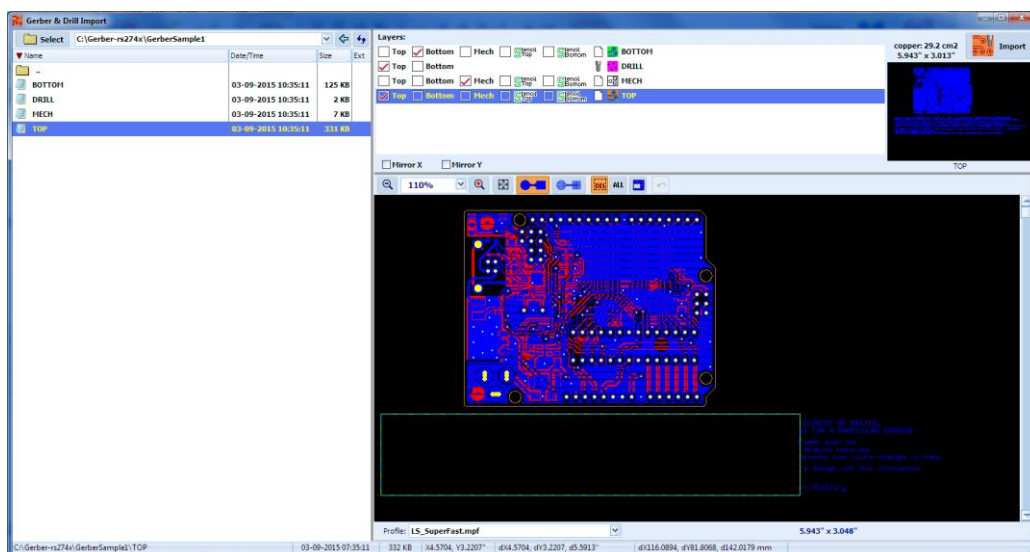


2.1.9 Gerber editor

PhCNC LSSE has a function that allows the operator to remove certain Gerber components when they are considered not needed for the machining. Typical examples include text, dimensional lines etc., that are (in some cases) included in the Gerber file. In the example below a large text field need to be removed from the Gerber file.



Activate the button . Using the mouse select the area to be deleted. All Gerber primitives that are completely included in the selected area will disappear from the selected file in the Layers list. If **ALL** button is active, it will remove the Gerber components from all layers. It is a real time operation that allows the operator to see if the selection is right.



Button  (Undo), reverses the changes made. The editor allows up to 32 stages of Undo command. Button  changes the function of the selected region. When activated the editor deletes all Gerber primitives outside the selected region. In some cases, this is the more convenient way to edit the files.

Below are links to videos that show details of the Gerber editor in PhCNC LSSE. Please make sure that you are using full screen video in order to be able to see all details.

Gerber import & export example (editor) #1

<https://youtu.be/JY9lpaCIgEq>

Gerber import example (editor) #2

<https://youtu.be/4MSkDD8JGjw>

Gerber import example (editor) #3

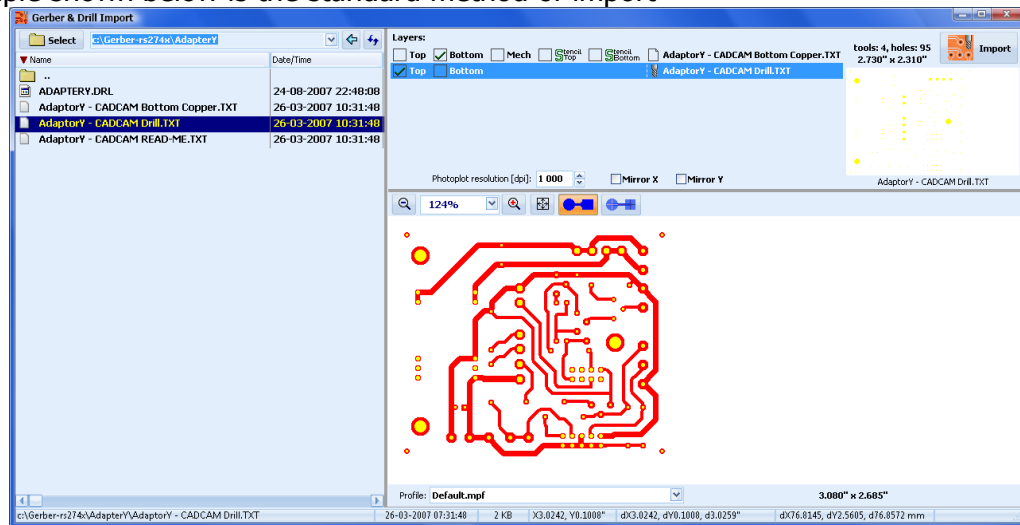
<https://youtu.be/v7VtASRzN0I>

2.1.10 Single layer PCB import

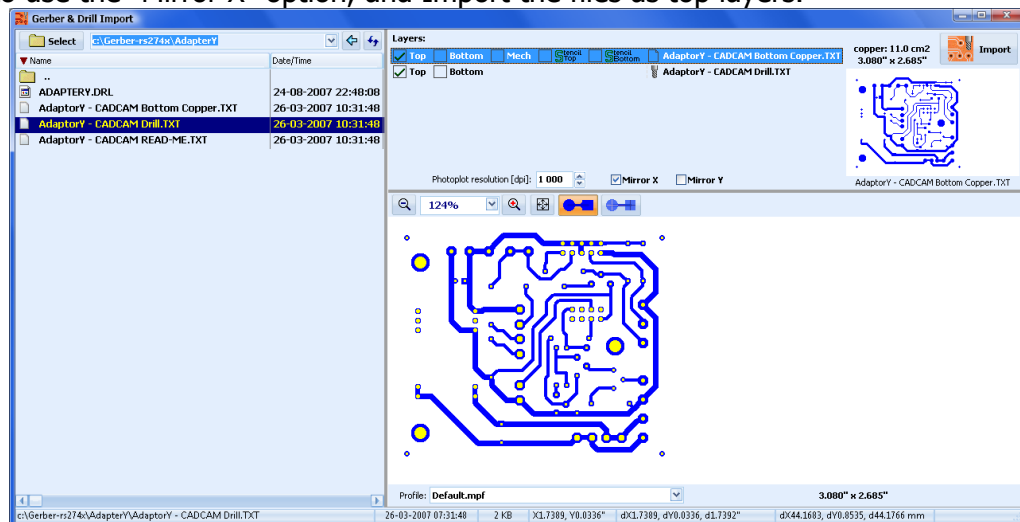
When you import single sided PCB designs the Gerber file for the solder layer (normally bottom layer) comes from your cad system as if you are looking at the board from the top. Our system mirrors this when you declare it "bottom layer", but this creates unnecessary difficulties in placing the project on your work piece. This option makes the setup process more convenient when you are handling single sided projects.

Using the mirror option, you can import your Gerber and Drill files as a top and they will not be mirrored during the machining (CNC mode).

In the example shown below is the standard method of import



It is easier to use the "Mirror X" option, and Import the files as top layers.



This way the imported files are all located as top layers, and the machining and setup are easier.

Loaded PCB projects will be placed at the center of the workspace.

If you would like to load more than one project, move the loaded project to the desired place (it is not a final position you may move it again and again).

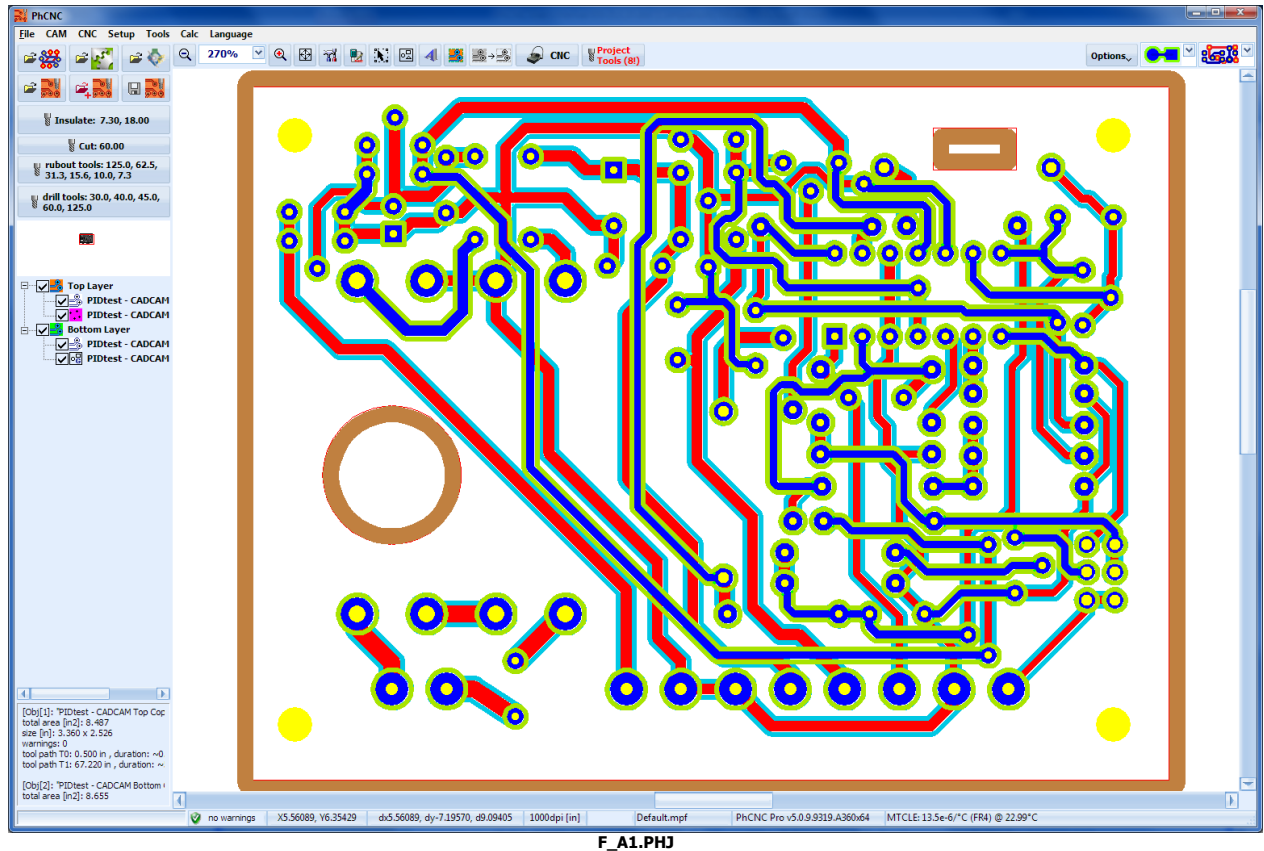
If you need to make more than one copy of any board, use clone (copy) function instead of loading your files again.

Cloned objects have some advantages:

- Reducing calculation time for insulation and rubout calculations
- Saving memory space. Ten copies take as much memory as an original.

In the example below, four files have been loaded: Top, Bottom, Drill and Mechanical files.

All files loaded together in a single project are presented as a single object in order to manipulate them without generating undesired inconsistencies between the layers. You may clone, move, rotate etc. If you need to move any of the layers individually use the **Ungroup selected objects**, command described later in this manual.



2.2 PhCNC LSSE Project Files

PhCNC LSSE saves the projects in .PHJ files, where all the necessary information concerning the project is saved in a compressed format. Its file size is 4-5 times smaller than the volume of all files imported in the project. The extension .PHJ is registered in Windows Explorer so that if you want you can open .PHJ files directly from Windows Explorer. When opening projects with Windows Explorer the projects will be opened using the **Import** mode.

PhCNC LSSE supports two modes of loading .PHJ project files:

Button  when pressed closes the current project (if it is not saved yet, you will receive a message asking if you want to save it) and opens the .PHJ file you select with all the tools for insulation, cut-out and rubout defined (including DPI resolution) in the .PHJ project that you open.

Button  when pressed adds the .PHJ file you select to the current project, but the tools specified in the imported file will be ignored in favor of the tools specified by the current project.

For your convenience, the "File" menu lists recently opened .PHJ projects (up to 8 files). Upon exiting PhCNC LSSE, you will be asked to save the current project (if it has been modified) and even if you answer "No" it will be saved to a special file called "Recent Project" available in the file menu automatically and can be opened from the file menu by clicking on "Recent Project". The last file opened with the latest changes will always be here just in case you accidentally close the file without saving.

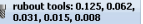
2.3 Buttons and fields used in CAM mode

The Buttons & drop-down area  is for Zoom In/Out or to set the zoom percentage. If your monitor is set for the correct resolution, you will have the correct size image at 100% zoom. It is very convenient to use a mouse with a wheel for zoom In/Out. The zoom is always centered to the mouse pointer.

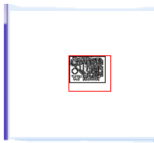
The button  Insulate: 7.30, 18.00 is a **Very important setting!** It is for the insulation tools, used in the calculation of the insulation tool paths. If you change insulation tools, PhCNC LSSE automatically recalculates and redraws all insulation tool paths.

Important:

PhCNC LSSE insulation tool path calculations, **do not allow intrusion into the traces and pads**, if your tool diameter is bigger than the space between traces and/or pads they will be left non-insulated. PhCNC LSSE detects and displays the problem areas (See 2.13 Design rules violation (DRV) detector). It is strongly recommended to view and check all critical zones of your design, using a high level of zoom!

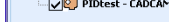
Button  is for Rubout tools selection (described later in this manual).

Drop down  Cut: 60.00 is to set cutout tool diameter (if any). If you change this value, PhCNC LSSE automatically recalculates and redraws all cutout tool paths.



Window  is the panoramic window it shows all objects placed within the machine workspace. The Panoramic Window is a convenient way to view your entire project. You can left click inside this window, to view the desired portion of the workspace. This is convenient if you do not have a mouse wheel.



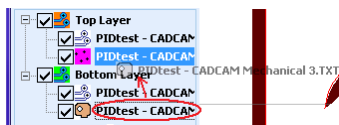
Window  is the loaded files window it shows all the loaded files and the layer they are on. Clicking on the checkmark ☒ will turn it into a cross-mark, disabling and hiding the corresponding layer.

Note:

When you enable or disable layers in CAM mode you may use the CONTROL button on the keyboard. When it is hold down the enable/disable function apply to all layers with the same name. This offers convenience to handle the function when you have many copies of the same project on the screen.

Important:

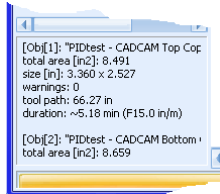
Disabled layers are not manipulated by the CAM edit functions. For example, if you like to select a rubout area in one of the layers, you can disable (hide) the other ones. If you need to move a file from one layer to another, click on file name, hold and drag.



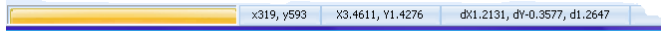
In the example above, the Mechanical layer is being moved from the Bottom to the Top Layer.

File type symbols:

-  - Gerber file
-  - Drill file
-  - Mechanical layer ("PhCNC LSSE Printer" or "Insert Text")
-  - Mechanical layer (Gerber)



Window is the statistical information window. Here, you can find the calculation for the time needed to complete the job based on the feed rate. For more detailed information, make a left click inside this window and then press F1.



This is the status bar it contains the following fields in the order listed below:

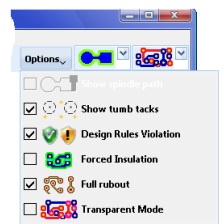
- The progress bar, which shows the progress of the current task
- The screen coordinates of the mouse pointer
- The real coordinates of the mouse pointer
- The relative coordinates of the mouse pointer to a specified origin and distance from that origin. Use the "space bar" on your keyboard to zero the relative coordinates at any point on the project. This feature can also be used to measure the distance between any two points.

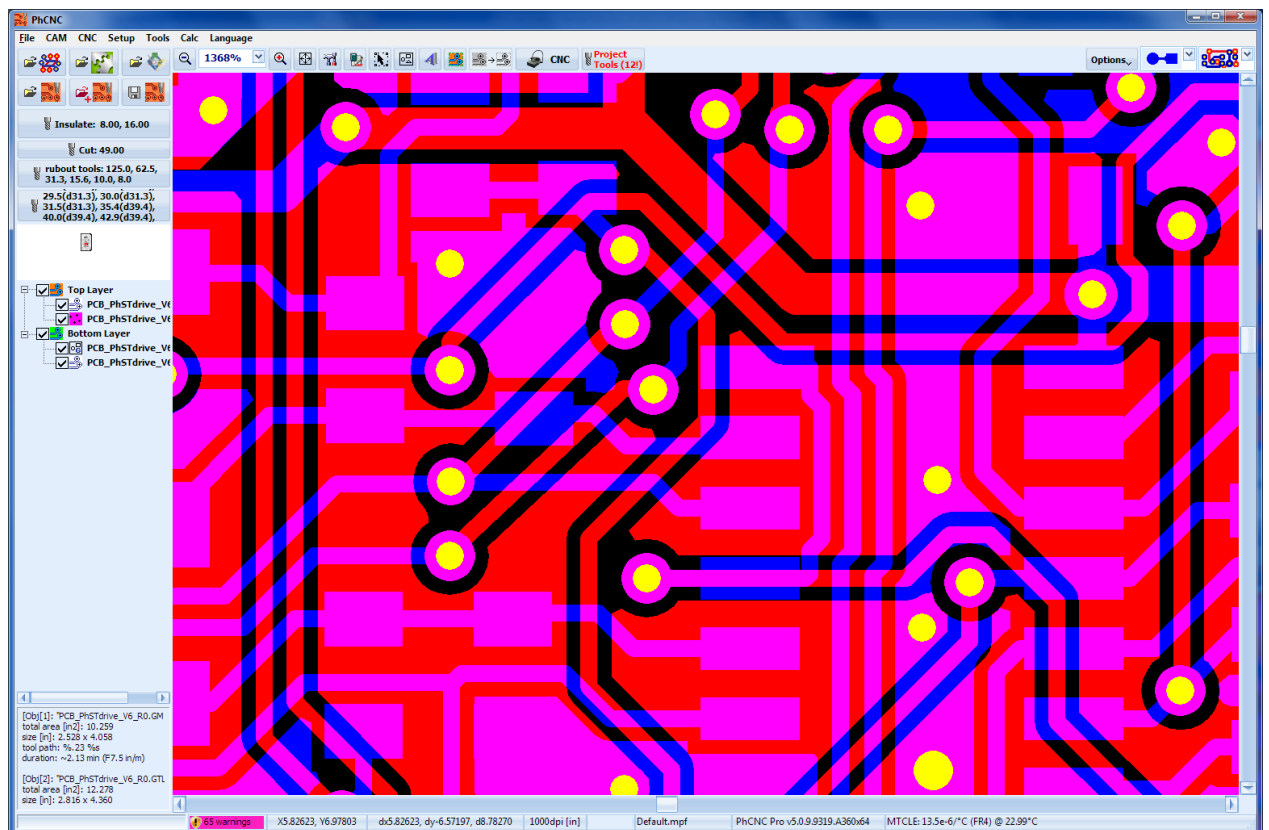
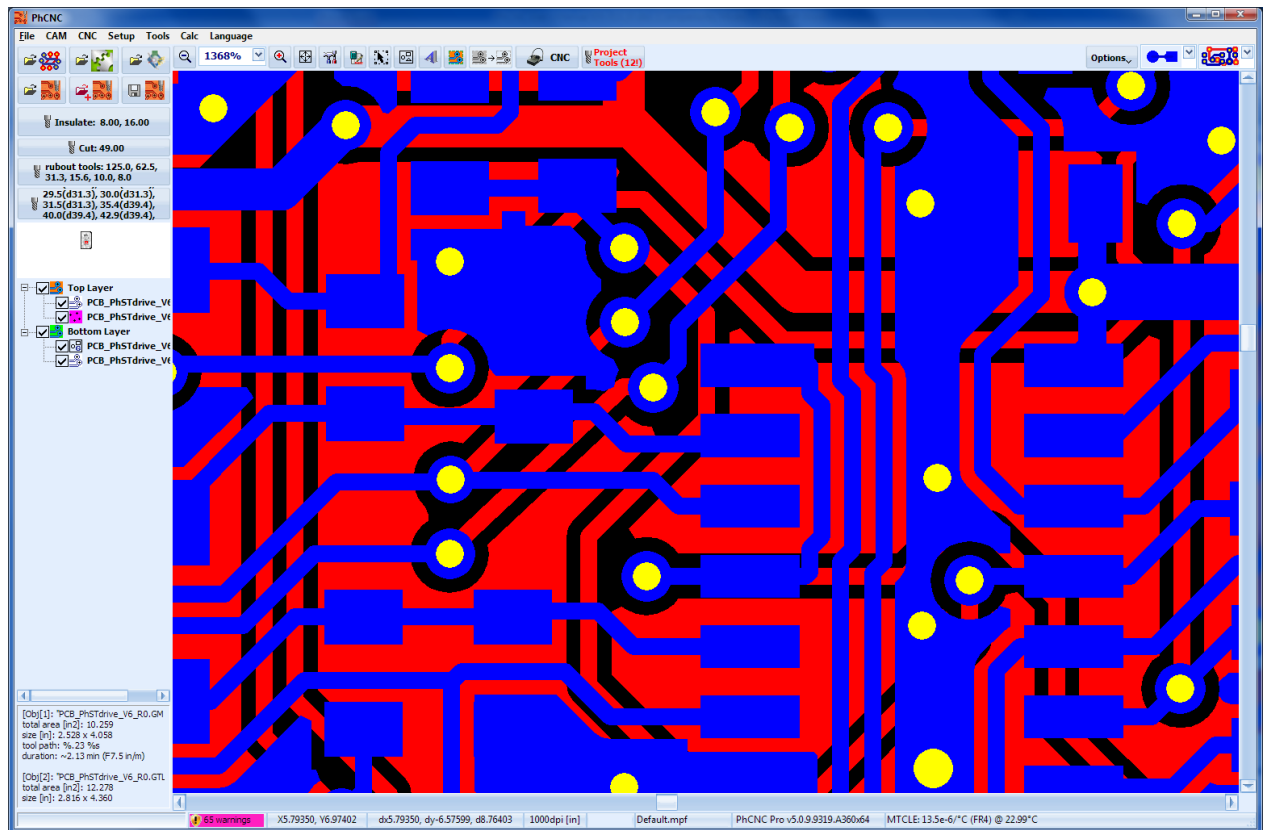
Drop down  is for controlling the way objects are displayed on the screen and will give you the following choices:

- Selecting  image will display the tool path center line only
- Selecting  image will display the tool path & real thickness
- Selecting  image will display the tool path & real thickness & center line
- Selecting  image will display the tool path & real thickness & Gerber
- Selecting  image will display the tool path & real thickness & center line & Gerber
- Selecting  image will display the Gerber only
- Selecting  image will display the tool path, center line only & Gerber vectors
- Selecting  image will display the tool path & real thickness & Gerber vectors
- Selecting  image will display the tool path & real thickness & center line & Gerber vectors
- Selecting  image will display the tool path & real thickness & Gerber vectors & Gerber
- Selecting  image will display the tool path & real thickness & center line & Gerber vectors & Gerber
- Selecting  image will display the Gerber vectors & Gerber

The options drop down menu gives you the following options:

- Show spindle path - this PhCNC LSSE option will show/hide spindle positioning lines generated in CNC mode. Active when you return from CNC mode. Useful for positioning objects on the desktop.
- "Show thumb tacks" - this PhCNC LSSE option will show / hide thumbtacks and is discussed in detail in section 2.14
- "Design rules violation" - this PhCNC LSSE option will activate/deactivate the DRV and is discussed in detail in section 2.13
- "Forced Insulation" - this PhCNC LSSE option will activate/deactivate the forced insulation and Calculate tool paths used in "forced insulation". If the tool paths are never calculated the first activation will do so. If DRV function wasn't activated it will start it. This is discussed in detail in section 2.23
- "Full Rubout" - this PhCNC LSSE option will activate/deactivate the full rubout and is discussed in detail section 2.15
- "Transparent Mode" - this PhCNC LSSE option will give you a transparent top layer and the ability to see top and bottom layers at the same time. This option is useful for review/inspection of double-sided projects. Below is an example of the usage.

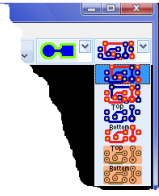




With "Transparent Mode" active it is recommended to turn off the tool path traces (☑) and enable Top & Bottom layers (☑).

Note:

This option has no effect if a single layer is enabled.

Drop down  is for visualization modes.

Buttons  &  are for a special visualization mode called "copper view". In this mode the screen shows what should be the final result after executing all machining programs. It uses close to real colors to show copper and typical base material. In CNC mode of PhCNC LSSE these buttons are inactive!

Buttons: , ,  &  are for review of top and bottom layers:

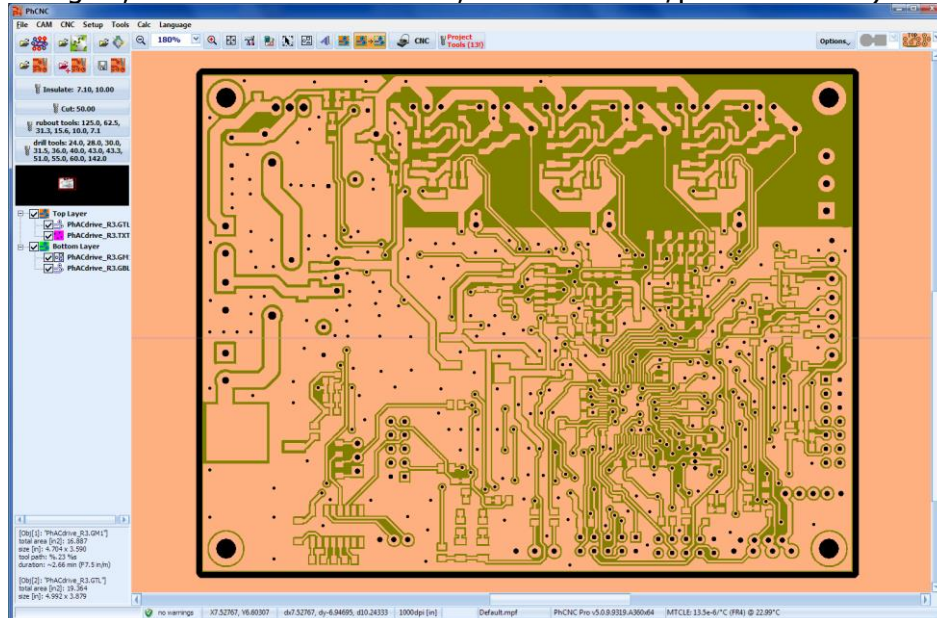
-  - top, bottom, drill & mechanical (natural layers order)
-  - bottom, top, drill & mechanical (bottom layer on top)
-  - top, drill & mechanical (bottom layer hidden)
-  - bottom, drill & mechanical (top layer hidden)

by default,  is active.

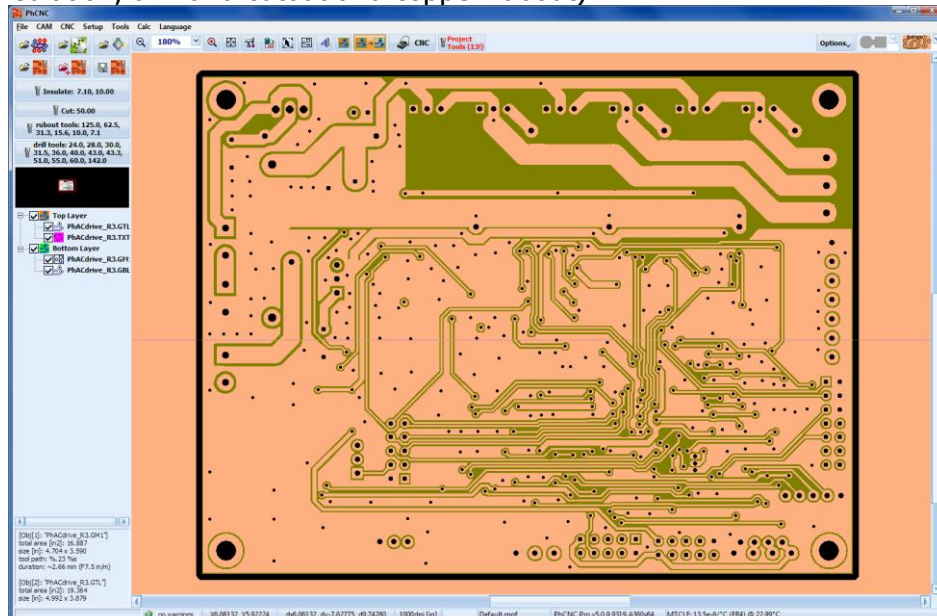
In CNC mode of PhCNC LSSE all these buttons are inactive!

Below are examples of "copper view":

Top layer (rubout region, cutout contour with tabs, drill and traces/pads insulation):



Bottom layer (insulation, drill and cutout and copper rubout):



All buttons above control the visualization of the project. They do not affect the settings or the machining process in CNC mode.

The following Quick access tool bar buttons are accessible on the tool bar:

Button  is for enabling the panoramic view window

Button  is for the setup of PhCNC LSSE

Button  is to access the built-in calculator

Button  is to enter select objects mode (selected objects can be moved, cloned, rotated, grouped in a single object, ungrouped, and deleted)

Button  is to enter tool diameter compensation selection mode (used in the mechanical layer)

Button  is to enter rubout regions selection mode

Button  is to start the rubout regions tool paths calculation

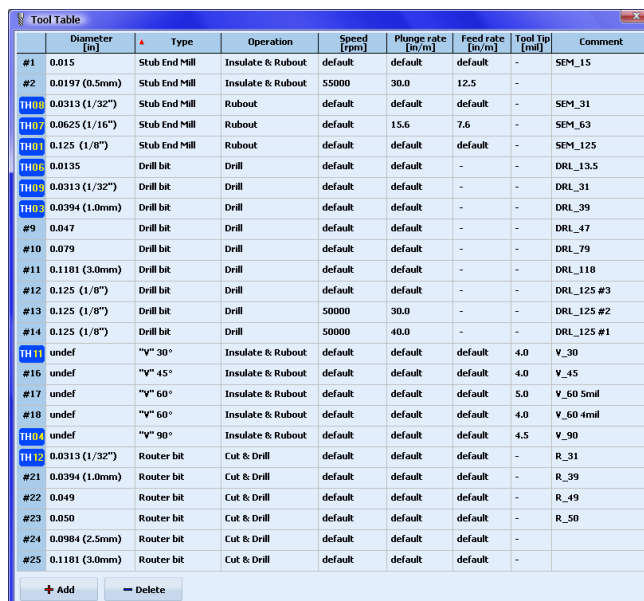
Button  is to switch between CAM/CNC modes

Button  is to enter the project tools list

2.4 Tools

In order to make the operation of PhCNC LSSE easier and to organize the tool settings we have tool tables. In the tool table you may enter all tools that are available for use. For each tool you may enter size (diameter [inch]), type, useful operations, Plunge Rate, Feed Rate, Speed [RPM]

To edit the main tool table click on "Tools" in the upper tool bar and select tool table from the drop down. The tool table will then open as a window as shown below:



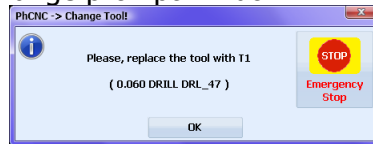
	Diameter [in]	Type	Operation	Speed [rpm]	Plunge rate [in/m]	Feed rate [in/m]	Tool Tip [in]	Comment
#1	0.015	Stub End Mill	Insulate & Rubout	default	default	default	-	SEM_15
#2	0.0197 (0.5mm)	Stub End Mill	Insulate & Rubout	55000	30.0	12.5	-	
TH09	0.0313 (1/32")	Stub End Mill	Rubout	default	default	default	-	SEM_31
TH07	0.0625 (1/16")	Stub End Mill	Rubout	default	15.6	7.6	-	SEM_63
TH01	0.125 (1/8")	Stub End Mill	Rubout	default	default	default	-	SEM_125
TH06	0.0135	Drill bit	Drill	default	default	-	-	DRL_13.5
TH05	0.0313 (1/32")	Drill bit	Drill	default	default	-	-	DRL_31
TH03	0.0394 (1.0mm)	Drill bit	Drill	default	default	-	-	DRL_39
#9	0.047	Drill bit	Drill	default	default	-	-	DRL_47
#10	0.079	Drill bit	Drill	default	default	-	-	DRL_79
#11	0.1181 (3.0mm)	Drill bit	Drill	default	default	-	-	DRL_118
#12	0.125 (1/8")	Drill bit	Drill	default	default	-	-	DRL_125 #3
#13	0.125 (1/8")	Drill bit	Drill	50000	30.0	-	-	DRL_125 #2
#14	0.125 (1/8")	Drill bit	Drill	50000	40.0	-	-	DRL_125 #1
TH11	undef	"V" 30°	Insulate & Rubout	default	default	default	4.0	V_30
#16	undef	"V" 45°	Insulate & Rubout	default	default	default	4.0	V_45
#17	undef	"V" 60°	Insulate & Rubout	default	default	default	5.0	V_60 5mil
#18	undef	"V" 60°	Insulate & Rubout	default	default	default	4.0	V_60 4mil
TH04	undef	"V" 90°	Insulate & Rubout	default	default	default	4.5	V_90
TH12	0.0313 (1/32")	Router bit	Cut & Drill	default	default	default	-	R_31
#21	0.0394 (1.0mm)	Router bit	Cut & Drill	default	default	default	-	R_39
#22	0.049	Router bit	Cut & Drill	default	default	default	-	R_49
#23	0.050	Router bit	Cut & Drill	default	default	default	-	R_50
#24	0.0984 (2.5mm)	Router bit	Cut & Drill	default	default	default	-	
#25	0.1181 (3.0mm)	Router bit	Cut & Drill	default	default	default	-	

Buttons  &  are used to add or delete a selected tool by left clicking on it in the list. Selecting a tool in the list will also allow you to edit the tools parameters. The description of each parameter field is as follows:

- **Diameter** is the cutting diameter of the tool. You can select it from drop down list or enter it manually. Entries are in inches, if the entry is bigger than 1.0, it is accepted as mils. For example, if you enter 15 – this will be 0.015"
- **Type** is the type of the tool. It is recommended to fill this field. It helps during the selection of tools for certain operations. The Possible types are "undefined", "Drill bit", "Router bit", "Stub End Mill", "V 25°", "V 30°", "V 45°", "V 60°", "V 90°", "V 100°" and "V 120°"
For tools of type "V", the track size (diameter column) depends on the tool penetration into the material. It is recommended to select "undef" (or enter 0.0)!
- **Operation** is to show the possible usage of the tool. The choices are "undefined", "Insulate", "Rubout", "Cut & Drill", "Drill", and "Cut". For example, if the type of tool is a "Router Bit", then

the possible choice of Operation will be limited to: "Cut & Drill", "Drill" and "Cut"; The reason for this is a router bit can be used for drilling holes larger than its diameter.

- **Speed** is for setting the RPM of the spindle for this tool. If the choice is "default", the setting speed is selected from global parameters table for this operation (see 3.4 CNC process control settings). Possible settings are 5,000...60,000 and 5,000...100,000 RPM depend of the model.
- **Plunge Rate** is for setting the speed of Z axis toward the table, e.g. the speed that tool will penetrate the material. If selection is "default" the setting is from global table for selected operation (see 3.4 CNC process control settings). Possible values are 40...0.1 in/min.
- **Feed Rate** is for setting the tool movement speed in XY plane (in/min). If "default" is selected it takes the speed setting from global table (see 3.4 CNC process control settings). Possible settings are 15...0.1 in/min. This parameter has no meaning for "Drill bit" type.
- **Comment** is a field for making notes, maximum length 256 characters. Notes are shown in the tool change prompt window.



- **Tool Life [in]/#** is a field that shows maximum tool usage acquired by the manufacturer specs or by own experience

In this column of the tool table (**Tool Life [in]/#**) you can enter a value that if exceeded will prompt you to change tool. It can contain the information below:

- total length of cut for this tool (for routers, end mills and V-tools)
- total number of material entries for this tool (for drill bits and routers)

It can be set for every tool (if needed) and will prompt the operator that the limit set is exceeded. PhCNC LSSE records both (total number of material entries and total length of cutting) for routers and end mills. It records only the number of drill cycles for the drill tools.

Example:

	Diameter [in]	Type	Operation	Speed [rpm]	Plunge rate [in/min]	Feed rate [in/min]	Tool Life [in]/#	Comment
TH 12	0.035	Drill bit	Drill	default	default	-	10000	DRL_39
#13	0.047	Drill bit	Drill	default	default	-	15000	DRL_47
#14	0.079	Drill bit	Drill	default	default	-	15000	DRL_79
#15	0.1181 (3.0mm)	Drill bit	Drill	default	default	-	15000	DRL_118
#16	0.125 (1/8")	Drill bit	Drill	default	default	-	15000	DRL_125 #1
#17	0.125 (1/8")	Drill bit	Drill	default	default	-	15000	DRL_125 #2
#18	0.125 (1/8")	Drill bit	Drill	default	default	-	15000	DRL_125 #3
#19	undef	"V" 30°	Insulate & Rubout	default	default	3.5	2000 in	V_30
TH 16	undef	"V" 45°	Insulate & Rubout	45000	default	default	6.5	2000 in
#21	undef	"V" 45°	Insulate & Rubout	default	default	default	4.5	2000 in
#22	undef	"V" 45°	Insulate & Rubout	default	default	default	4.0	2000 in
TH 22	undef	"V" 60°	Insulate & Rubout	default	default	5.0	2000 in	V_60
#24	undef	"V" 90°	Insulate & Rubout	default	default	default	4.0	500 in
TH 10	undef	"V" 90°	Insulate & Rubout	default	default	default	4.5	500 in
#26	0.0313 (1/32")	Router bit	Cut & Drill	default	default	default	-	R_31
TH 13	0.0394 (1.0mm)	Router bit	Cut & Drill	default	default	default	-	10000 in
#28	0.049	Router bit	All	default	default	default	-	R_49
TH 14	0.055	Router bit	Cut & Drill	default	default	default	-	R_50
#30	0.0984 (2.5mm)	Router bit	Cut & Drill	default	default	default	-	
#31	0.1181 (3.0mm)	Router bit	Cut & Drill	default	default	default	-	

Shown above is the column in the tool table that handle the tool life (in red circle).

Clicking on the column for any tool (in green circle) will allow you to enter needed parameters in the window below. Holes/plunges represents number of entries in the material. Current tool usage is listed in the lower portion of this window.

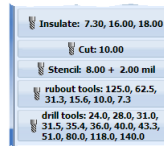
Important:

You may sort out the table by clicking on the column title.

("▼" shows which column is selected for sorting order).

Second click on the column title changes the order of sorting (ascending to descending and back).

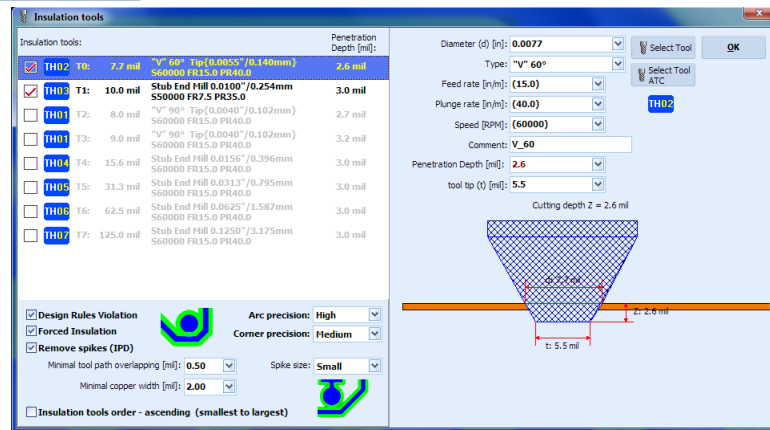
In the left portion of the screen (PhCNC LSSE in CAM mode) you will find buttons for tools and tool tables for each operation.



2.4.1 "Insulate" Tools

PhCNC LSSE offers traces and pads insulation using up to 8 different tools. The selection and settings of the tools are performed in the "Insulation tools" window.

To select it click on



In the left part of this window you will find the tools entered as tools for insulation. In the right part are the parameters of the currently selected tool.

During the insulation process, PhCNC LSSE is using only the selected tools (☑).

There is a requirement for tools diameters. The minimum difference in the tools' diameters must be more than 1 mil (0.001 inch).

It is allowed to use a V tool with different penetration, thus achieving different tool diameters. In this case the tool will be listed as two different tools.

The number of tools that will be used depends only on the project structure and is determinate by the operator. Typical number is 2 or 3 tools. Very complicated designs may need 4 or even 5 tools. There are some samples below that are giving you a picture of different number of tools used in the same design. We create a possibility to use up to 8 tools, just to cover potential needs in the future. Limiting the number of tools has its own advantages, for example less tool changes.

PhCNC LSSE optimizes the usage of the small tools (usually far more expensive and with shorter life). This creates conditions to use them only in cases, that the larger tool is unable to do the insulation.

In "Forced Insulation" mode, PhCNC LSSE will always use the smallest tool available.

Additional options related to insulation process

• Design Rules Violation (DRV)

Activates DRV option that checks the project for regions that have spacing smaller than the smallest insulation tool. Normally these regions (or simply spots) will not be insulated. In most cases the problem can be resolved using the option "Forced insulation". Details about DRV option are described under "2.13 Design Rules Violation".

• Forced Insulation

This option allows the insulation of critical regions/spots with small compromise, that should be a matter of decision of the operator of the system. The option description is under "2.23 Forced Insulation".

• Remove Spikes (IPD)

"Remove Spikes" option (renamed from IPD) finds the spots that have extremely small copper edges that can be a source of future shorts on the ready board. This option eliminates them. Detailed info is under "2.24 Remove Spikes (IPD)"

The option "Insulation tools order - ascending (smallest to largest)", sets the order of tools usage. When not selected (default), the insulation process starts with the largest insulation tool and ends with the smallest, thus creating additional savings of the smallest tool life.

If the option is selected the process will go in opposite order (smallest to largest). This create kind of convenience when a V tool is used with different penetration levels.

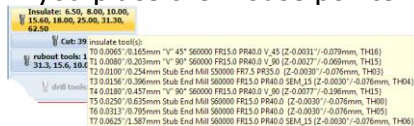
If the selected tool is type "V" with "undef" diameter, the current diameter will remain the same (as shown 0.0080").

When you select an insulation tool from the main table, only tools that have "Insulate & Rubout" and "Insulate" in the "Operation" column will be present in the list.

	Diameter (in)	Type	Operation	Speed (rpm)	Plunge rate (in/min)	Feed rate (in/min)	Tool Tip (in)	Tool Life (min)	Comment
[H10]	0.010	Stub End Mill	All	50000	35.0	7.5	-	3000 in	
[H11]	0.016	Stub End Mill	All	default	default	default	-	-	SEH_15
[H12]	0.0313 (1/32")	Stub End Mill	Insulate & Rubout	default	default	default	-	-	
[H13]	0.0625 (1/16")	Stub End Mill	All	default	default	default	-	-	SEH_63
[H14]	0.125 (1/8")	Stub End Mill	All	default	default	default	-	-	SEH_125
[H15]	0.0125	Drill bit	Drill	default	default	-	-	7500	DRL_12.5
[H16]	0.022	Drill bit	Drill	default	default	-	-	10000	DRL_21.65
[H17]	0.025	Drill bit	Drill	default	default	-	-	10000	
[H18]	0.0313 (1/32")	Drill bit	Drill	default	default	-	-	10000	DRL_31
[H19]	0.035	Drill bit	Drill	default	default	-	-	10000	DRL_39
[H20]	undef	"V" 45°	Insulate & Rubout	default	default	default	4.5	2000 in	V_45
[H21]	undef	"V" 60°	Insulate & Rubout	default	default	default	5.5	2000 in	V_60
[H22]	undef	"V" 90°	Insulate & Rubout	default	default	default	4.0	-	V_90
[H23]	0.055	Router bit	Cut & Drill	default	default	default	-	-	R_50

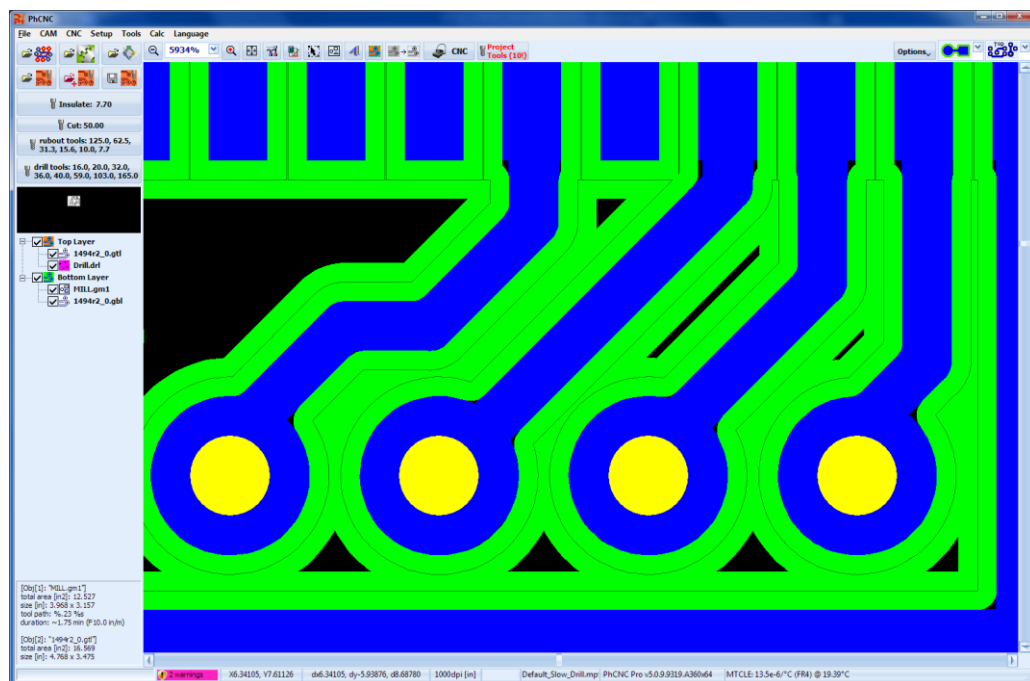
The information contains actual parameters (If "default" is selected you will see the value from the global table). These are the parameters that will be used for the program generation for this process.

If you place the mouse pointer on the tool button, you will see the information for the selected tool.



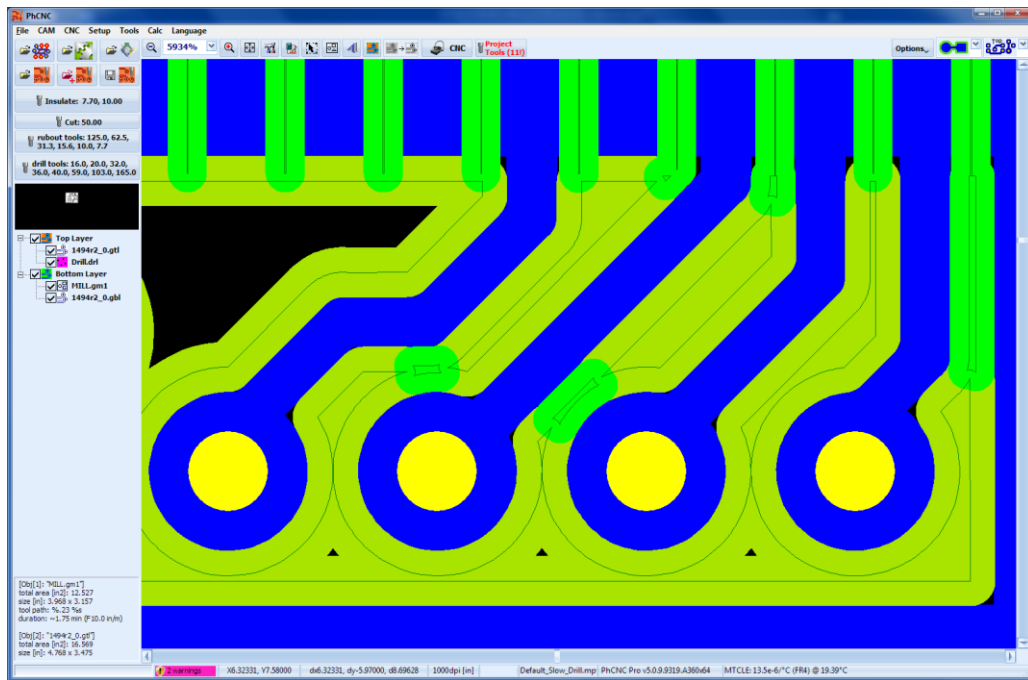
As we mentioned above a PCB design can be insulated with one two or more tools. Shown below is a single design machined with 1,2 and 3 tools

Single tool (7.7 mil V60):



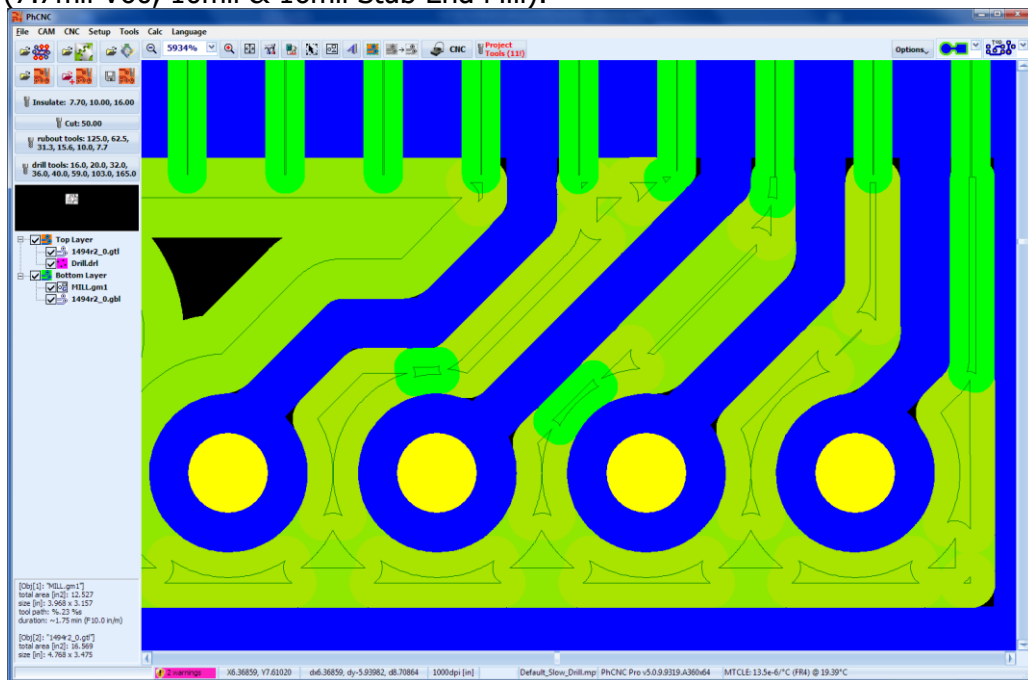
tool path T0: 511.990 inch

Two tools (7.7mil V60, 10.0mil Stub End Mill):



tool path T0: 114.540 inch
 tool path T1: 462.920 inch

Three tools (7.7mil V60, 10mil & 16mil Stub End Mill):

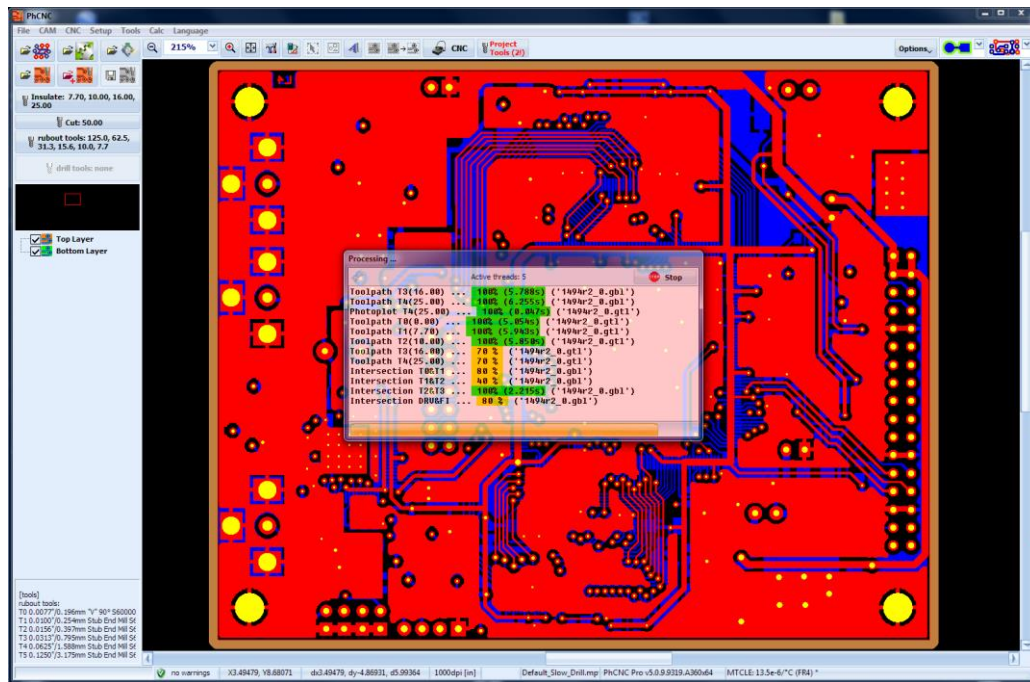


tool path T0: 114.540 inch
 tool path T1: 177.600 inch
 tool path T2: 389.940 inch

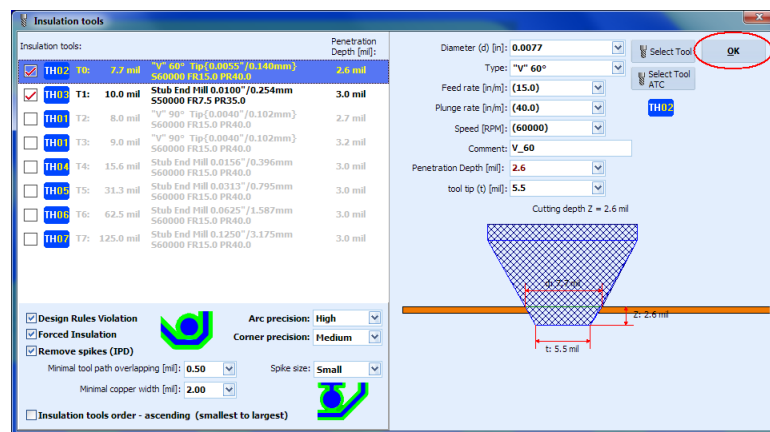
Important:

The tool paths are calculated with maximum precision. Using more than one insulation tool will give you the same precision as with the smallest one but will significantly reduce the usage of the smaller tools when possible.

Tool paths calculation process can be stopped using the "Stop" button in the progress window shown below.



In case of interrupted tool paths calculations, they can be resumed by pressing "OK" button in "Insulation tools" window.



If there are no changes of the tools and settings resuming the process will just complete the calculations interrupted by using the "Stop" button. In case that any changes are made the tool paths calculation will start all over again.

In all cases entering the CNC mode will initiate all needed calculations if for some reason they are not completed in CAM mode: insulation, rubout, stencil, cutout, FI, IPD.

2.4.2 "Cut" Tool

To select the cutout tool left click on the button.

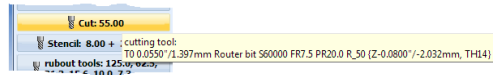
The same window that was used for Insulation tool appears, but this time only shows tools with "Cut" and "Cut & Drill" in the "Operation" column.

To select a tool from main table click on the button.

Diameter (in)	Type	Operation	Speed (rpm)	Plunge rate (in/m)	Feed rate (in/m)	Tool Tip (in)	Tool Life (in/#)	Comment
TH03 0.010	Stub End Mill	All	50000	35.0	7.5	-	3000 in	
TH04 0.016	Stub End Mill	All	default	default	default	-	-	SEM_15
TH06 0.0625 (1/16")	Stub End Mill	All	default	default	default	-	-	SEM_63
TH07 0.125 (1/8")	Stub End Mill	All	default	default	default	-	-	SEM_125
TH14 0.055	Router bit	Cut & Drill	default	default	default	-	-	R_50

If the selected tool uses default parameters its actual value is in brackets. For example if the RPM is set to default (60000) will be shown. This means that, according to current tool profile, the spindle speed will be 60,000 RPM because the global table RPM value was set to 60,000.

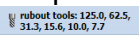
If you place the mouse pointer on the tool button, you will see the information for the selected tool in a popup window.

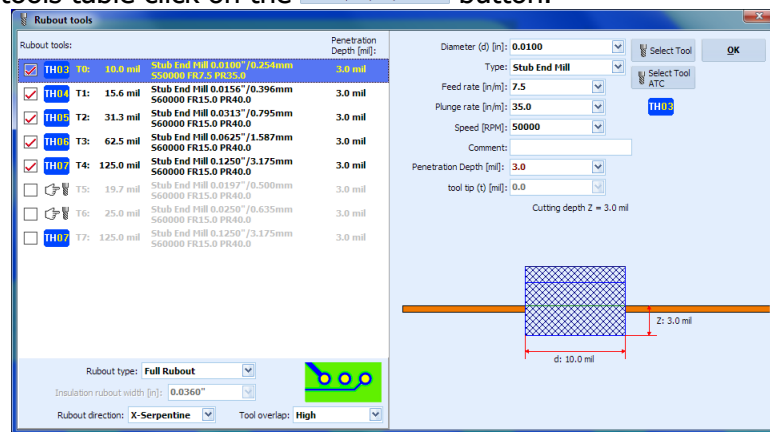


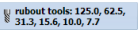
The information contains the actual parameters. If the selected tool uses default parameters, its actual value is in brackets. For example, if the RPM is set to default (60000) will be shown. This means that, according to current tool profile, the spindle speed will be 60,000 RPM because the global table RPM value was set to 60,000. These are the parameters that will be used for the program generation for this process.

2.4.3 "Rubout" Tools

PhCNC LSSE now allows usage of up to 8 additional tools for the "copper rubout" process in addition to 8 tools available in "insulation" process. This way the maximum number of tools used in rubout process can go up to 16 (8 insulation tools and 8 rubout dedicated tools). Copper rubout process is actually an extension to the process of insulation and can be considered or rejected depend on the final goals of the design. We consider the total number of tools quite excessive but like to have it in order to meet the requirements in technology that may come up in the future. This will allow us to meet more complicated needs without making structural changes in the program. Currently a medium complicated project needs total of 4 to 5 tools to execute the processes of Insulation and Copper rubout.

To select the Rubout tools table click on the  button.



Important: In the table above, you have to enter the desired copper rubout tools. PhCNC LSSE software will use all insulation tools declared as the smallest rubout tools. If the rubout tools table contain any tool smaller than largest insulation tool it will not be used in the process. In the rubout tools button  you can see the list of the tools that will be used in the process. The requirement for the tool diameters is the same as for Insulation tools. The difference in diameters must be larger than 1 mil. Only tools with check mark will be included in the process.

The process has 3 options for the rubout direction

- X serpentine
- Y serpentine
- Conical

They determine the moving direction of the tool when large copper areas are processed.

PhCNC LSSE offers 2 types of copper rubout:

- Full rubout

A standard (and recommended) process, that removes all excessive copper in the entire project or in the selected area of it.

- Insulation rubout

This process removes the copper using a preset distance (when available) from the pads and traces. This option saves machining time and tools life, but in many applications may create conditions for the circuits to behave differently from the original design. Commonly not recommended for any circuits using 100 KHz and up signals. In "Auto" mode it clears the copper to the size of the larger rubout tool (when possible).

Rubout process starts with the smallest tool in ascending diameter order. This process is usually next after Insulation and this order saves one tool change.

The information contains the actual parameters. If the selected tool uses default parameters, its actual value is in brackets. For example, if the RPM is set to default (60000) will be shown. This means that, according to current tool profile, the spindle speed will be 60,000 RPM because the global table RPM value was set to 60,000.

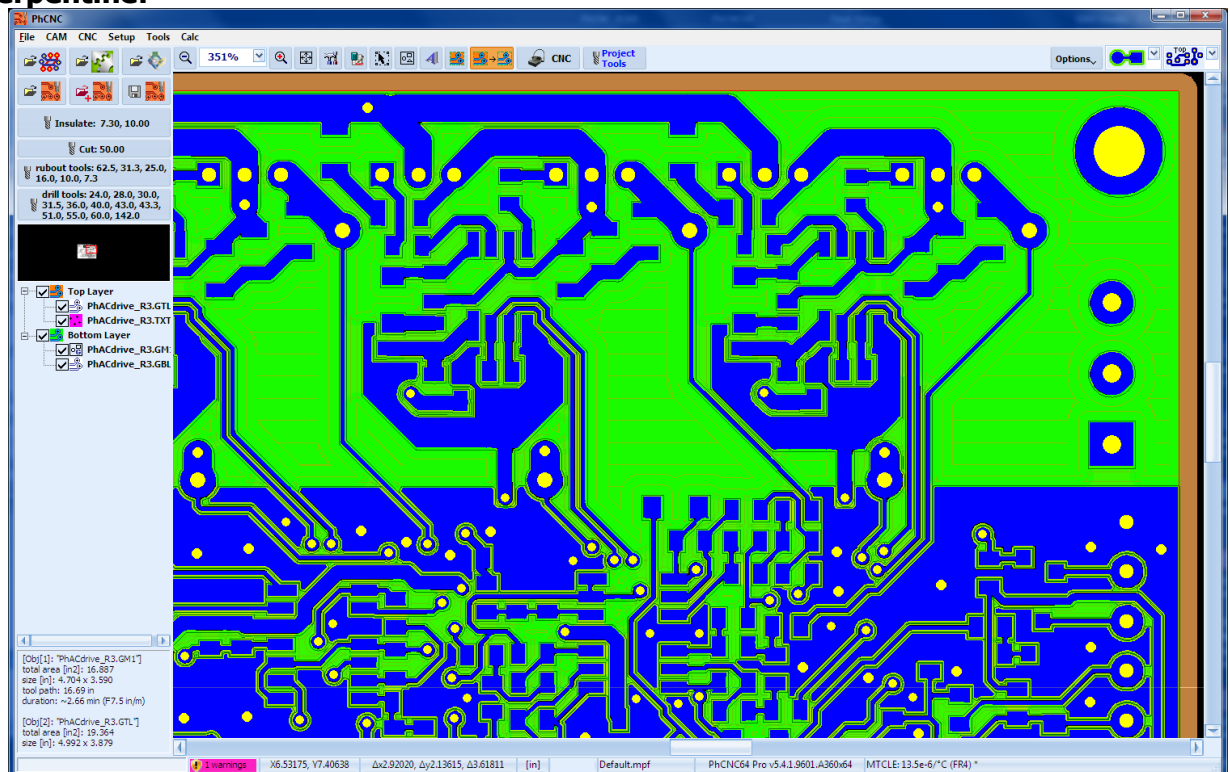
Important:

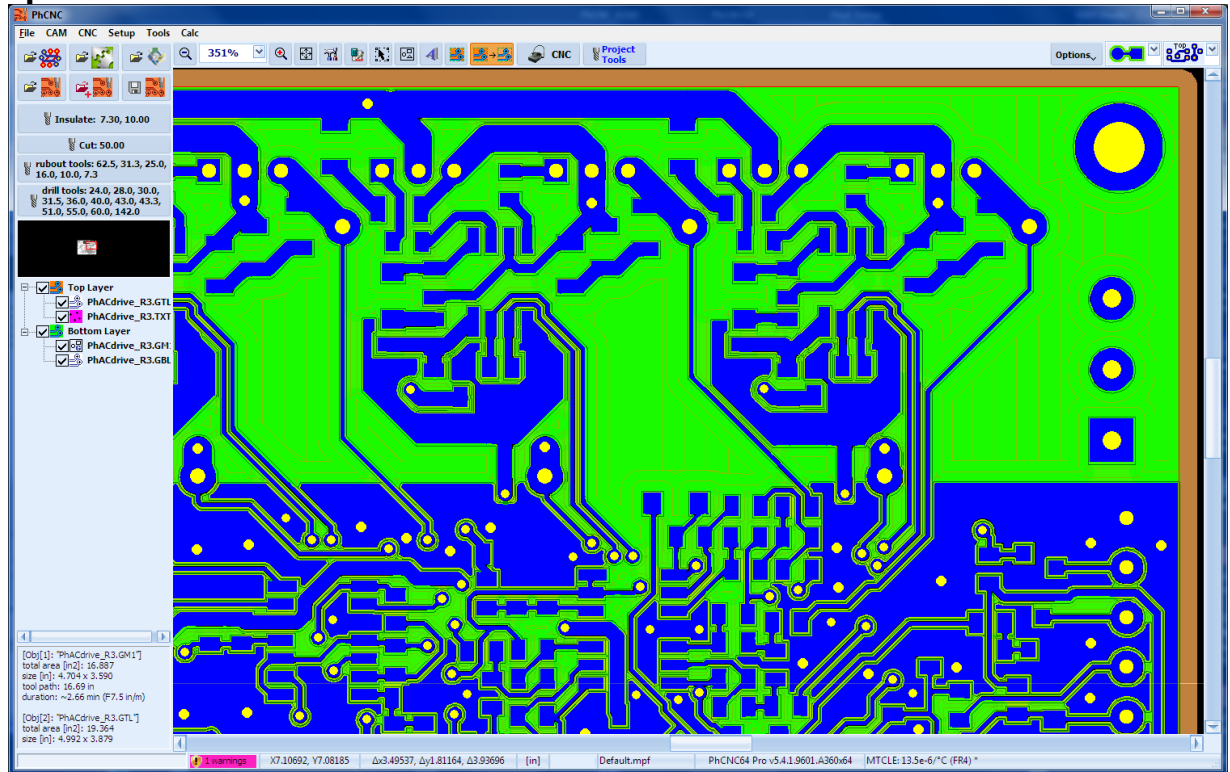
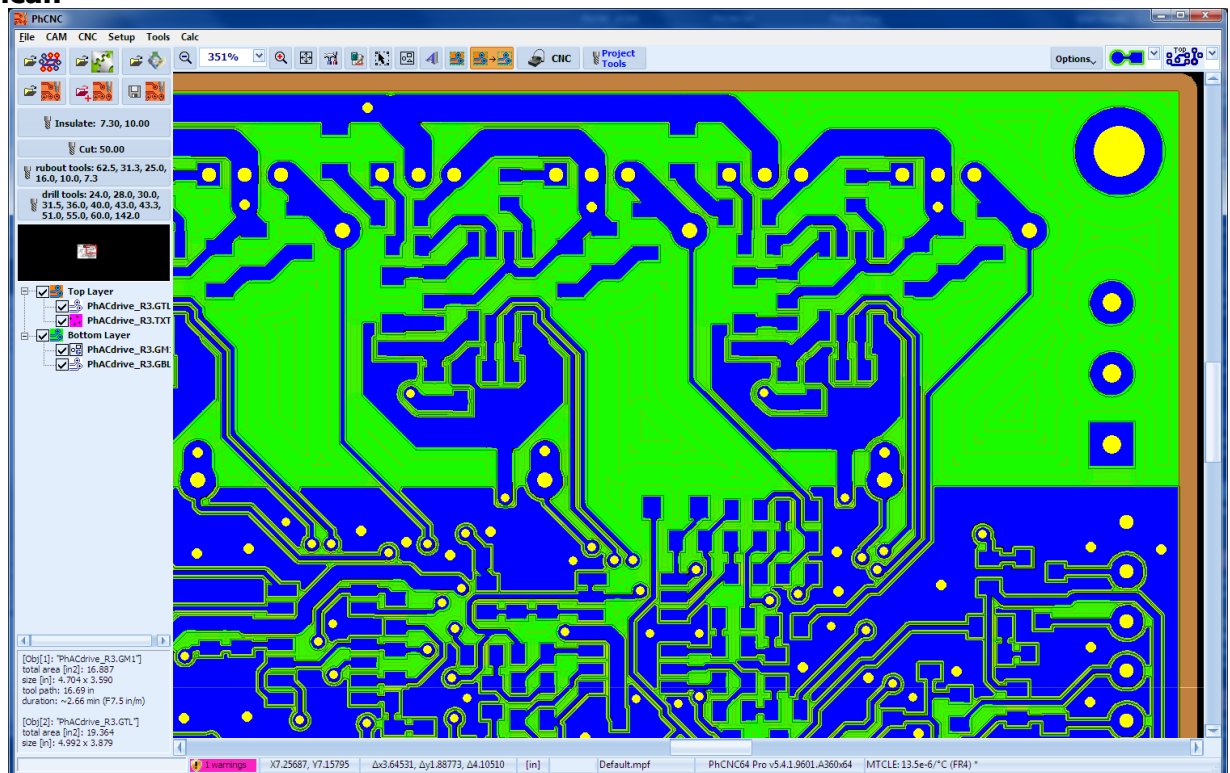
When Full rubout is selected, it is not needed to select the option Remove spikes. Full rubout covers Remove spikes process. If you change your mind after performing Remove spikes this will not create any problem. Simply the time used for the Remove spikes will be a wasted time.

Examples:

Below you can see the patterns of tool paths in different modes.

X-Serpentine:



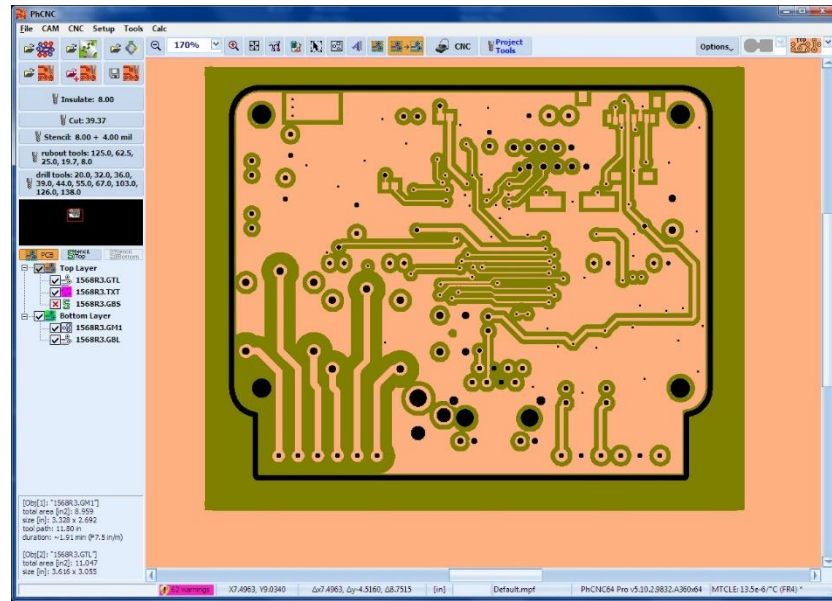
Y-Serpentine:**Conical:****Notes:**

"X-Serpentine" and "Y-Serpentine" are recommended for usage in PCB projects that contain significant surface of copper to be removed.

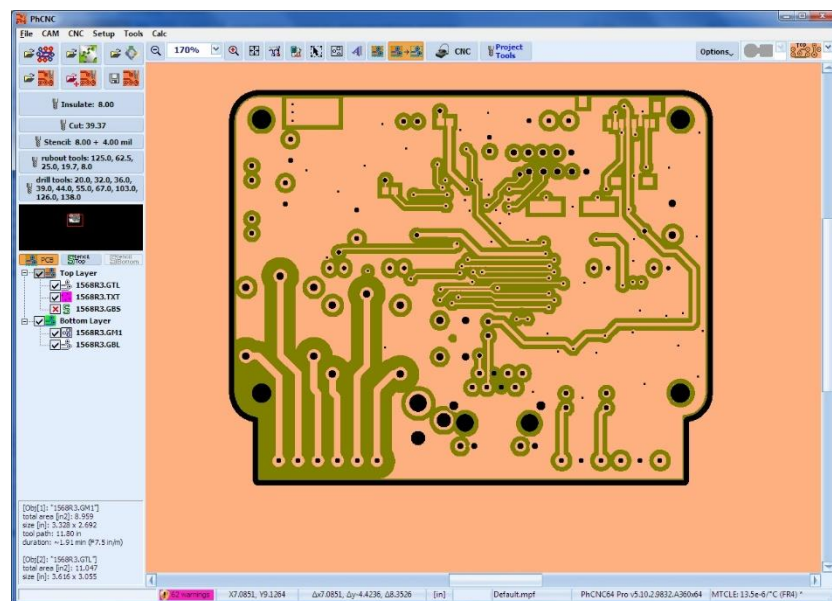
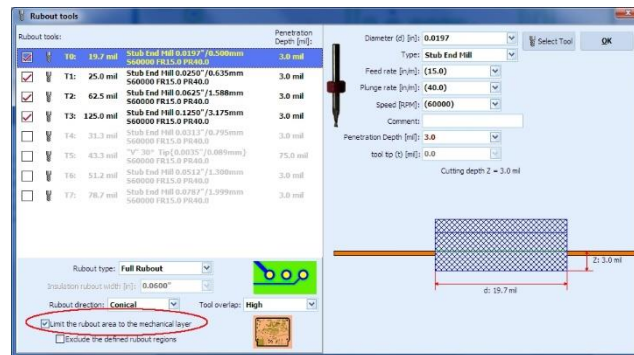
"Conical" is more efficient for designs that have "poured copper" or when "Insulation rubout" has been used.

The options below **"Limit the rubout area to the mechanical layer"** Allows to use the area defined by the "mechanical layer" as a "copper rubout" area. Very convenient option for boards with more complicated mechanical shapes and internal cuts.

When "Limit the rubout area to the mechanical layer" is not checked, the rubout area is rectangular:

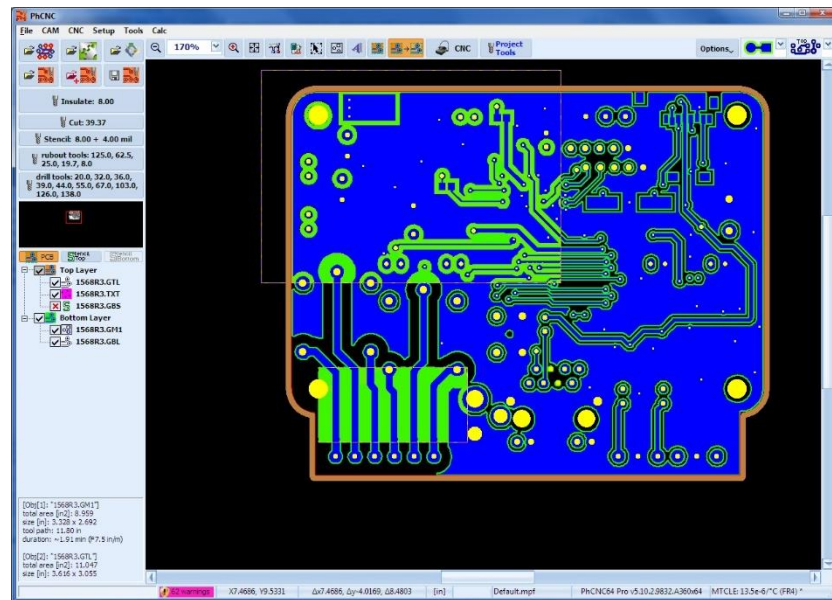


When "Limit the rubout area to the mechanical layer" is checked, the mechanical layer is used like rubout region:

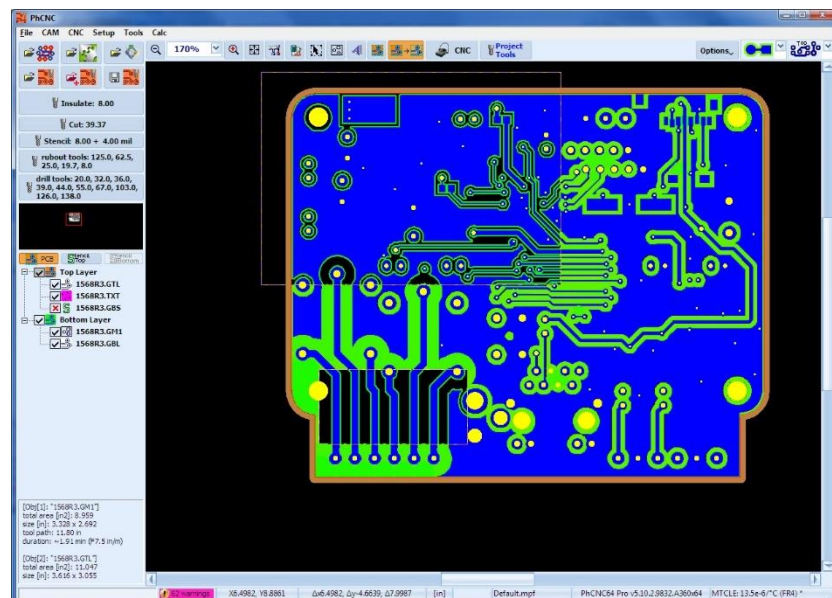
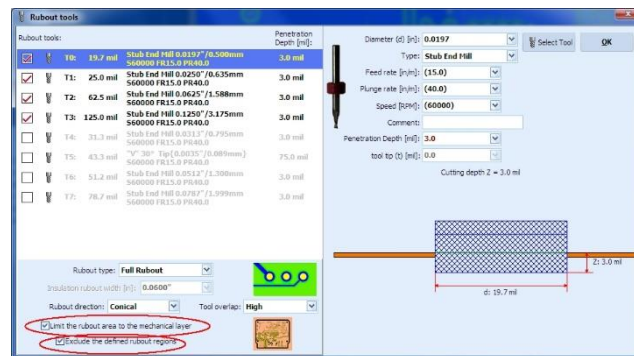


The option "Exclude the defined rubout regions" has meaning, only when "Limit the rubout area to the mechanical layer" is checked.

For example, when "Limit the rubout area to the mechanical layer" is checked and "Exclude the defined rubout regions" is not checked:



Example, when "Limit the rubout area to the mechanical layer" is checked and "Exclude the defined rubout regions" is checked:

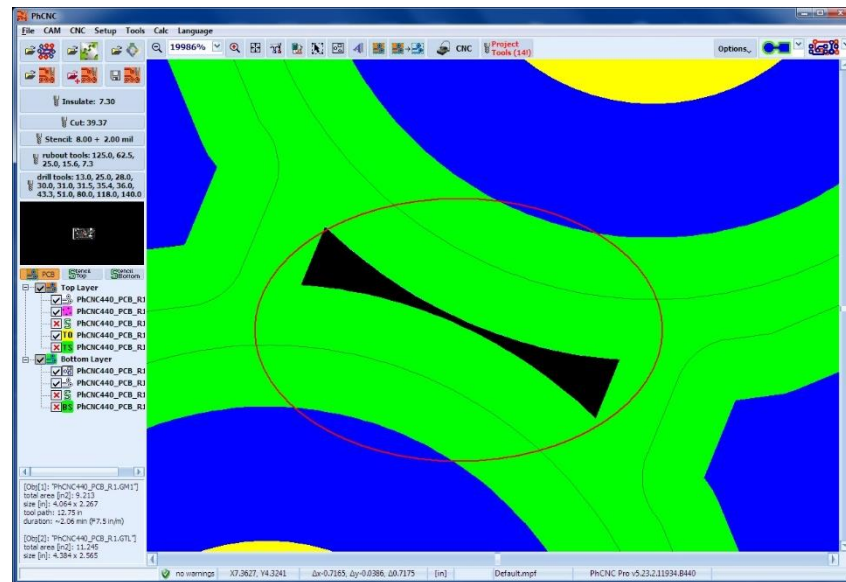


"Fish-tail SEM optimizations" menu option

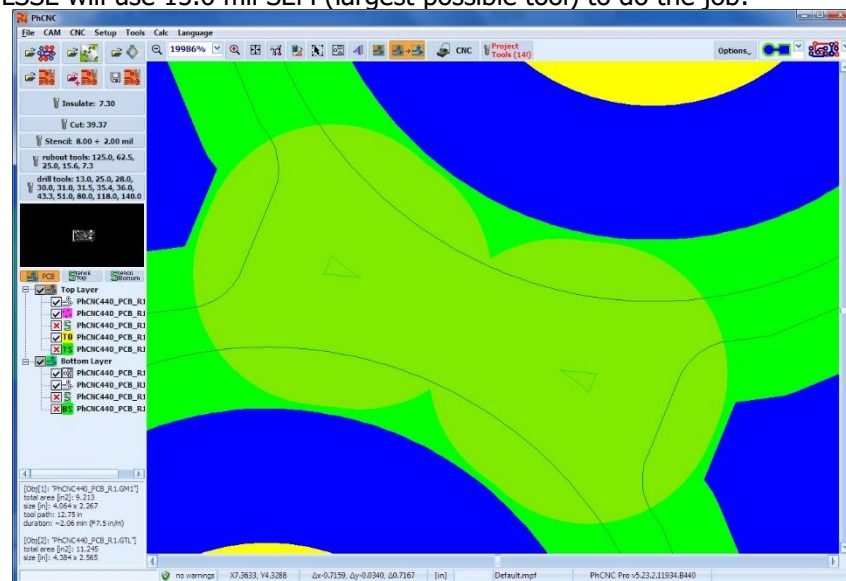
PhCNC LSSE always is trying to use the largest possible tool for insulation & rubout programs, but in case of using Fish-tail Stub End Mills (SEM, shown below) because of its geometry this can create some problems.



One rubout example:

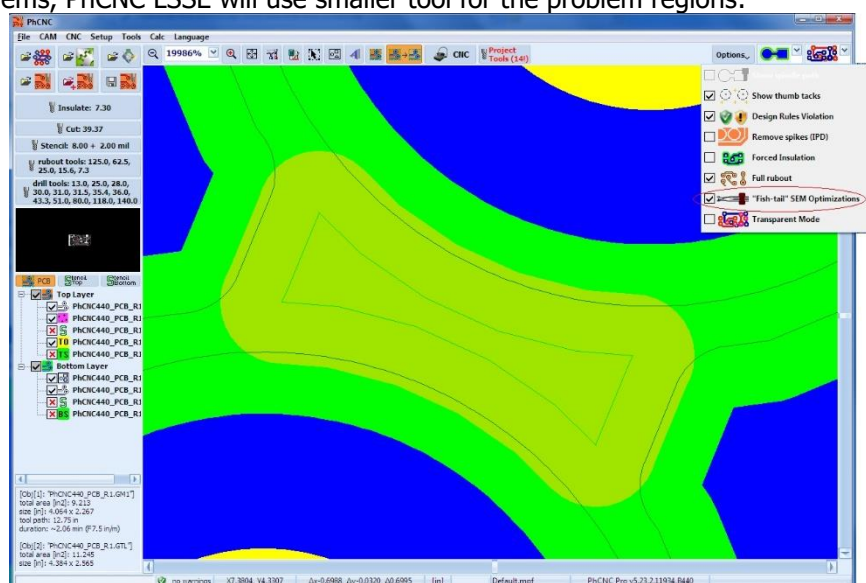


In this case, PhCNC LSSE will use 15.6 mil SEM (largest possible tool) to do the job:



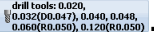
Because of fish-tail geometry - in the center of the very small regions is possible the copper to stay (Fish-tail tool needs movement of at least one R to work like standard SEM).

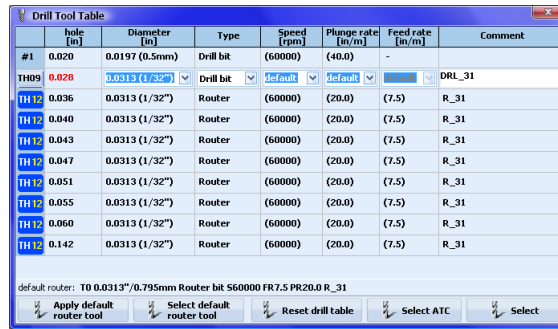
To avoid such problems, PhCNC LSSE will use smaller tool for the problem regions:



The the "Fish-tail SEM optimizations" option is enabled by default.

2.4.4 "Drill" Tool Table

To select the Drill tools table left click on .



#	hole [in]	Diameter [in]	Type	Speed [rpm]	Plunge rate [in/m]	Feed rate [in/m]	Comment
#1	0.020	0.0197 (0.5mm)	Drill bit	(60000)	(40.0)	-	
TH09	0.028	0.0313 (1/32")	Drill bit	(60000)	(20.0)	(7.5)	DRL_31
TH12	0.036	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
TH12	0.040	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
TH12	0.043	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
TH12	0.047	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
TH12	0.051	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
TH12	0.055	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
TH12	0.060	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
TH12	0.142	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31

default router: T0 0.0313"/0.795mm Router bit S60000 FR7.5 PR20.0 R_31

Buttons: Apply default router tool, Select default router tool, Reset drill table, Select ATC, Select

The column "hole [in]" can NOT be edited; it contains the hole sizes from your CAD system (usually Excellon2 format file).

You must select a tool for each hole size. In some cases, the operator may select to drill the hole using router bit. To use this option, you have to enter a router bit with diameter smaller than the hole. The example above shows what happens if you select a drill bit larger than the hole. The 2nd tool in the tool table has a slightly larger size drill bit then its hole and will be shown in Red. (If selected from the main tool table a prompt message will appear). The rest of the holes in the tool table, from 0.036" to 0.142" will use a Router bit with a diameter of 0.0313".

If some of selected tools use default parameters its actual value is in brackets (60000) this means that, according to current tool profiles, the spindle speed will be 60,000 RPM.

The new drill table contains a few new buttons related to usage of router tool for drilling of larger holes. If your project has a small number of larger holes, it is more efficient to drill them using router tool, thus saving tool expenses and tool changes during drill operations.

Button  selects the router tool that will be used to "drill" large holes.

Button  assigns all holes with diameter larger or equal to the default router diameter for "drilling" with selected router. If a router tool is not selected this button is inactive.

Button  resets all tool selections.

Important:

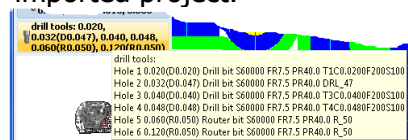
All tool tables and selections will be saved in the project files (.PHJ).

When you use "Open" to load an existing project you do not need to select your tools again.

When you use "Import" to add a project to one already loaded it will keep current settings and tool tables and ignore those of the imported project!

Rubout tool table is global, e.g., you do not need to create it if you start a new project or import an existing project.

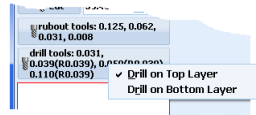
Drill tools table usually is specific for each project. Therefore, you have to select your tools and fill out the drill tools table for every new or imported project.



If you place the mouse pointer on the drill tools button, you will see the information for the selected tools listed.

Note:

Right click on the "drill tools" button will provide you a popup menu that allows you to change the drill layer. All drilling programs will move to the selected layer.



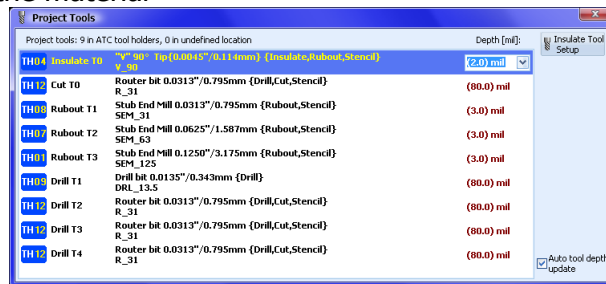
2.4.5 "Stencil" Tool (Pro version only)

"Stencil" button allows you to select a tool and set the parameters for it. For more details see 2.4.1.

2.4.6 Project Tools

The window "Project Tools" contains a list with all used tools in the project (for all operations). In the example below (when insulation tool is selected), the button opens a window that contains a list of tools, consisting of default "insulation" and "cut-out" tools and all specific tools involved in your project. The list contains information for each tool. Here is what information is included:

- Type of operation and sequence number in it (T0 ... Tn)
- Tool type description (Router Bit, Stub End Mill, Drill Bit, etc...)
- Penetration depth into the material



If "Auto tool depth update" check box is checked, changing the penetration depth of a tool will change the penetration of all tools in this operation. As an example, if you change the penetration of one tool in the "copper rubout" operation, the change will spread to the rest of the tools involved in this operation. If you like to have individual tool penetration depth for each tool this option should not be checked.

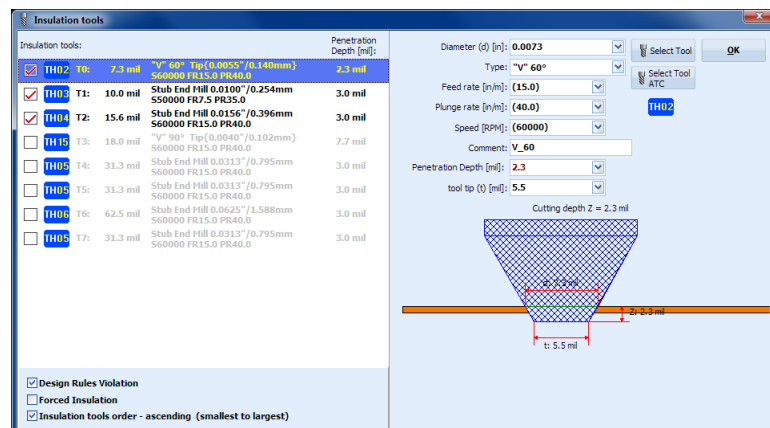
Here the operator will be able to make changes in the selected tool penetration depth (if needed). When the penetration depth values are in brackets, they are default values for this operation or tool. "V" types of tools have as default penetration the one calculated in the "insulation tool" window.

Tool penetration values are entered as positive values to cut into the material. A negative value will not cut the material and will display a warning message before you start this is handy for test runs. A value of Zero "0" will recall the default value for this operation or tool. For changes of tool type, you may use the button "Xxxx Tool Setup" (where Xxxx is the type of operation). For example, you will be able to change your "V" insulation tool with a small stub end mill.

Some examples of tool selection:

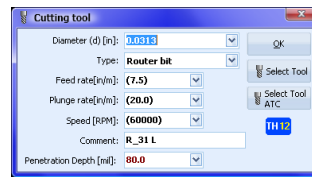
"Insulation Tools"

To change/select the insulation tool click on "Insulate Tools Setup":

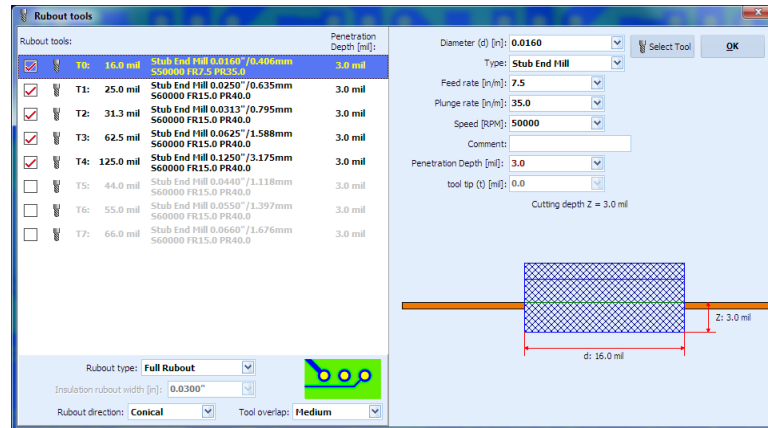


If you have (or already arrange) the necessary tool in the machine magazine simply click on "Select Tool ATC", it will show you only the tools that are currently in your machine magazine. Using similar technique you can make your selection for the tools for other operations.

"Cut Tool"



"Rubout Tools"



"Drill Tools"

Drill Tool Table							
	hole [in]	Diameter [in]	Type	Speed [rpm]	Plunge rate [in/min]	Feed rate [in/min]	Comment
[H09]	0.015	0.0135	Drill bit	(60000)	(40.0)	-	DRL_13.5
[H09]	0.030	0.0135	Drill bit	(60000)	(40.0)	-	DRL_13.5
[H12]	0.045	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
[H12]	0.125	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31

default router: T0 0.0313"/0.795mm Router bit 560000 FR7.5 PR20.0 R_31

Buttons: Apply default router tool, Select default router tool, Reset drill table, Select ATC, Select

Note:

The "Project Tools" table actually duplicates the function of buttons "Insulate", "Cut", "Drill tools", "Rubout tools". It is simply more convenient to have all needed information in a single window when you have to edit it.

IMPORTANT (ATC models only):

Button  will appear with red text if any tool(s) involved in your current project are not found in the machine magazine. The number in brackets shows how many tools are not in the machine magazine. These tool(s) are marked with a hand with pointing finger when you open the project tools list.

Project Tools				Depth [mil]	Insulate Tool Setup
Project tools: 9 in ATC tool holders, 0 in undefined location				5.0 mil	
[H12]	Insulate T0	"60" Tap (0.0630"/0.1675mm) (Insulate/Rubout/Stencil)			
[H12]	Cut T0	Router bit 0.0313"/0.795mm (Drill/Cut/Stencil)		(80.0) mil	
[H12]	Rubout T1	Stub End Mill 0.0313"/0.795mm (Rubout/Stencil)		(3.0) mil	
[H12]	Rubout T2	Stub End Mill 0.0625"/1.587mm (Rubout/Stencil)		(3.0) mil	
[H12]	Rubout T3	Stub End Mill 0.1250"/3.175mm (Rubout/Stencil)		(3.0) mil	
[H12]	Drill T1	Drill bit 0.0135"/0.343mm (Drill)		(80.0) mil	
[H12]	Drill T2	Drill bit 0.0135"/0.343mm (Drill)		(80.0) mil	
[H12]	Drill T3	Router bit 0.0313"/0.795mm (Drill/Cut/Stencil)		(80.0) mil	
[H12]	Drill T4	Router bit 0.0313"/0.795mm (Drill/Cut/Stencil)		(80.0) mil	

Auto tool depth update

During the execution of the operations the PhCNC LSSE software will offer manual tool change for the tools that are not in the machine magazine. After the machining with this tool ends the control software will ask for manual removal of the tool.

2.5 Object selection mode

Button  is used to enter object selection mode or left click and select "Select/Clone/Move objects" from the pop-up menu.

During the active Object selection mode, the cursor has the shape shown below.



Selected objects can be moved, cloned, rotated, grouped in a single object, ungrouped, and deleted. To select an object, click on it or click & hold and move the mouse to draw a rectangle that includes all the desired objects. Selected objects can be manipulated using the functions described below. To deselect selected objects, click away from them inside the main window, click on "Clear selection" from the pop-up menu, or hit the (Esc) key.

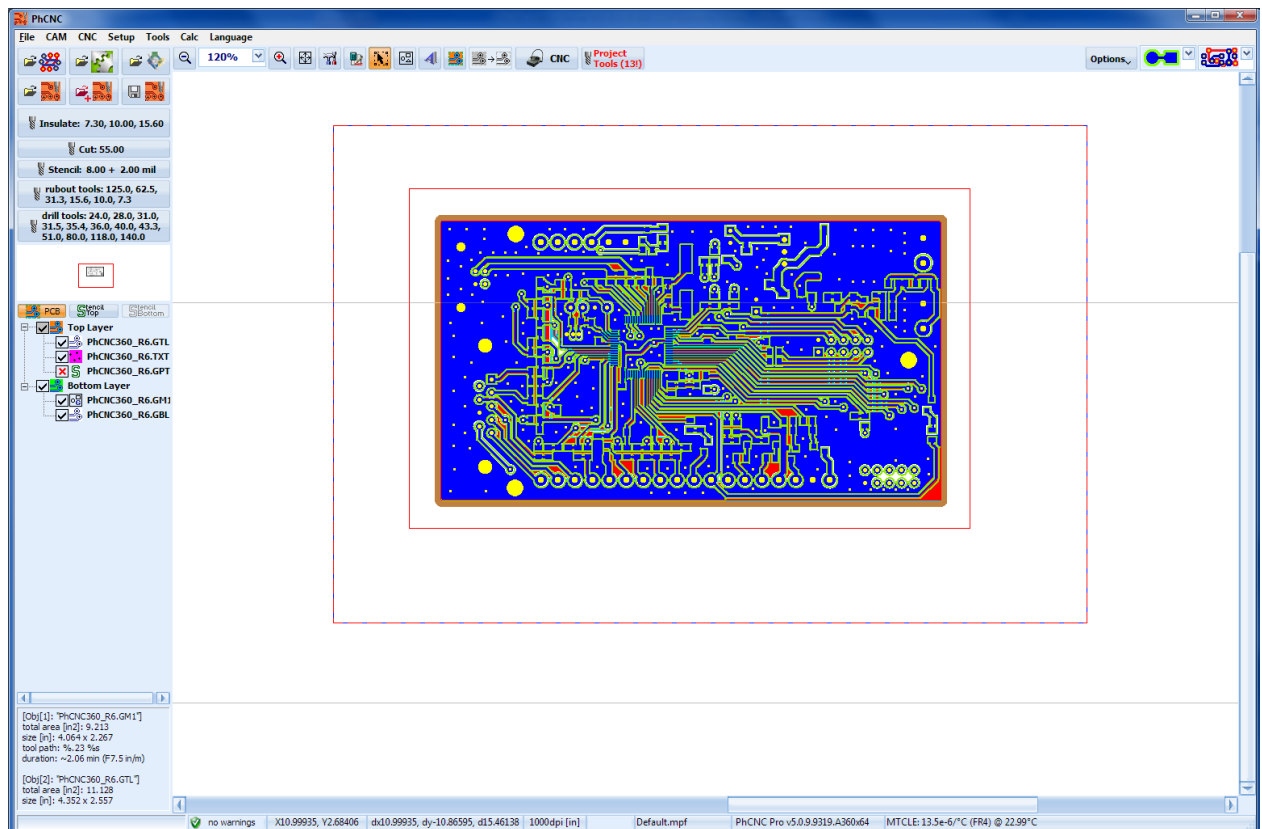
Important:

During the object selection some objects may be covered by other objects, thus the object on top will be selected instead of the one targeted for selection. In this case use "Send to bottom" (Ctrl+B) function from popup menu. Selected objects will move "under" the targeted object and it can now be selected.

Note:

"Send to bottom" function has nothing to do with "Send object to Bottom Layer".

"Send to bottom" only changes the drawing order on the screen to improve selection process of objects that overlap each other.



2.6 Move Objects

PhCNC LSSE in CAM mode has several ways to move objects:

- click and drag using mouse
- "Rotate & Move Selected Objects ..." function from popup menu
- "Move To ..." function from popup menu

In all cases the operator must select the object(s) first. To select an object(s), click on  (Object selection mode). Now you can make a left click within the desired object(s) to select it (them).

Left click outside an object will clear selection (unselect). For more information see 2.5 Object selection mode.

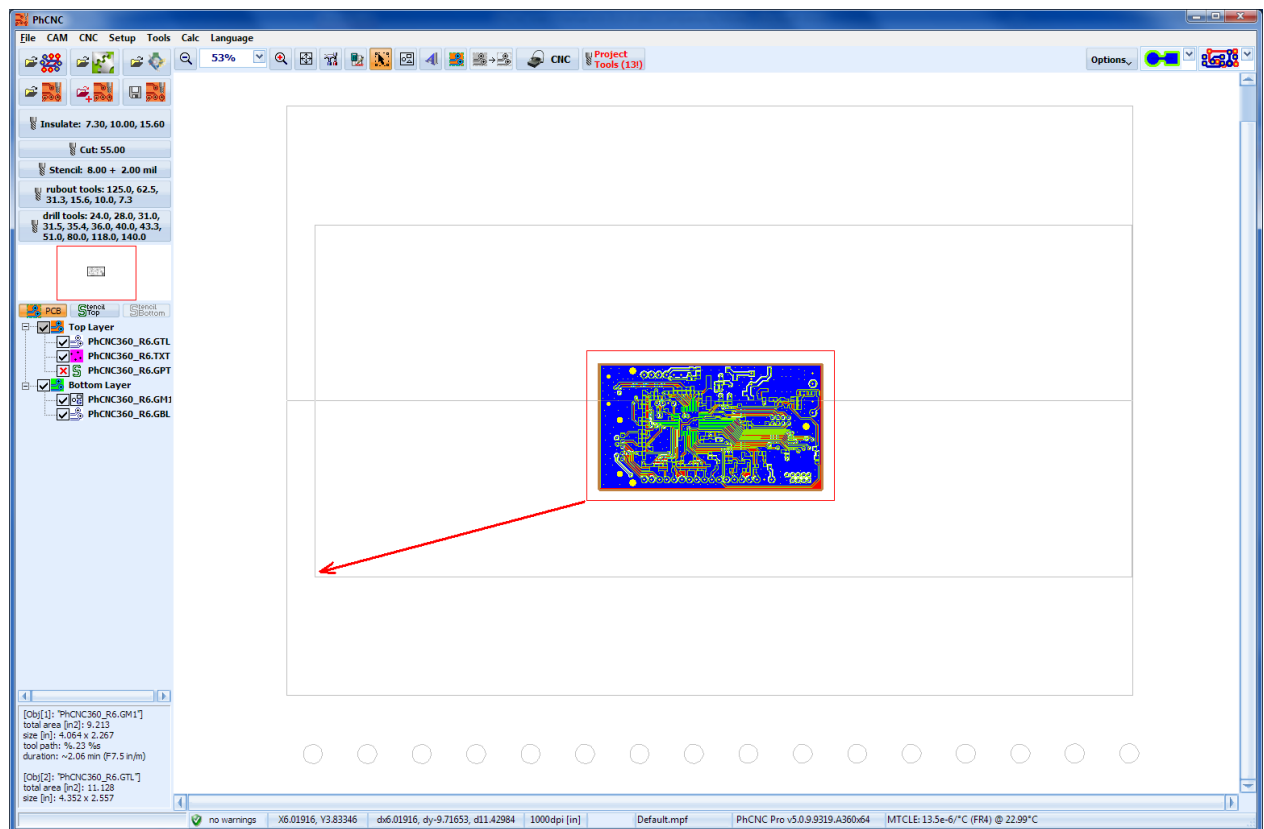
2.6.1 Moving objects using the mouse

In CAM and CNC mode of PhCNC LSSE, you can push and hold the left mouse button. Mouse movements now will scroll the screen (convenient option, especially when you have high zoom).

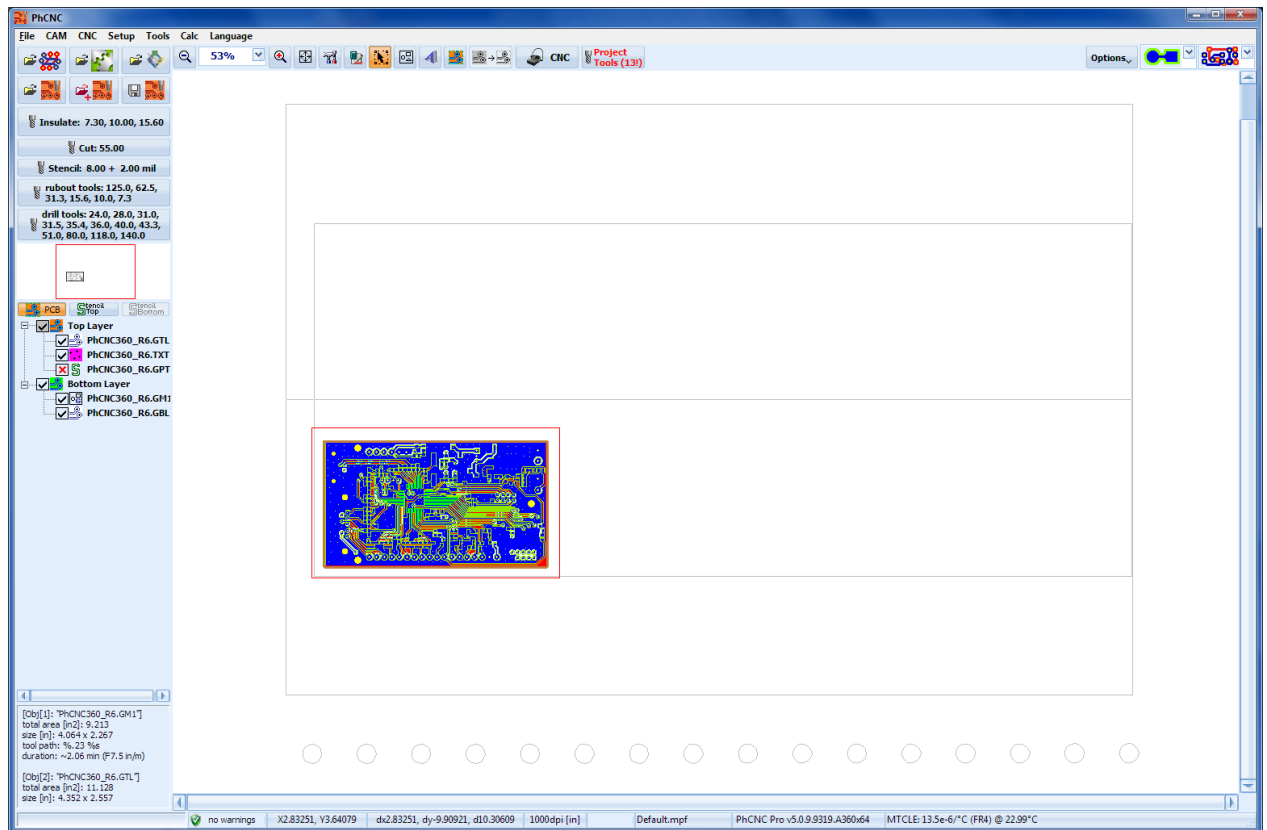
In CAM mode left clicking and holding the left mouse button on a selected object(s) will move the object(s) together with the mouse cursor (known as click and drag).

When more precision is needed you can increase the zoom and repeat the operation without the need to unselect the object(s).

On the screen below an object is selected (note the frame around it). Left click and hold inside the frame will move the object to a new position when you move the mouse and release the left mouse button.

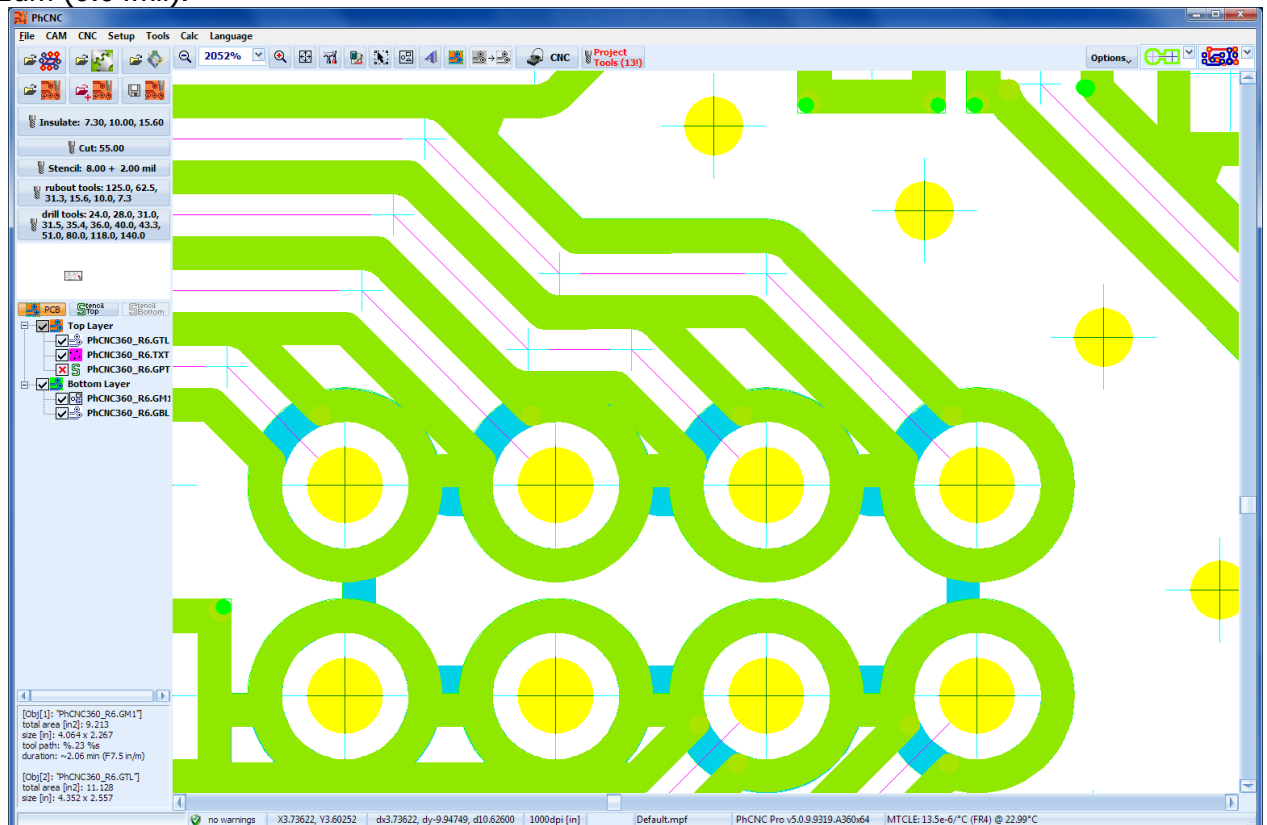


After moving the mouse to a new position and releasing the left mouse button:



During the object(s) movement, object(s) are continuously redrawn, this may create some screen flicker in computers with slower graphics or when the object is more complicated.

For extra precision positioning we recommend selecting the mode that shows Gerber vectors. In this mode the centers of every flash aperture and start/end points of each vector in your Gerber file are clearly marked. In addition, all traces and pads are "transparent". Using this option will allow you (in high zoom: 20000%) to match two layers at the level of resolution of 1um (0.04mil).

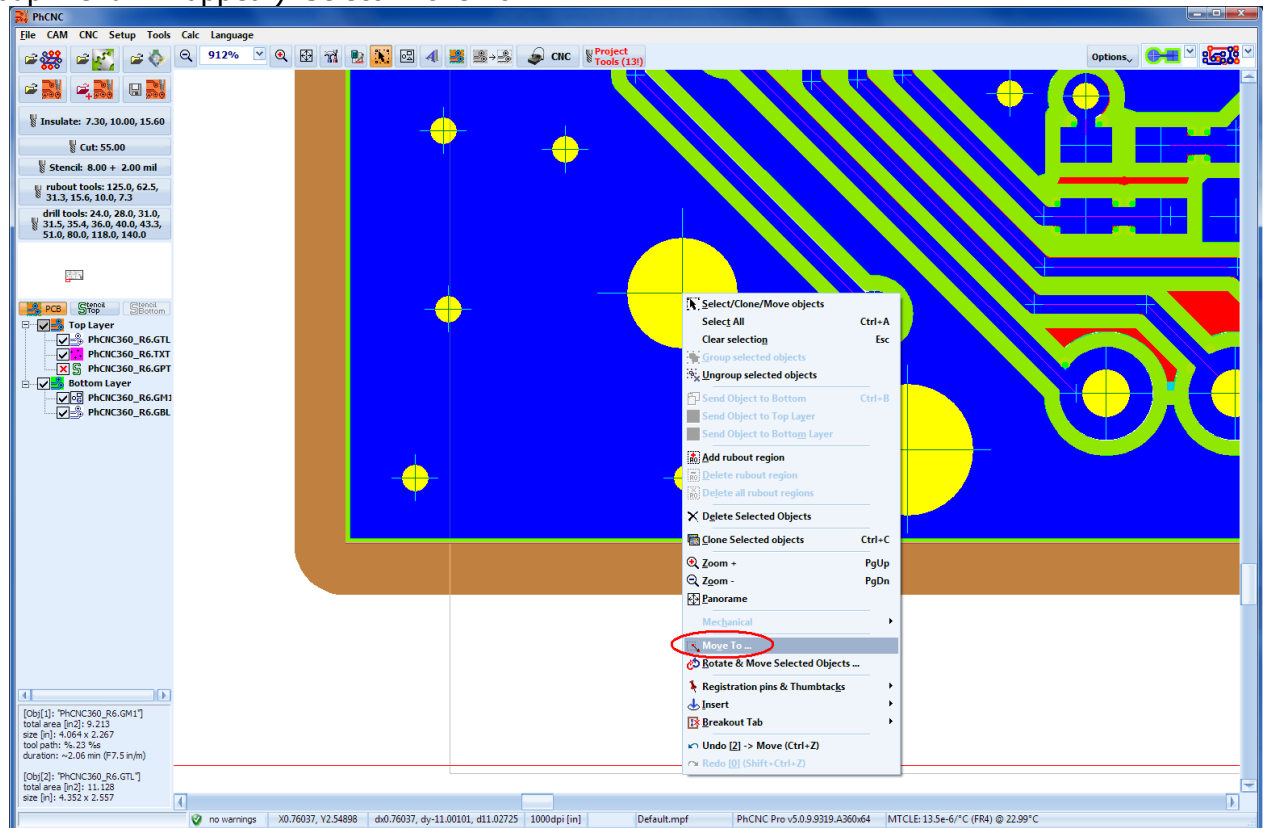


2.6.2 Move using "Rotate & Move Selected Objects ..."

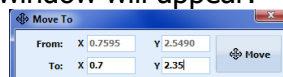
This is the best mode when you need to move an object(s) to a predetermined distance. Described in paragraph 2.8.

2.6.3 Move using "Move To ..."

In this mode you can move an object(s) from its current coordinates to new coordinates a specific distance in relation to a specific point on the object(s) utilizing full accuracy. First (the object must be selected) go to the desired position, in this case the exact center of a hole. Make a right click (the popup menu will appear). Select "Move To ...".



Selecting "Move To ..." the "Move To..." window will appear:



"From" contains (for reference only, not editable) the coordinates of your right click.

You should enter the new coordinates for the selected point in "To" and left click on the "Move" button. In this example the center of this hole will be at X = 0.7000, Y = 2.3500.

2.7 Clone (Copy) objects

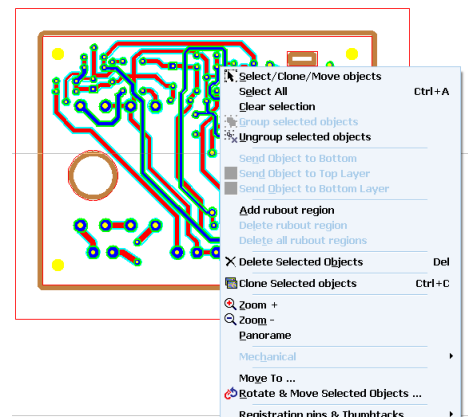
- Select an object or group of objects
- Select "Clone Selected objects" from popup menu

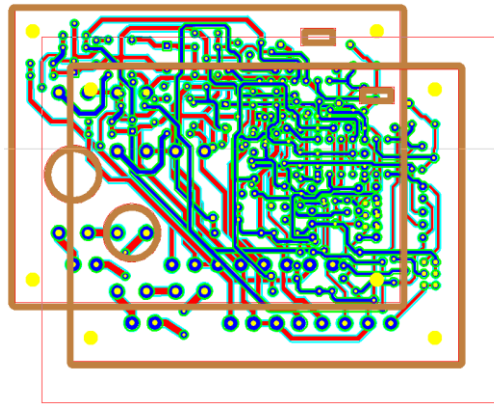
Cloned object(s) appear selected and shifted on the x and y axis at + or - 0.5 depending on the location of the original. The original is deselected automatically. Newly cloned object(s) can be moved to any position, rotated, or cloned again.

To move a cloned object, click, hold, and drag the object.

Note:

See also the Panelization (2.29 Panelization)!

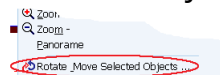




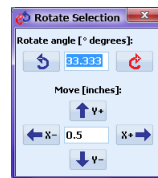
Cloned object

2.8 Rotate objects

Select an object and click on "Rotate & Move Selected objects" from popup menu:



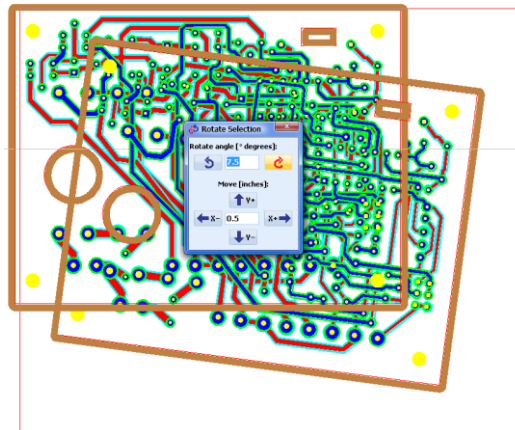
Rotate Selection Window will appear:



You may set any angle and distance.

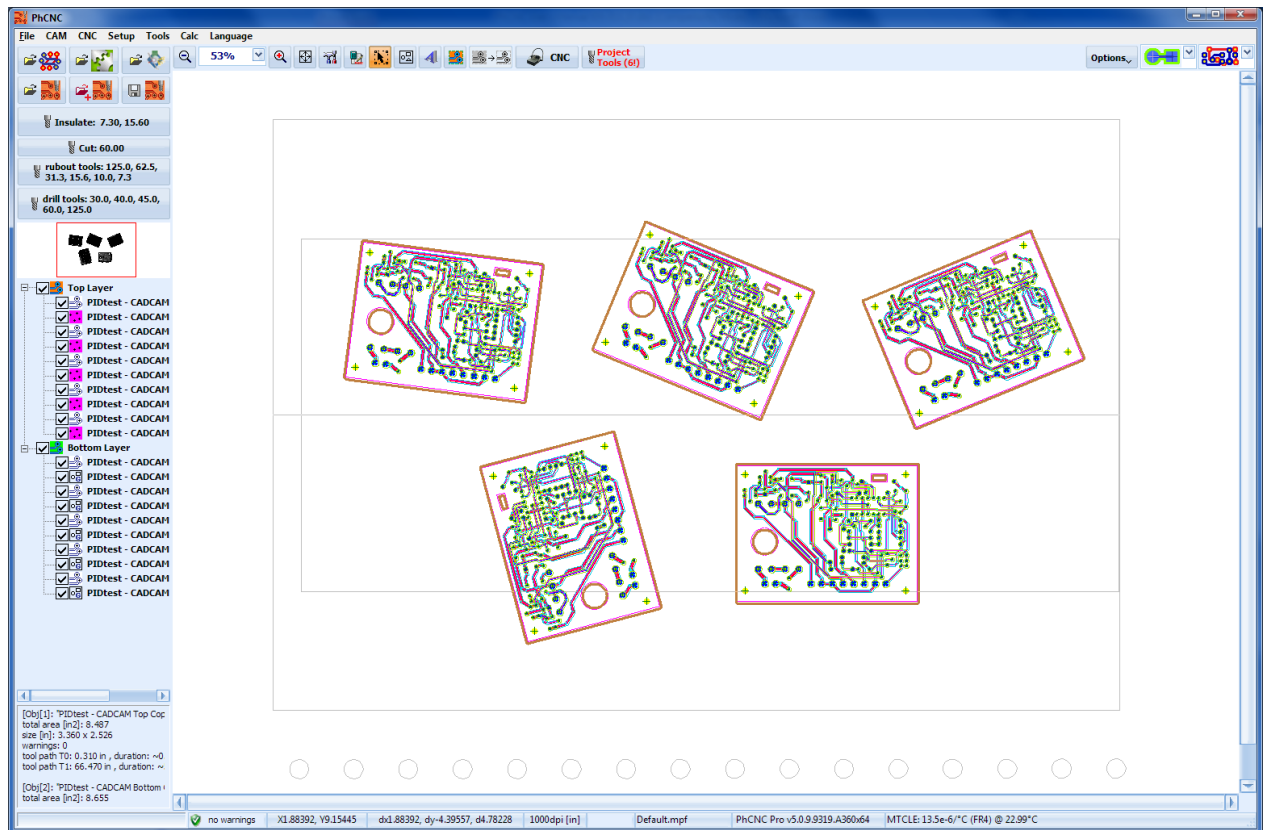
Using the direction buttons, you will be able to move and rotate the object.

For example, the object below has been rotated at 7.5° CW

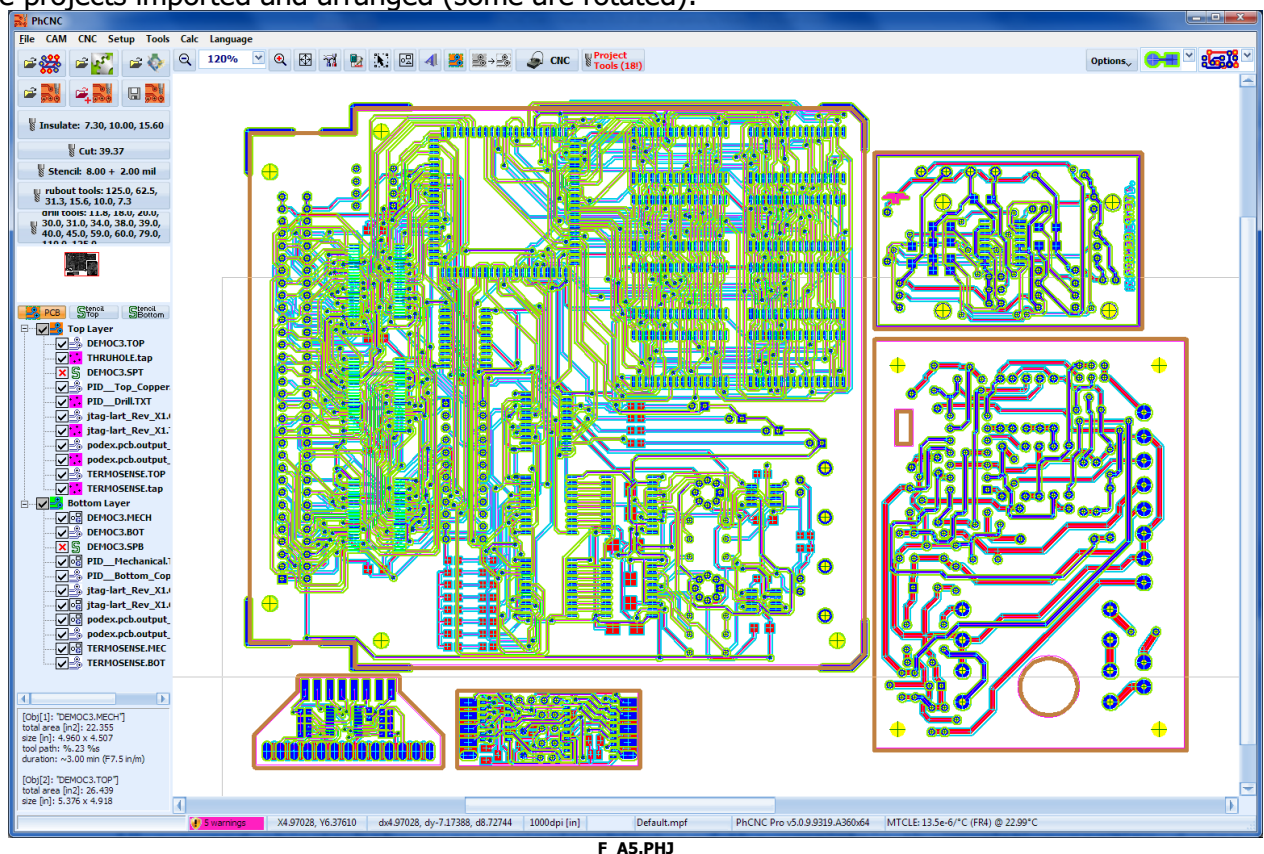


Example 2:

Five copies of a board rotated at different angles:

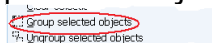


Example 3:
Five projects imported and arranged (some are rotated):

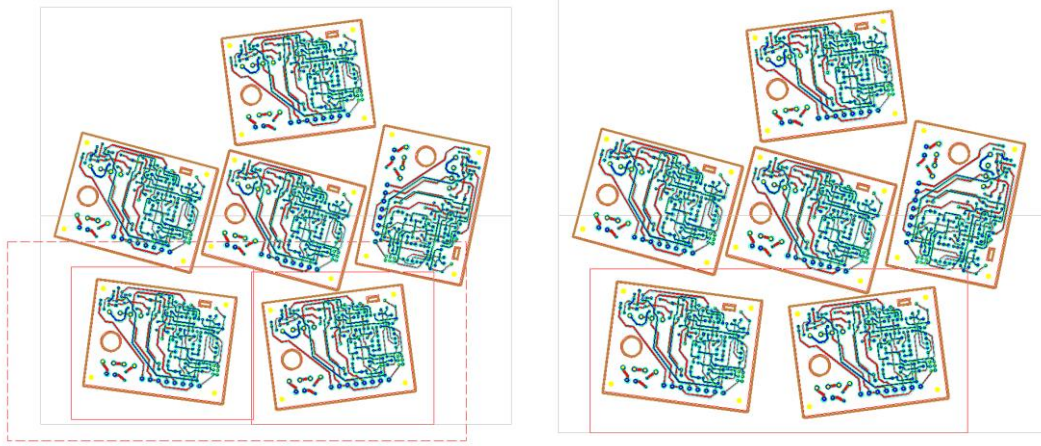


2.9 Group/ungroup objects

Select more than one object, click on "Group selected objects" from popup menu:

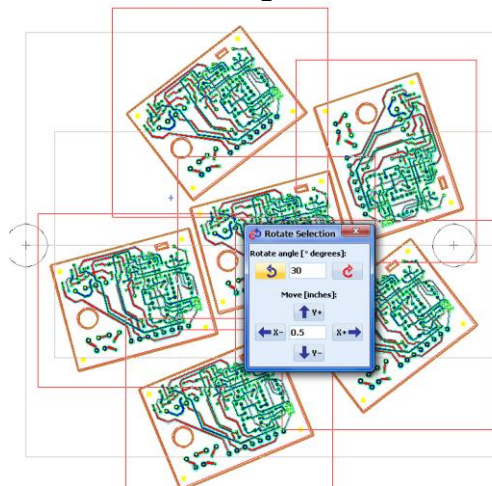


All grouped objects behave as a single object and they maintain their relative position during further manipulations.



To ungroup, select the desired objects and then click on “Ungroup selected objects” from the pop-up menu. It is not necessary to create group in order to rotate, move or delete. Just select all objects that should be affected.

For example, all below were rotated at a 30 DEG angle CCW.



2.10 Delete objects

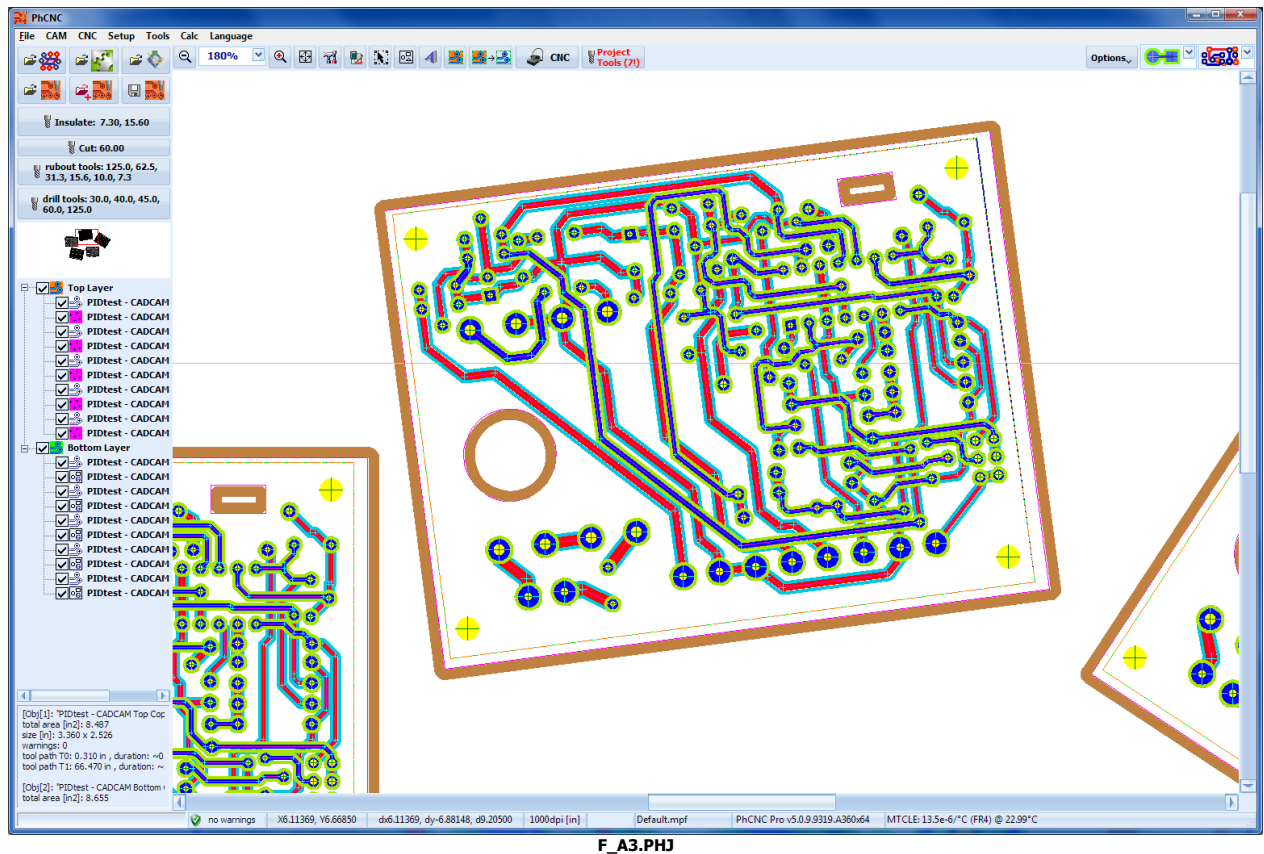
To delete objects, select them, and then click on “Delete Selected objects” from the popup menu.

2.11 Rubout – excessive copper removal

Button  is used to remove any unnecessary copper that is not defined in the gerber files from selected regions of the board. To create the regions that will be removed you have to click on the rubout button. Now, you are in the rubout definition mode. Use click & hold and move to create a rectangular region. If your object is rotated, the rectangle created will be rotated too. The number of regions is unlimited. If you have cloned objects, the region specified on one object will apply to all clones. During the active rubout region selection mode, the cursor has the shape shown below.

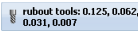


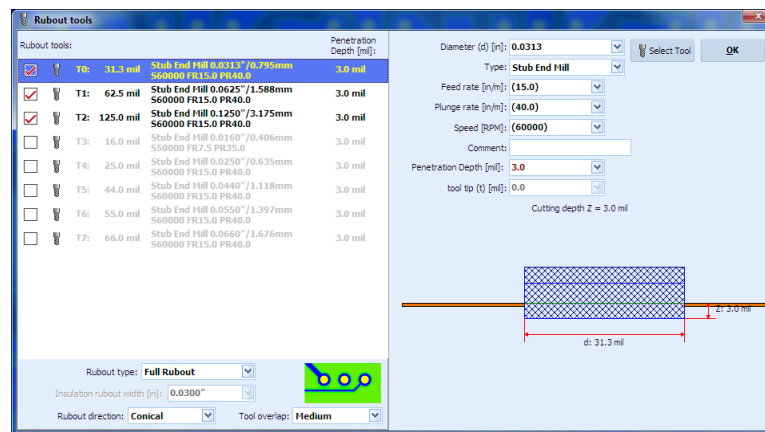
Example of rubout region creation (short dash line):



F_A3.PHJ

A created rubout region will apply to all visible (enabled) layers, in this example Top & Bottom. If you need to create different regions in top and bottom layers, hide (disable) one and create the region in the other, then swap them and create the regions in the other.

Button  allows you to set the right sizes for your rubout tools.

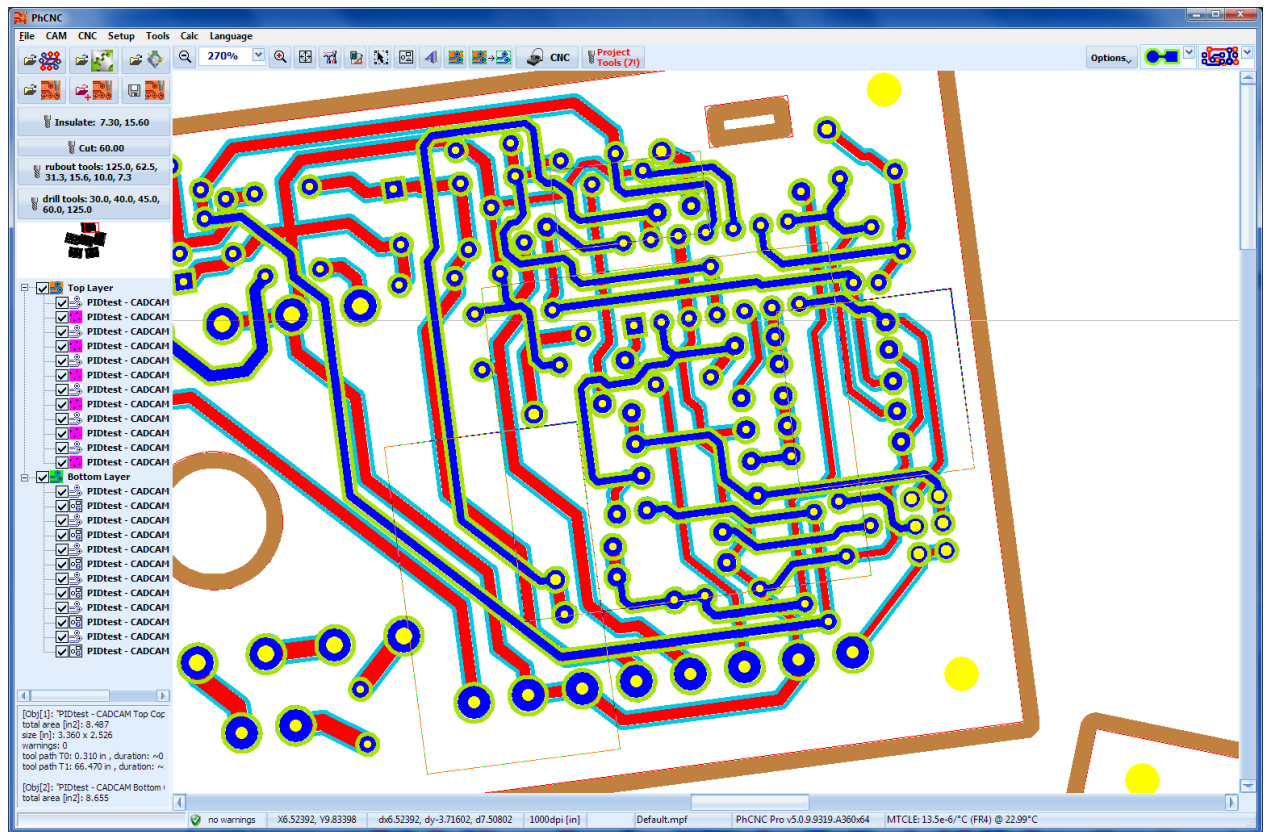


The first and the smallest tool in the rubout process will always be the insulation tool. The rest depends on your design and you can create any combination you want. Enter 0.0 for the positions that will not be used (or press "Delete"). Enter the right sizes for the tools you will use. Rubout will sort them by size and will start with the smallest. If you change a tool in this table recalculation of rubout region(s) is needed.

Important:

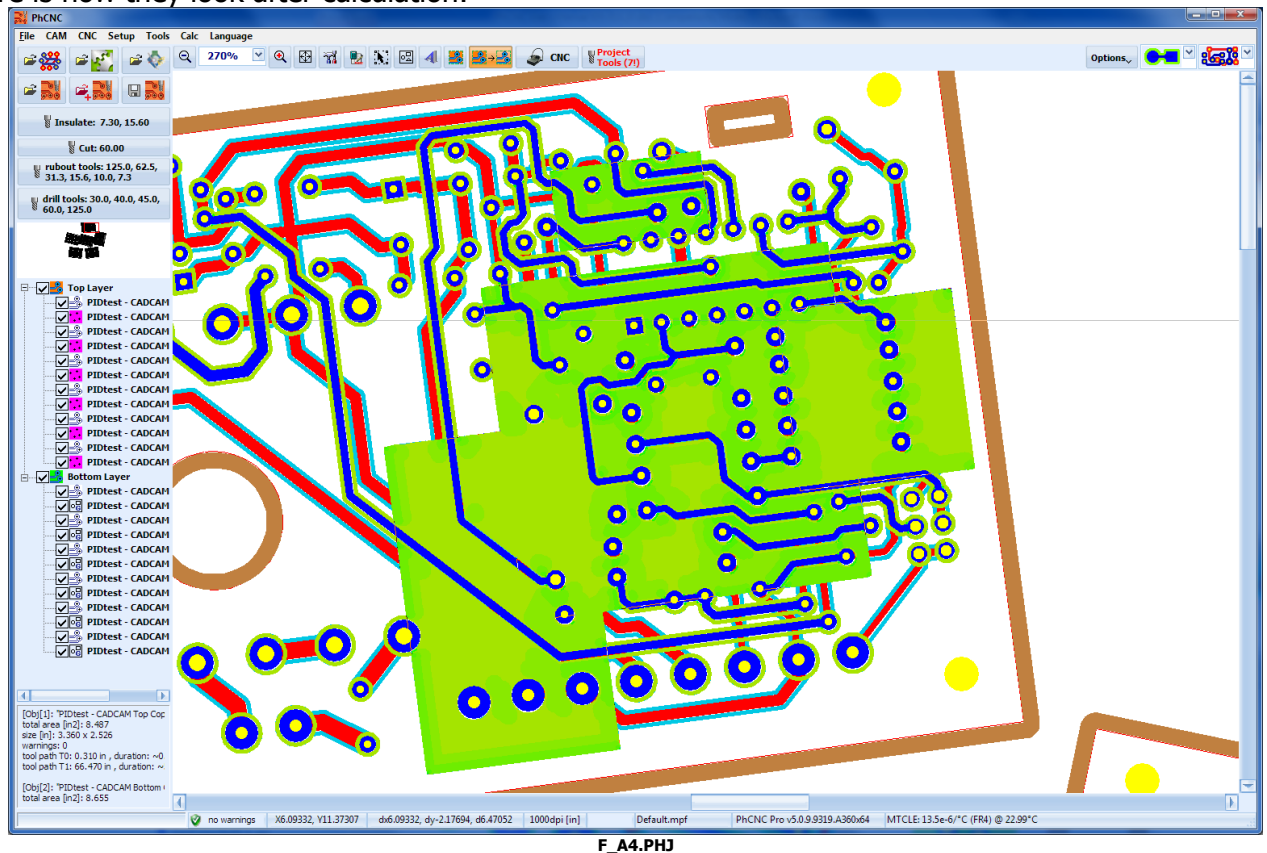
Tool sizes entered in the rubout table must be accurate! The overlap between the tools is approx. 0.001 inch.

In the example below 4 regions are created:

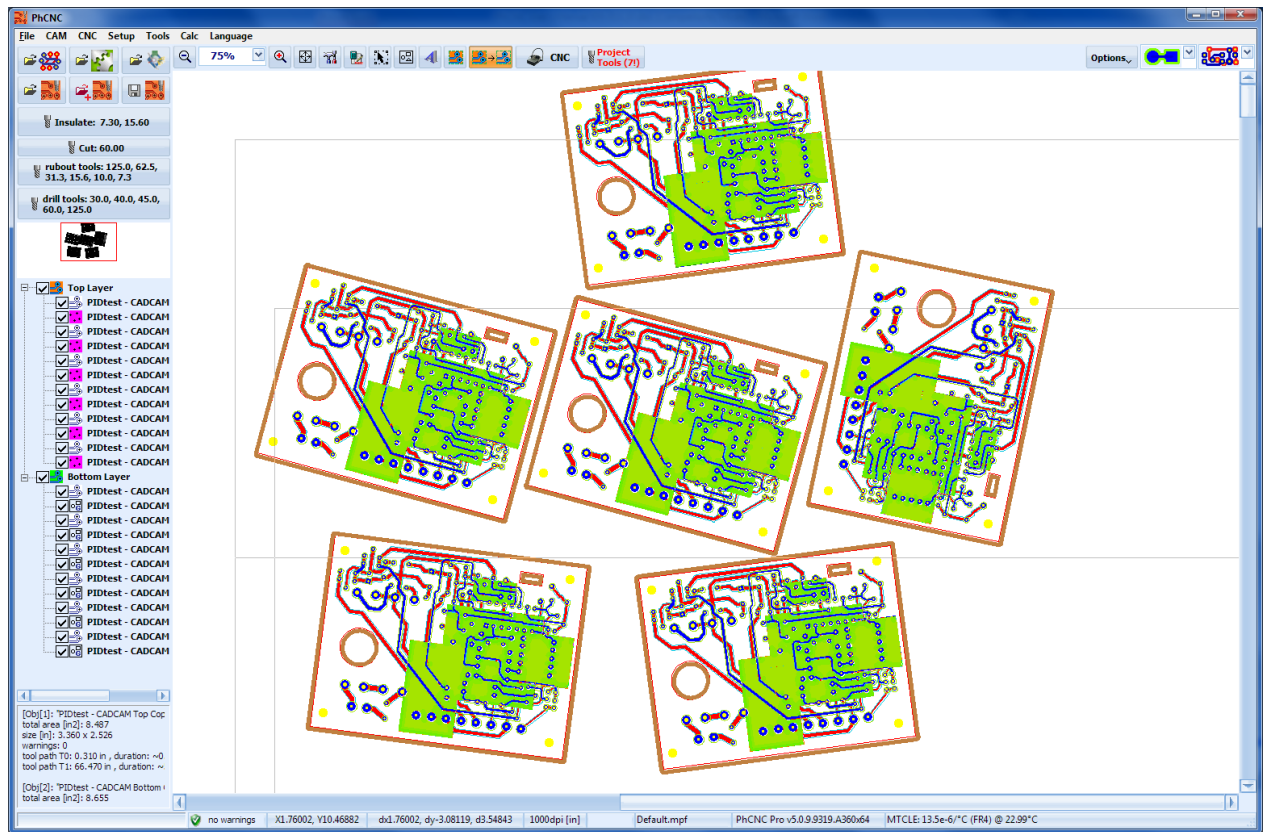


To calculate regions, click on the  button. First click will calculate the "Rubout regions" and all consequence clicks will toggle between hide/show them on the screen (v1.17.5+)

Here is how they look after calculation:



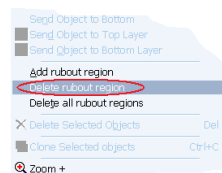
Here is the example above spread to all copies:



F_A3.PHI

If for any reason you do not calculate the regions before entering CNC mode, PhCNC LSSE will calculate them automatically!

If you would like to delete some of the rubout regions, place the mouse pointer on the region and click on "Delete rubout region" from the popup menu. When you delete a region from a cloned object, all copies will be affected.



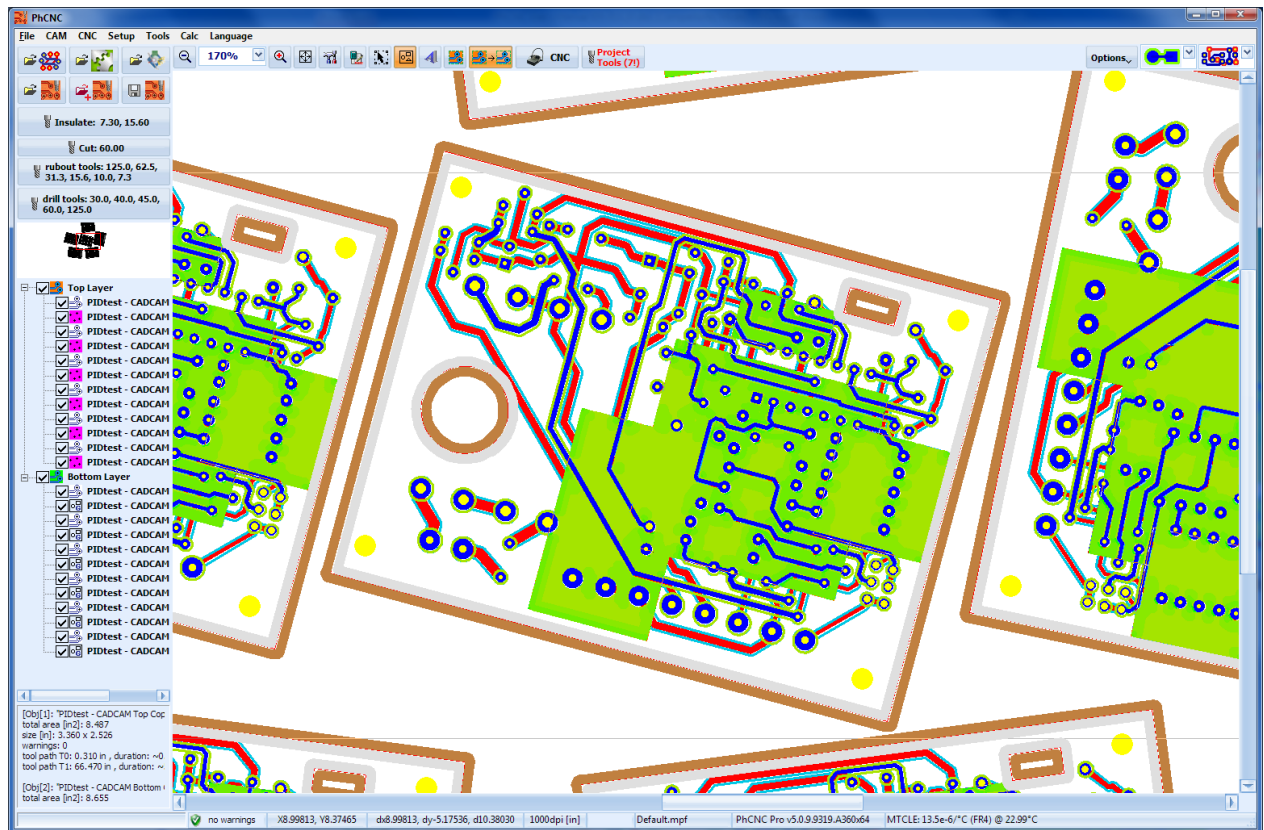
If you would like to delete all rubout regions click on "Delete All rubout regions" from the popup menu.

2.12 Editor for the mechanical layer

2.12.1 Contour cutting selections

If you have loaded a mechanical layer for contour cutting, PhCNC LSSE automatically selects outside tool diameter compensation for the most outside contour and inside compensation for the contours inside (if any). If for some reason you need to change the automatic selection or disable some contours you have to click on the  button.

The example below shows active (contour cutting selection)  mode:



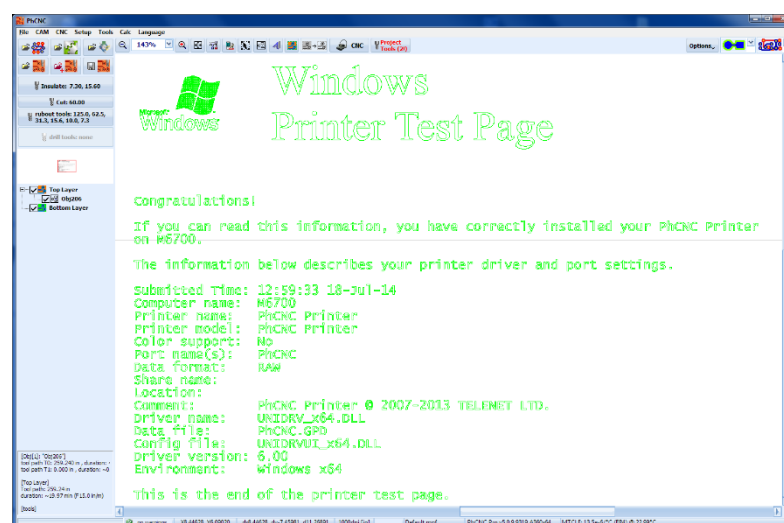
Selected contours are in light brown and the possible, but not selected, opposite compensation contours are in light gray. You may change the selection with a mouse click on the contour. When you leave contour cutting selection mode, unselected contours will become invisible and will not be executed.

2.12.2 Contours selection in the mechanical layer

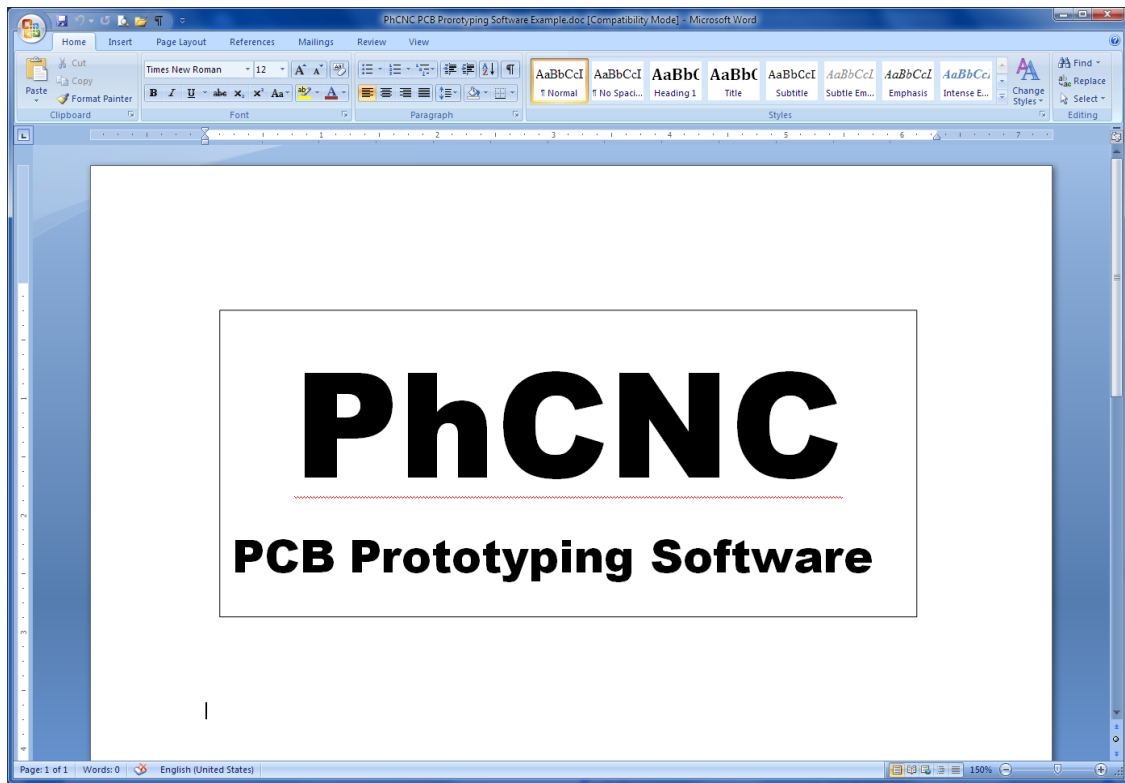
In this mode, you can perform the following operations with the contours generated by "PhCNC LSSE Printer" and  (import of text and graphics).

2.12.3 Contours generation with "PhCNC LSSE Printer"

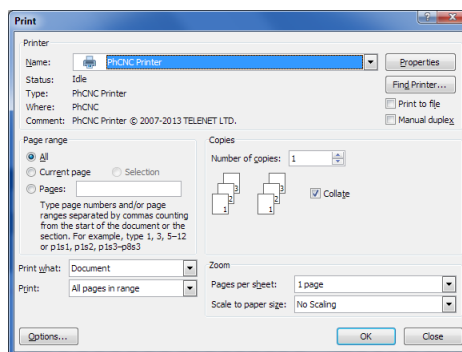
Upon starting PhCNC LSSE, the "PhCNC LSSE Printer" driver is installed, which allows you to import vector information in PhCNC LSSE from any application or CAD/CAM system. On the figure is shown the test page of the printer.



Example:
Name plate generated with MS Word:

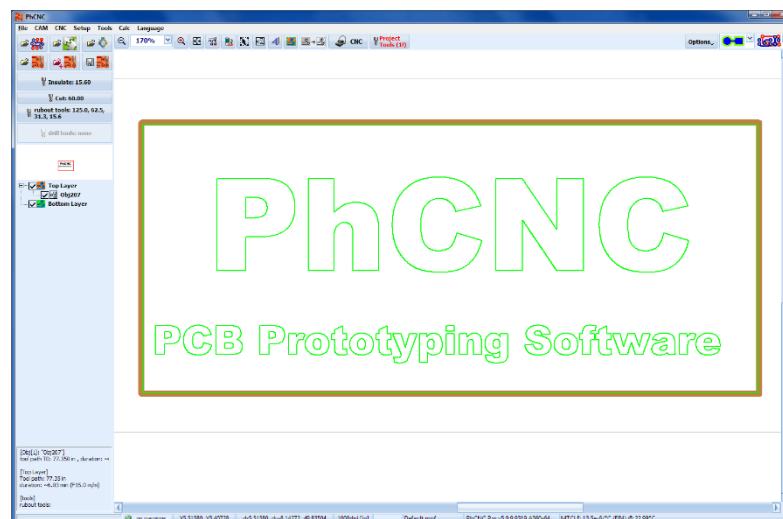


To import a name plate into PhCNC LSSE – simply select the printer “PhCNC LSSE Printer” and press “Print”.



Remark:

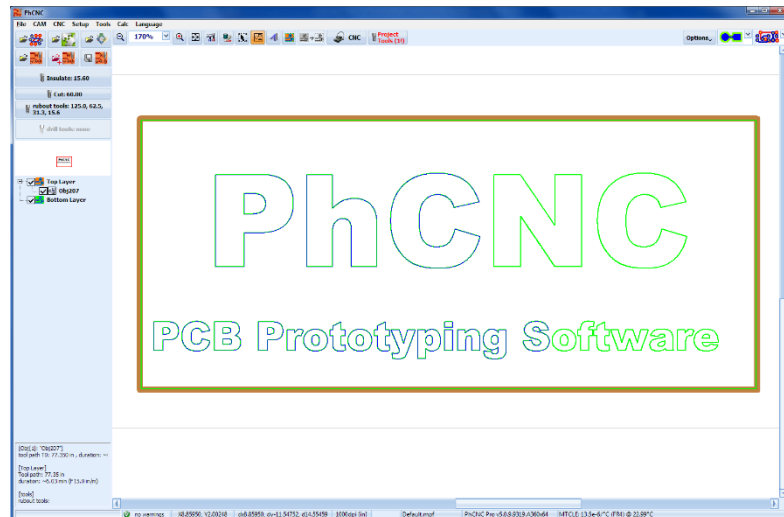
PhCNC LSSE must be started (you can also start it after the printing)!
The vector image is imported automatically in PhCNC LSSE (as a separate object):



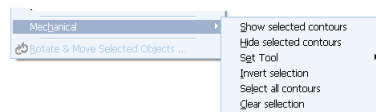
By default, all contours will be processed with the tool for insulation (T0). If you want to select contours for cutout (for example the external contour), enter the mode for editing of mechanical contours by left clicking the  button.

There are two possibilities for selection of contours:

- By clicking with the mouse over a single contour. In this way you can select/deselect the desired contour
- By defining a rectangular region with the mouse. All contours fully located in the region are selected.



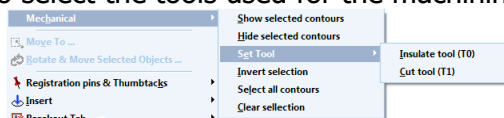
Right clicking on selected contours allows you to enter the "Mechanical" menu.



This menu will allow you to select the following options:

- Show the selected contours.
- Hide the selected contours. The hidden contours are not processed and are viewed only in the mode for editing of the mechanical layer
- Set tool menu tools that will be used to process the job
- Invert selection
- Select all contours used to select all contours
- Clear selection used to unselect the selected contours. This can also be accomplished by left clicking with the mouse outside of the contours, all selected contours are deselected.

The set tool menu allows you to select the tools used for the machining process.



Here, T0 is the current tool for insulation and T1 is the current instrument for cut-out.

On the figure below, the frame and the inscription PhCNC LSSE will be processed with T1 (cut-out) and the remaining text will be processed with the tool for insulation T0:

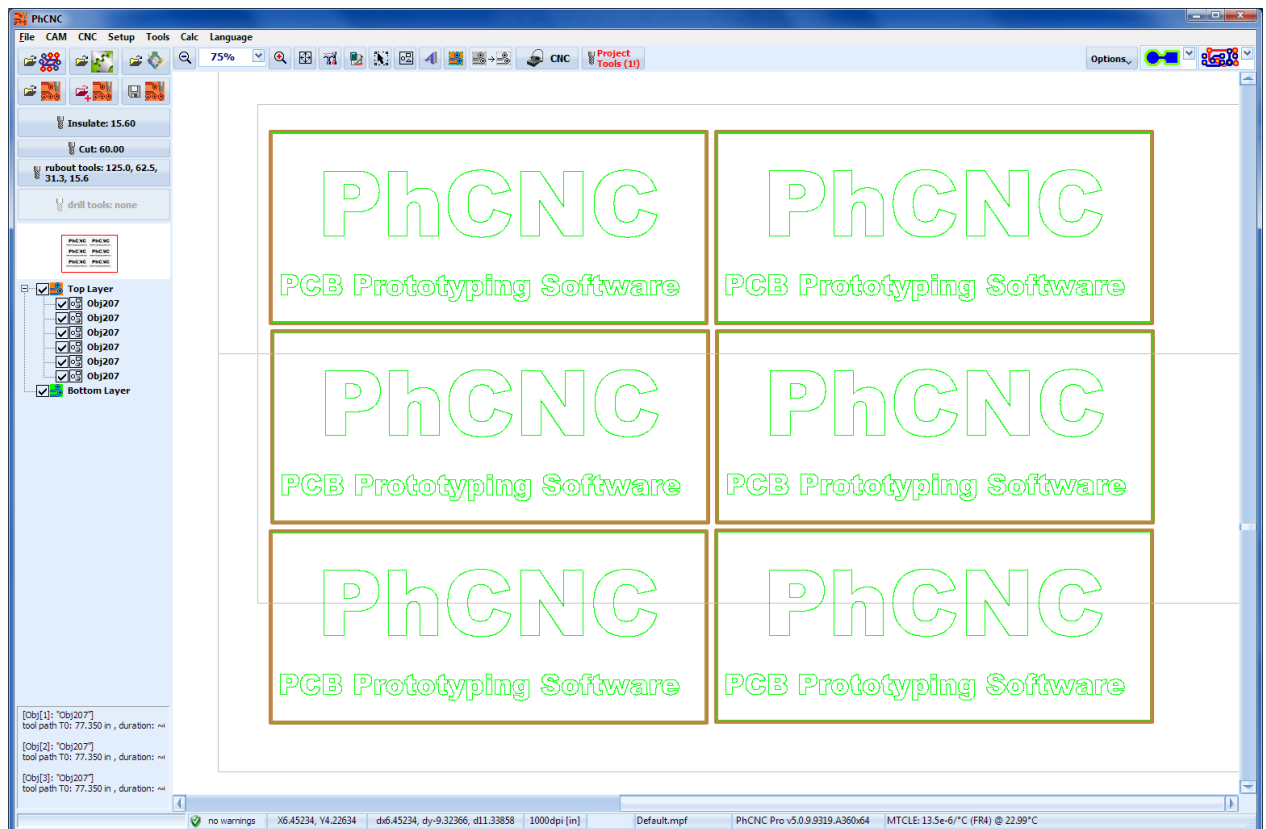


You can also use rubout. On the figure below the inscription which was selected only for insulation is now selected for rubout.

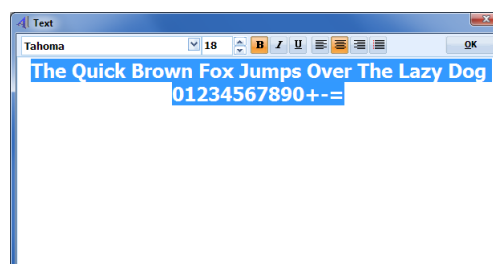


To produce the plate above, we can enter CNC mode and PhCNC LSSE will generate the following programs: Top Layer; Rubout (Top Layer); Cut (Top Layer);

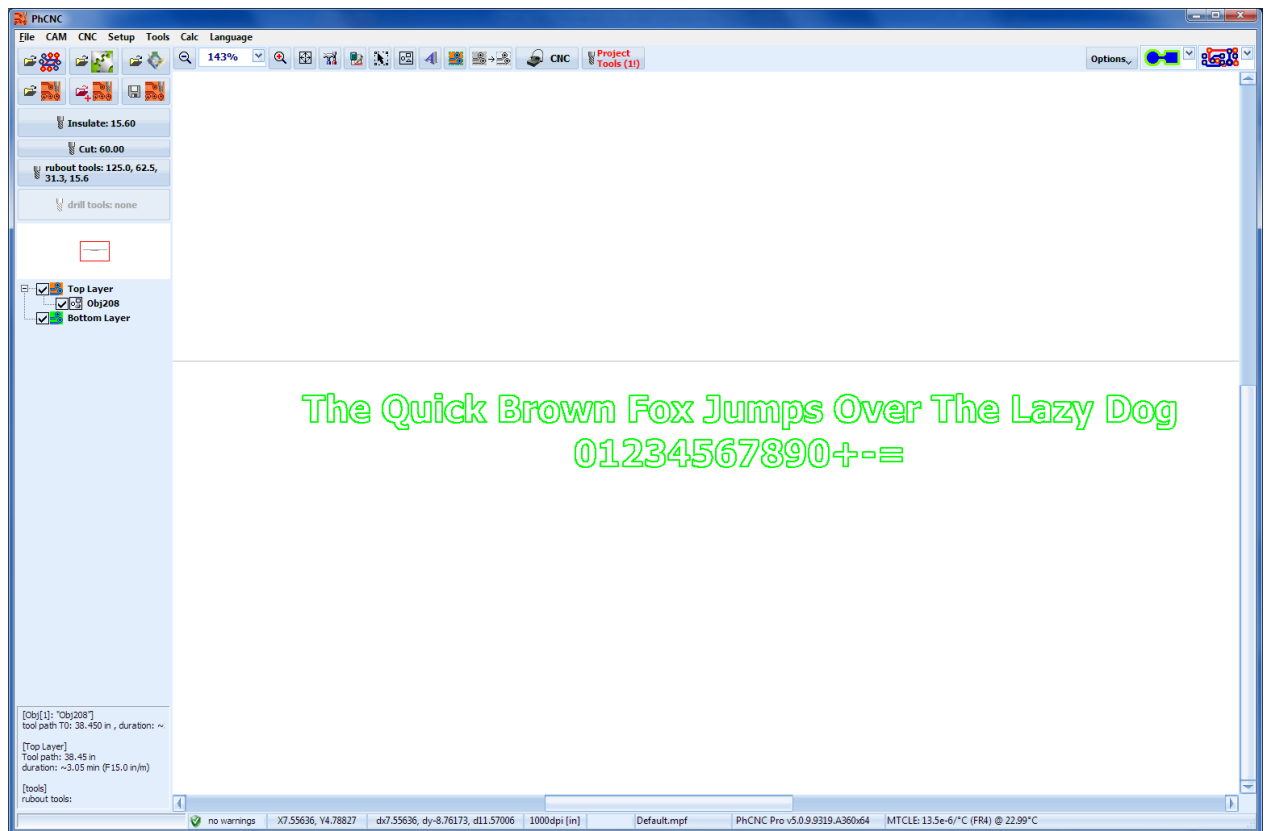
As with Gerber objects, you can rotate and clone the objects, generated with "PhCNC LSSE Printer":



2.12.4 Generation of contours with ("Insert Text")

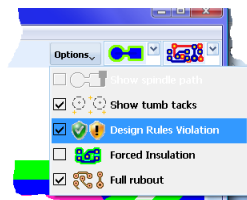


The result is shown on the figure below:

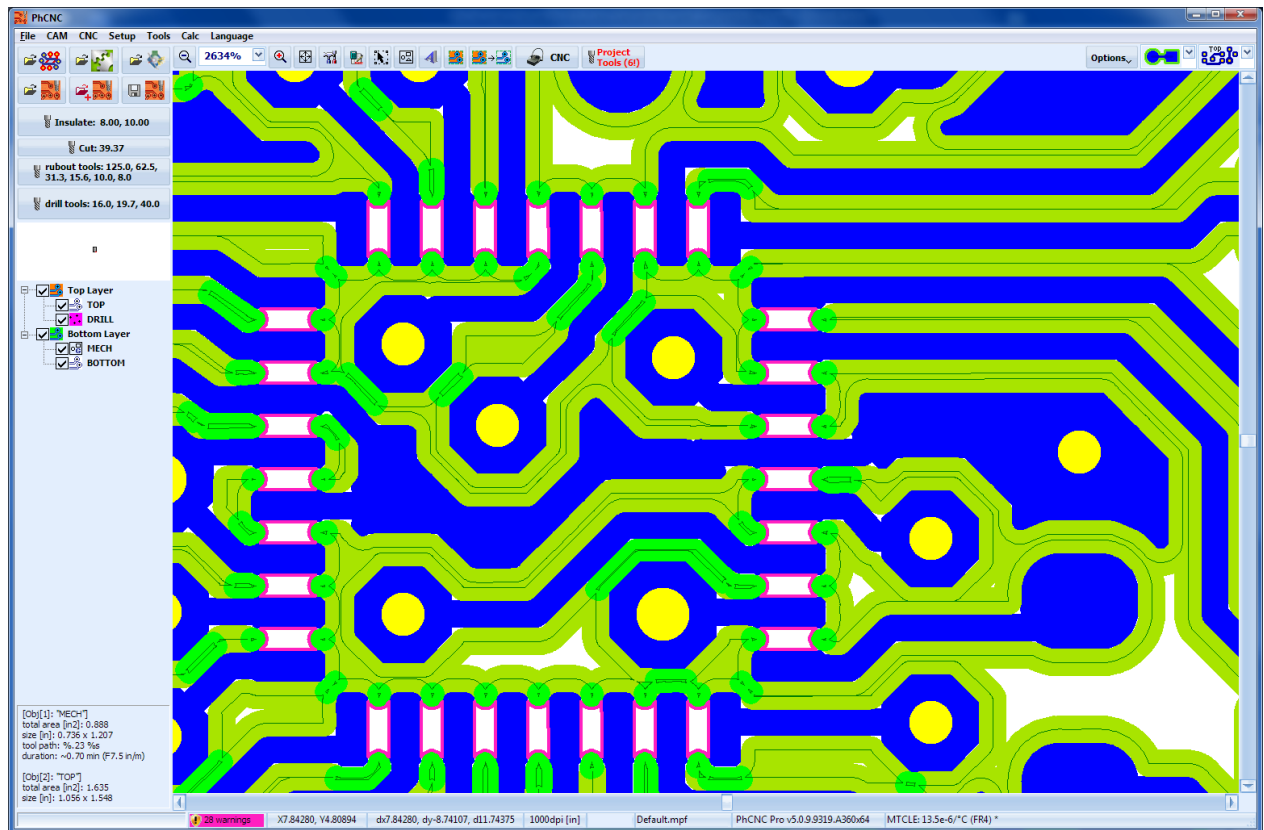


2.13 Design Rules Violation (DRV) detector

PhCNC LSSE utilizes DRV calculations. The main function of DRV is to check all tool paths for compatibility with the selected insulation tool (diameter). If the selected tool is bigger than the distance between the traces/pads, DRV detector will prompt you.



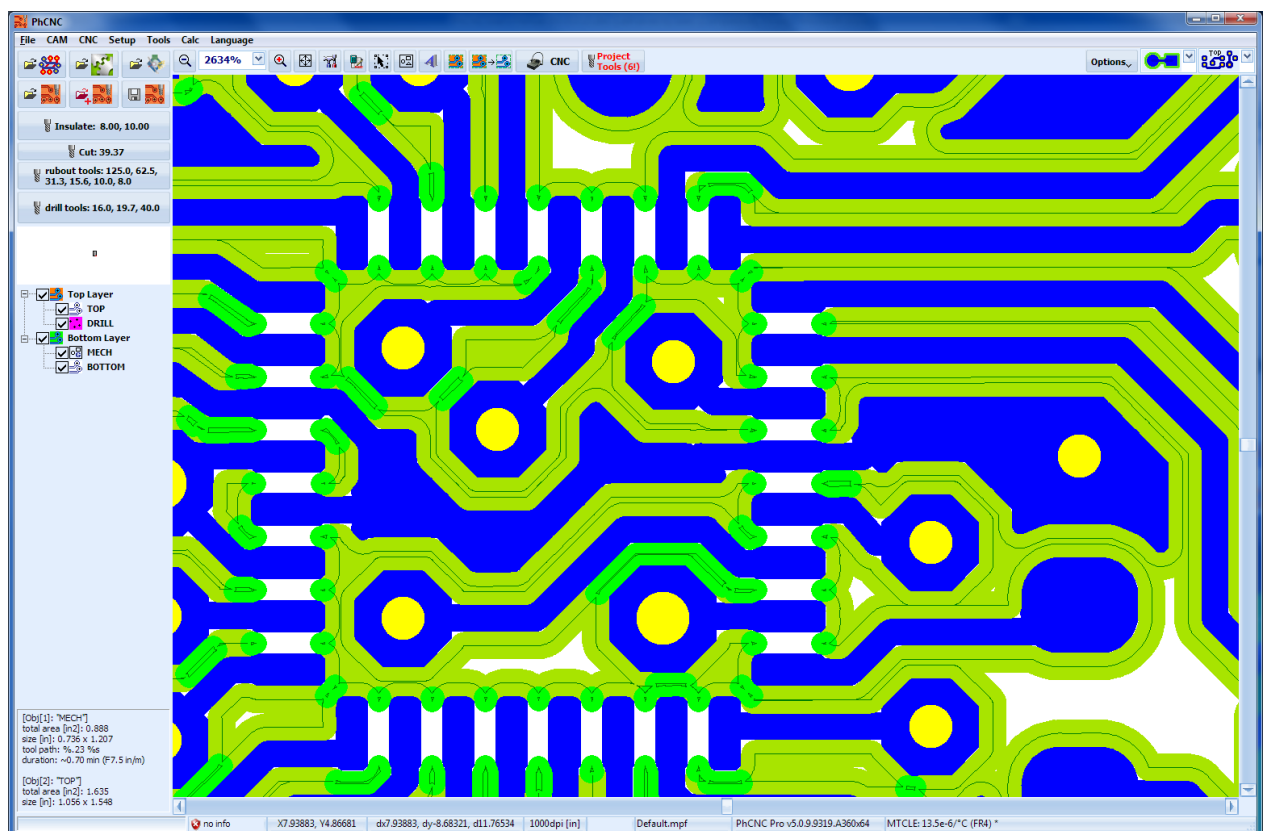
When this option  is checked DRV detector is active. It checks for DRV compatibility for each generated tool path. The DRV spots are presented with easy to see default pink:



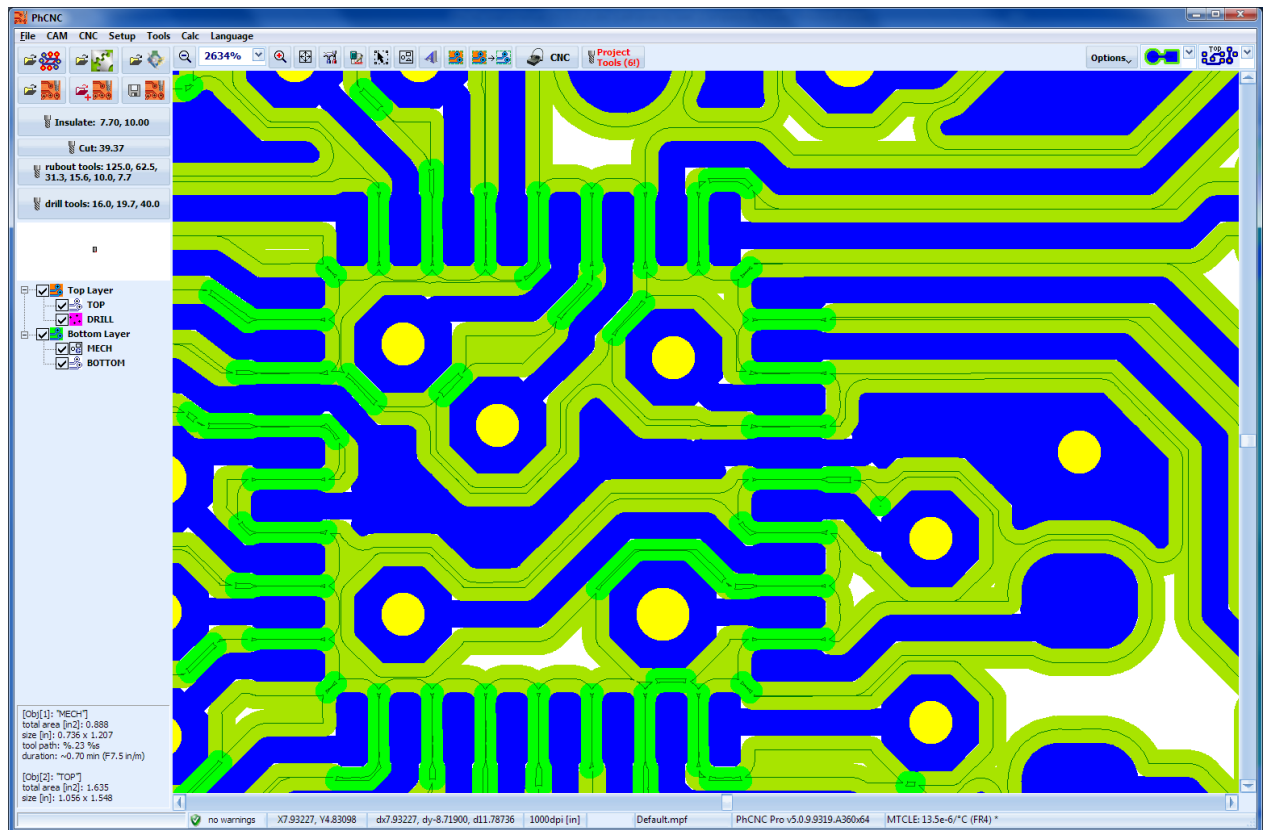
The status bar displays information () about DRVs and their number (for all layers on).

Top Layer -> TOP -> 28 warnings
total warnings: 28

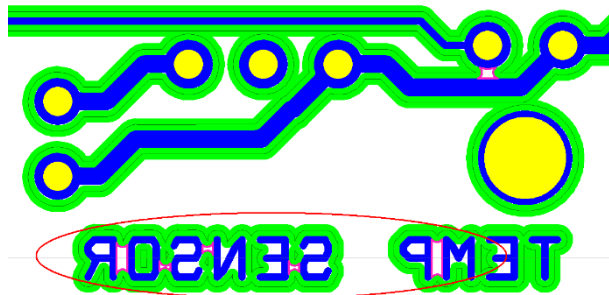
Button also shows/hides DRV spots. If it is not activated during the tool paths generation, you may turn it on later and DRV detector will check for DRVs.



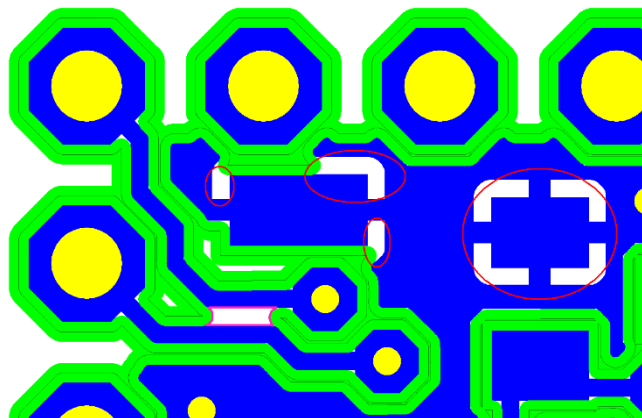
If there are no DRVs the status bar will show the no warnings indicator. If DRV detector was not activated during the tool paths generation the status bar will show the no info indicator. If DRVs exist the only practical solution is to reduce the diameter of insulation tool (See 2.23 Forced Insulation). In the example below we changed the insulation 8.0 mil tool to a 7.7 mil tool:



In many cases the DRVs are located in the "copper text" (as shown below) and can be ignored. It is advisable for the operator to check all listed DRVs.



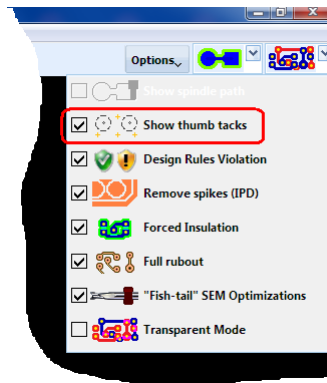
DRV detector will prompt you for shorts (non-insulated area that belongs to two different contours/pads/traces). If the area belongs to the same circuit (has a common contour) it will not appear as a DRV. See the case below (in red circles). The traces belong to the same circuit and are not DRVs.



2.14 Registration Pins and Thumbtacks

PhCNC LSSE allows you to add and display registration pins and thumbtacks visible in CAM & CNC modes of the software.

The menu "Options", contains menu option  which shows/hides the active registration pins and thumbtacks:

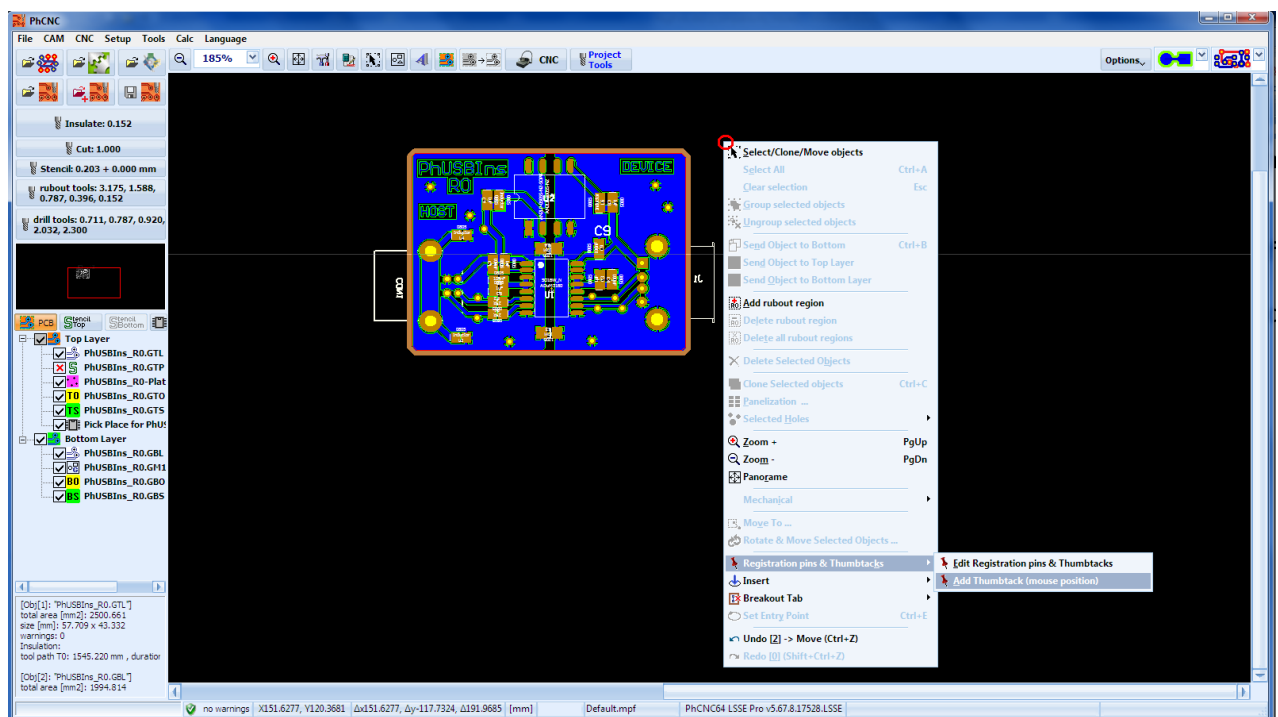


2.14.1 Thumbtacks

Thumbtacks are markers visible in CAM and CNC mode. They are a very convenient tool during the setup. You can set their position using one of the following methods:

- Current mouse pointer position (CAM & CNC mode), "Add thumbtack (mouse position)" from popup menu.
- Current spindle position (CNC mode only) "Add thumbtack (spindle position)" from popup menu (Simulator).

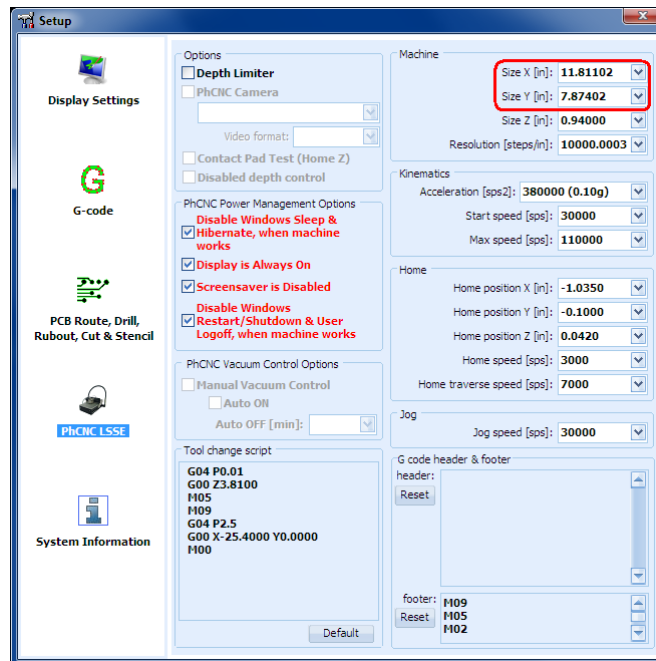
Below is an example of adding a thumbtack to the position of the mouse in CAM mode:



2.14.2 Registration Pins

Registration pins show the position of the (normally two) pins that create the mirror axis for double sided jobs.

By default, the mirror line is located at position Y with coordinate 1/2 of the Y size of the working area:

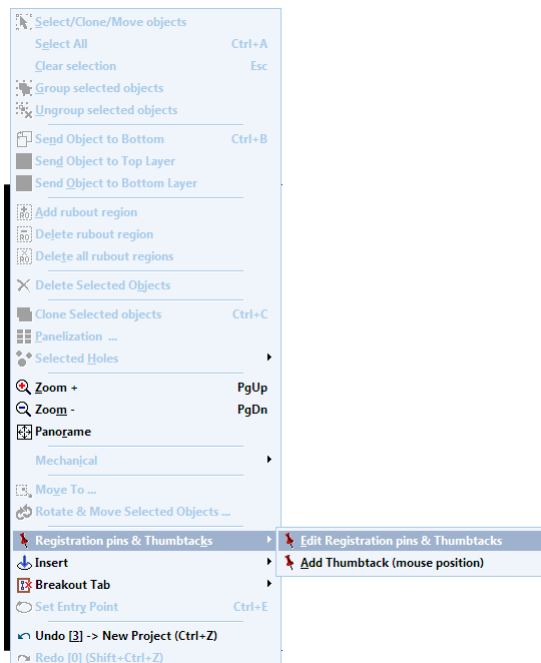


In the example above, the Y size is 7.87402" and the mirror line is located at Y=3.93701"

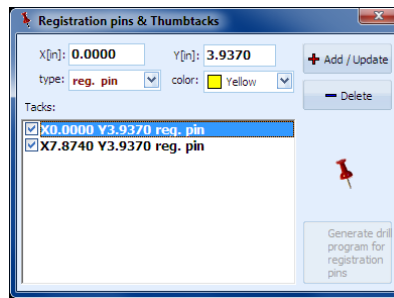
During the setup process you should not place any objects closer (than 0.5 inch) to the registration pins. Using this option, you will be able to see the real relative position between any pin and your design.

Around each pin you will see a circle (dashed line) with 1.0 inch diameter that represents the area that should be clear of project objects. The registration pins must be located on the center line in the middle of Y coordinates and strictly parallel to X axis. This is the line that creates a mirror image and tool paths for the machining of the bottom layer (The Y coordinate of the mirror line can be adjusted with special command).

Adding/editing of registration pins is in the popup menu:

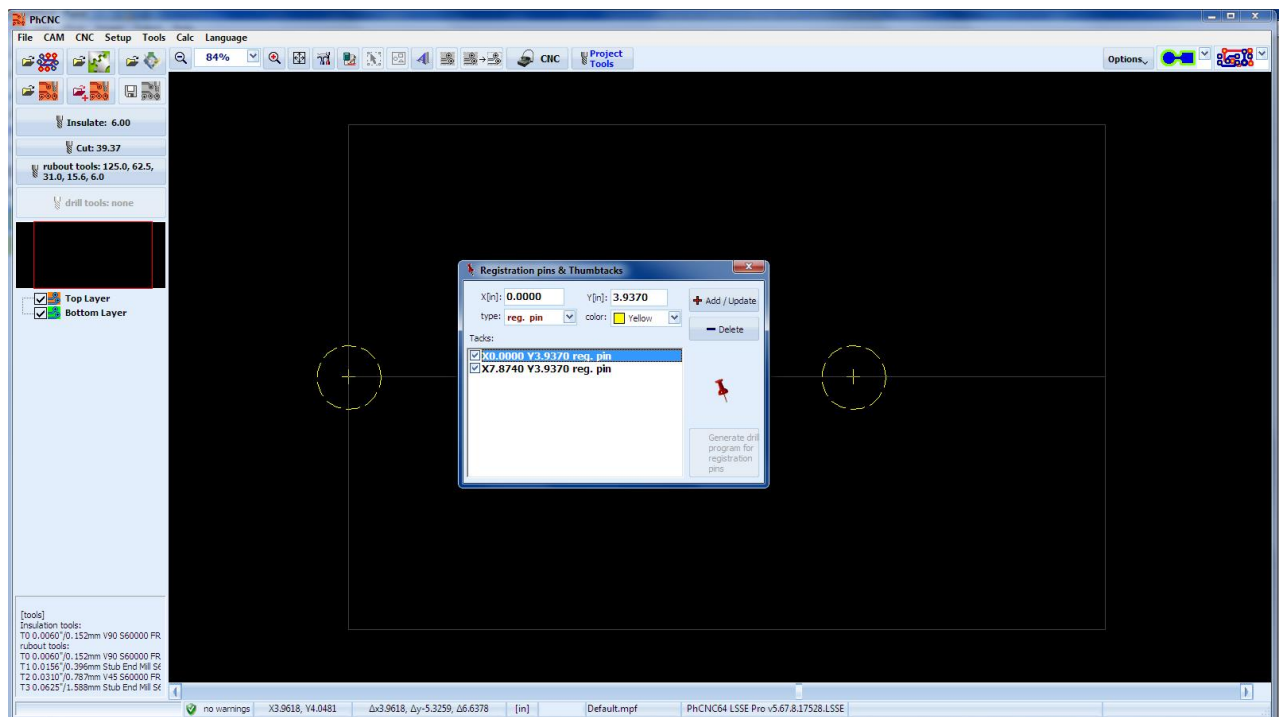


Below is the window that allows you to add/edit registration pins and thumbtacks:

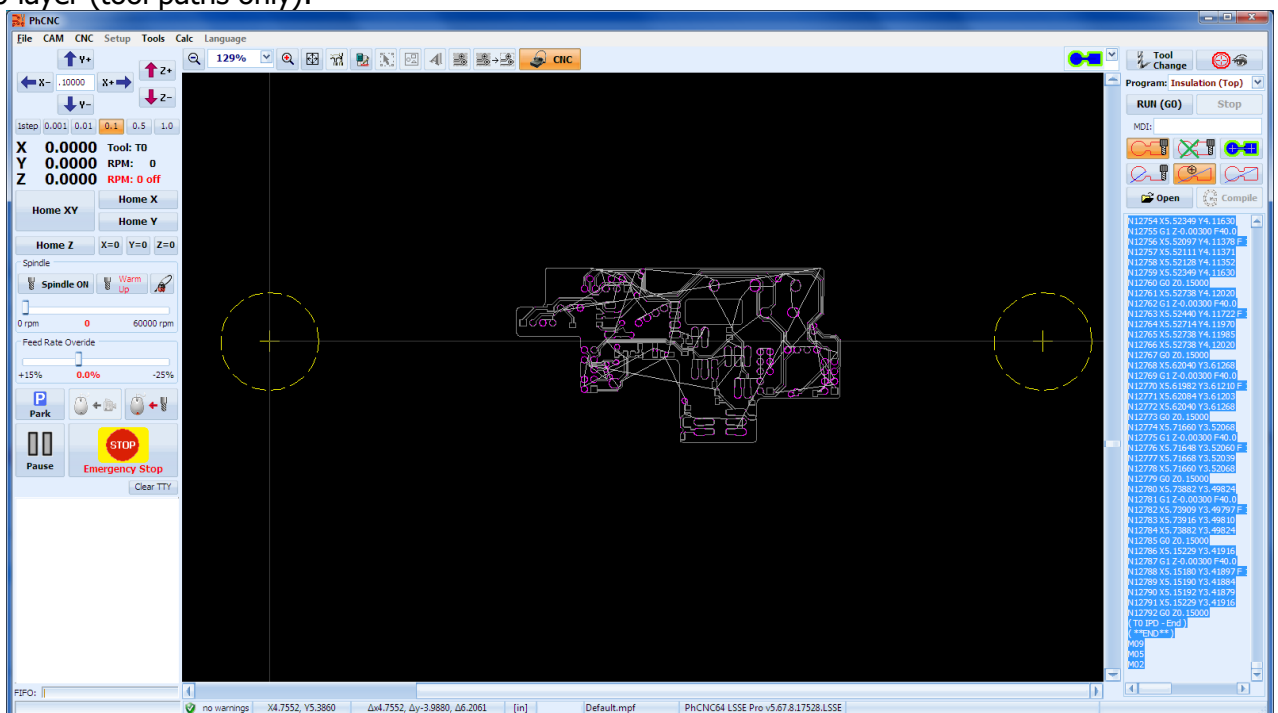


It is a good practice to disable (remove the check mark) instead of deleting a pin or thumbtack. Registration pins and thumbtacks' settings are "global" and not saved in the project files (.PHJ).

In the above example we have 2 registration pins at (0.0000,3.9370) and at (7.8740,3.9370), shown below:



Examples:
top layer (tool paths only):



bottom layer (tool paths only):

2.15 Insulating copper rubout

This mode is designed to save time and tools during the prototyping process. It creates wider strips of insulation around the traces and pads (where possible) in order to have a better condition to populate the board with components leaving most of the unused copper behind. It depends on your particular design, but basically, we do not recommend it for RF designs (10 MHz and up), and high-speed digital circuits etc. Remaining copper (unconnected) may serve as a vehicle (mostly capacitive) for interferences between the circuits and make your prototype different from the board in regular production. It will give you an advantage, saving time and tools, in your audio, power supplies and other low frequency designs.

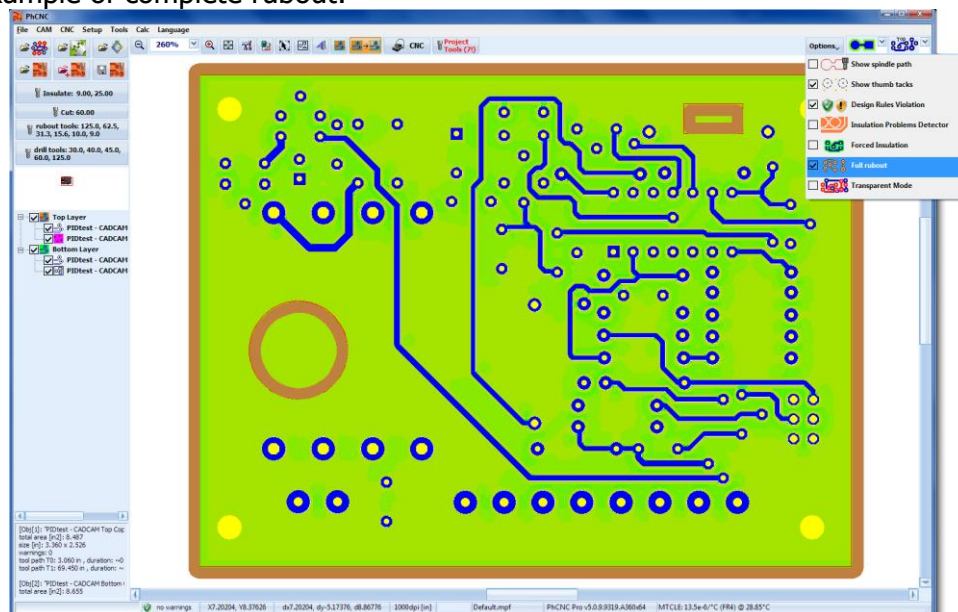
Rubout has two modes:

- Complete rubout removes all unused copper around the traces and pads in selected regions or the whole board
- Insulating rubout removes extra copper around the traces and pads.

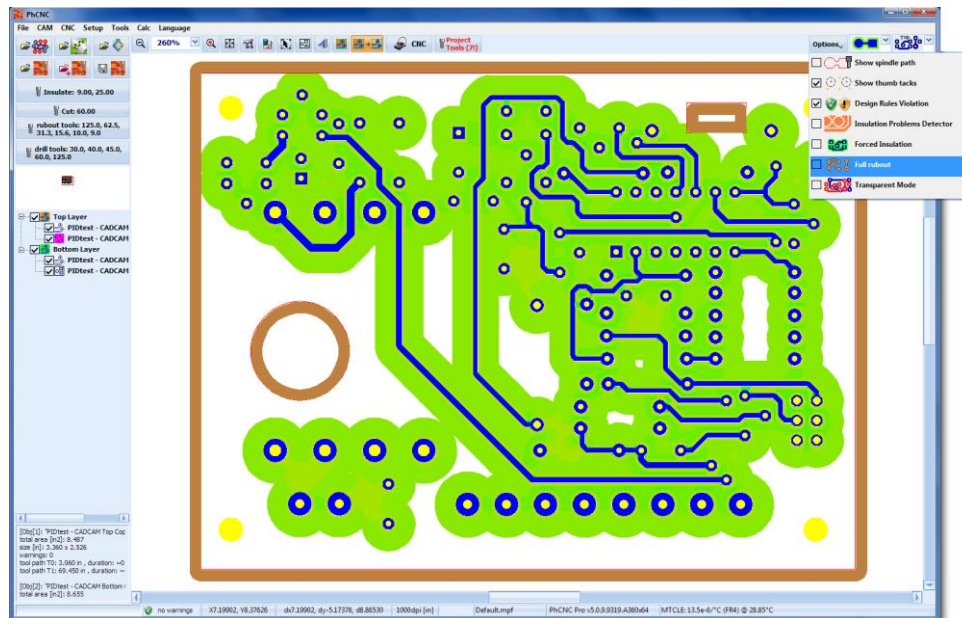
Complete or insulating rubout can be selected by this toggling button 

-  PhCNC LSSE calculates tool paths for complete rubout
-  PhCNC LSSE calculates tool paths for insulating rubout

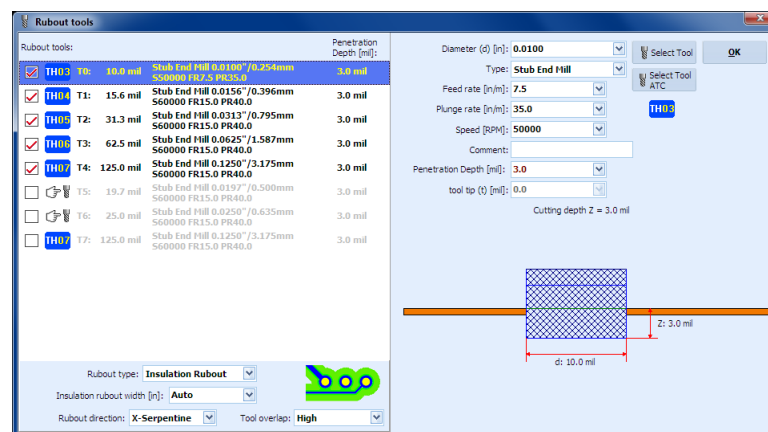
Below is an example of complete rubout:



Below is an example of Insulating rubout:



Insulating rubout will create a minimum insulation (D) of all selected regions. This does not apply for the spaces between the traces/pads that are less than the rubout minimum. The tool table for (full) copper rubout is used for Insulation rubout too. Now (rev 1.15.9 and up) you can select desired insulation width in the Insulation rubout mode. If you select "auto" the insulation width will be equal to the largest tool in the table.



In the example above 100 mils is selected. This means all pads and traces in the selected region will have an insulation of 100 mils (when possible). The algorithm that creates the tool paths is very effective and uses the smaller tool only in places that next (larger) tool cannot penetrate (without interfering with design features).

2.16 ODB++ import

Starting with version 6.0, PhCNC LSSE supports direct import of PCB designs in ODB++ format. Both compressed ODB++ archives in ZIP or GZIP (.gz) format and uncompressed folders containing an ODB++ job are supported. PhCNC LSSE can import complex production panels with a nested step-and-repeat hierarchy of up to 16 levels and as many as 2,048 PCB instances.

2.16.1 Automatic layer selection

When an ODB++ PCB design is imported, PhCNC LSSE automatically identifies and selects the layers required for machining from the layer information stored in the ODB++ job. Unlike Gerber RS-274X import, the operator does not need to determine which individual Gerber file corresponds to each PCB layer. This reduces project preparation time and the risk of missing, swapped, or incorrectly assigned layers. It also removes dependence on the file names and extensions generated by different PCB CAD systems.

2.16.2 Hierarchical panel processing

ODB++ preserves the hierarchical step-and-repeat structure of a production panel. PhCNC LSSE uses this information instead of flattening all repeated PCB instances into one large set of graphic objects. When a panel contains several instances of the same PCB, the contours and machining data for the unique design are calculated once. The result is then transformed and applied to all instances. For example, if a panel contains 16 copies of a single PCB design, its Gerber representation contains approximately 16 times more geometry to process. With ODB++, PhCNC LSSE calculates the machining data for the original design once and then places the result at all 16 panel locations.

2.16.3 Advantages of ODB++ import

Compared with importing a collection of separate Gerber and drill files, ODB++ provides the following practical advantages:

- Automatic identification and selection of PCB layers
- Reduced risk of operator errors during layer assignment
- Preservation of the original panel hierarchy and step-and-repeat structure
- Considerably faster contour and tool-path processing for repeated PCB designs
- Lower memory usage when processing complex production panels
- More efficient handling of panels containing many repeated or nested PCB designs

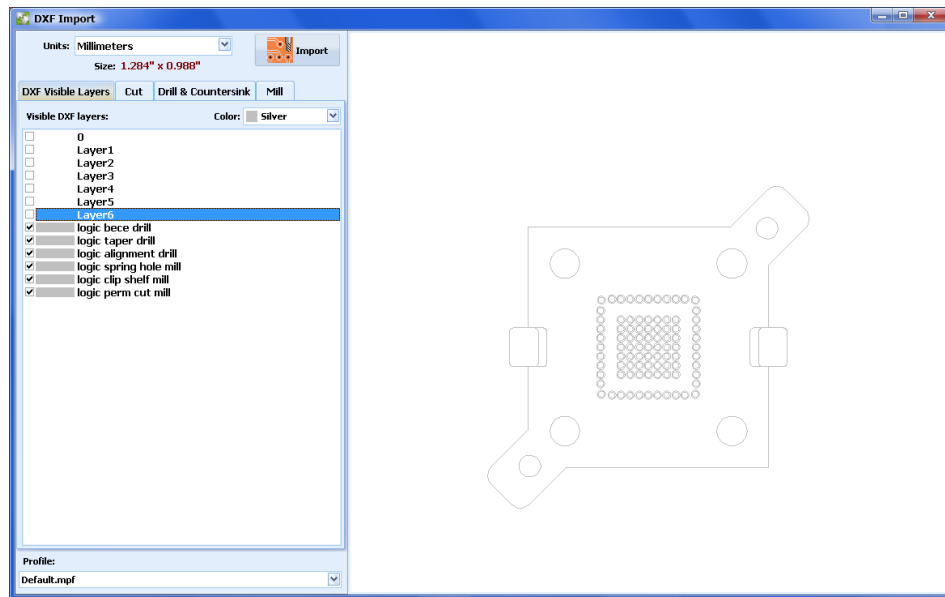
2.17 DXF import

PhCNC LSSE imports 2D DXF files (AutoCAD).

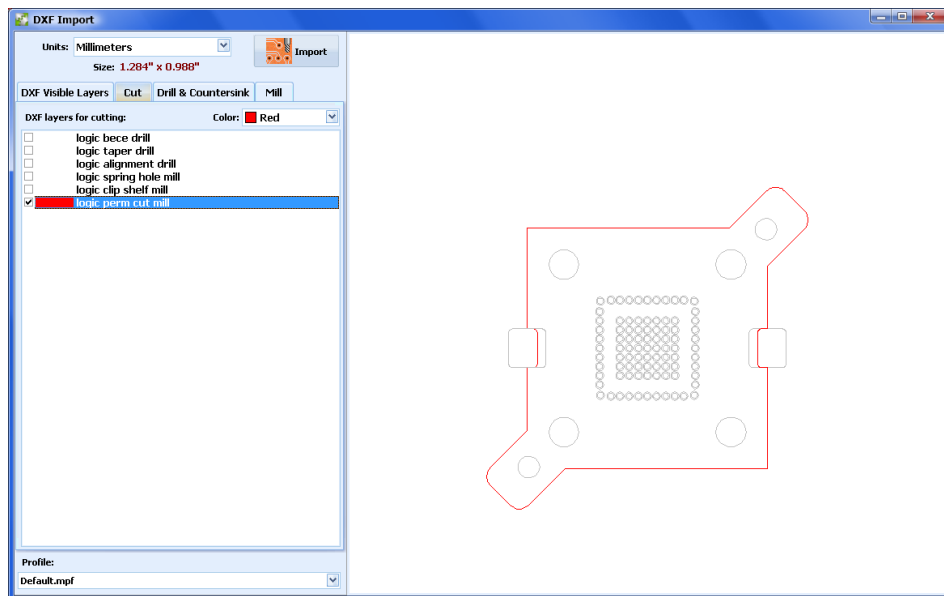
Button  initiates DXF files import in PhCNC LSSE. After DXF file selection a window for layers selection (for machining) comes up. Here are the conditions that are related to import of DXF files in PhCNC LSSE.

- Each operation (cutting, drilling, countersinking, and milling) must have a separate layer.
- Only closed contours are recognized for machining.
- In a layer that has 2 objects (for example lines) closer than 0.001", it is approximated that they are connected.
- Drill/countersink information is extracted from the existing circles in the DXF file.
- Text information in the DXF files is not processed.

Usually, DXF files contain more information than is needed for machining. By default, all layers are visible and included. As a first task you have to unselect all layers that are not related to the machining process.



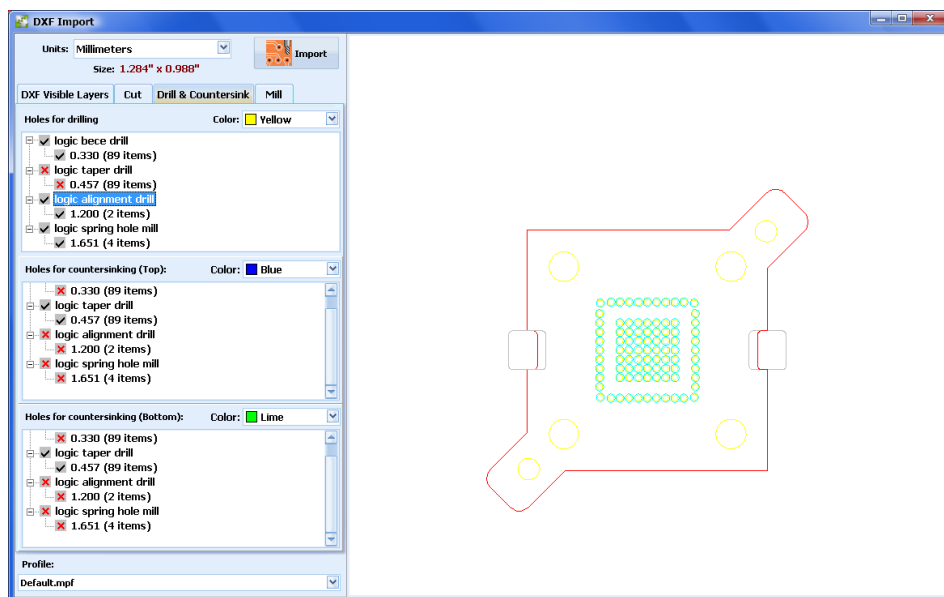
2.17.1 Selecting DXF layers for "Cut" operation



Here you have to select the layer for the "Cut" operation. Usually, it is the last one in the machining process.

In the example above it is in red (more than one layer is allowed).

2.17.2 Selecting DXF layers for Drill & Countersink



As you can see above, all layers in the list contain closed contours. They are interpreted as shapes, holes, and countersinks. Possible assignments of each layer are.

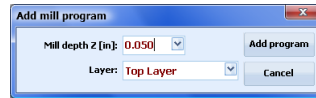
- Drill
- Countersink Top Layer
- Countersink Bottom Layer
- Milling
- Cutout

To assign a layer to desired operation replace the red "x" mark with check mark.

2.17.3 Selecting DXF layers for milling (MILL)

PhCNC LSSE is able to mill closed contours at pre-selected Z axis position (depth). For each different depth and each different layer (Top/Bottom) you have to set separate programs.

This button  adds a new milling program.



You have to fill in the "Mill depth Z" field (zero is the surface of your work piece), and "Layer" field (Top or Bottom). Z depth values are from 0.0001" to 1.0000" but actual limitation comes from the tool specifications. The deepest tool penetration that we know is 0.6".

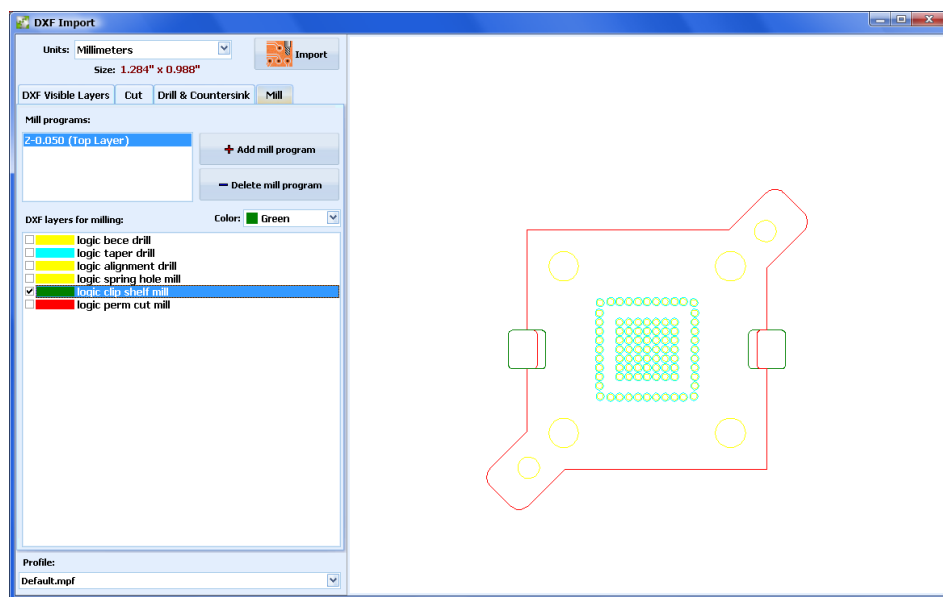
If you already have a program with the same definitions a new entry will be blocked.

Important:

Depending on the material properties, type of operations, machining aspect ratio and type of the tool you may need to use the "Peck" options. They are described in 3.4 CNC process control settings.

Once a program (Z depth/layer) is created it is necessary to select a corresponding DXF layer that will be machined with these settings (a DXF layer can be selected in more than one program).

First select the program from the list "Mill programs" and then select the DXF layer(s) that correspond to the settings.

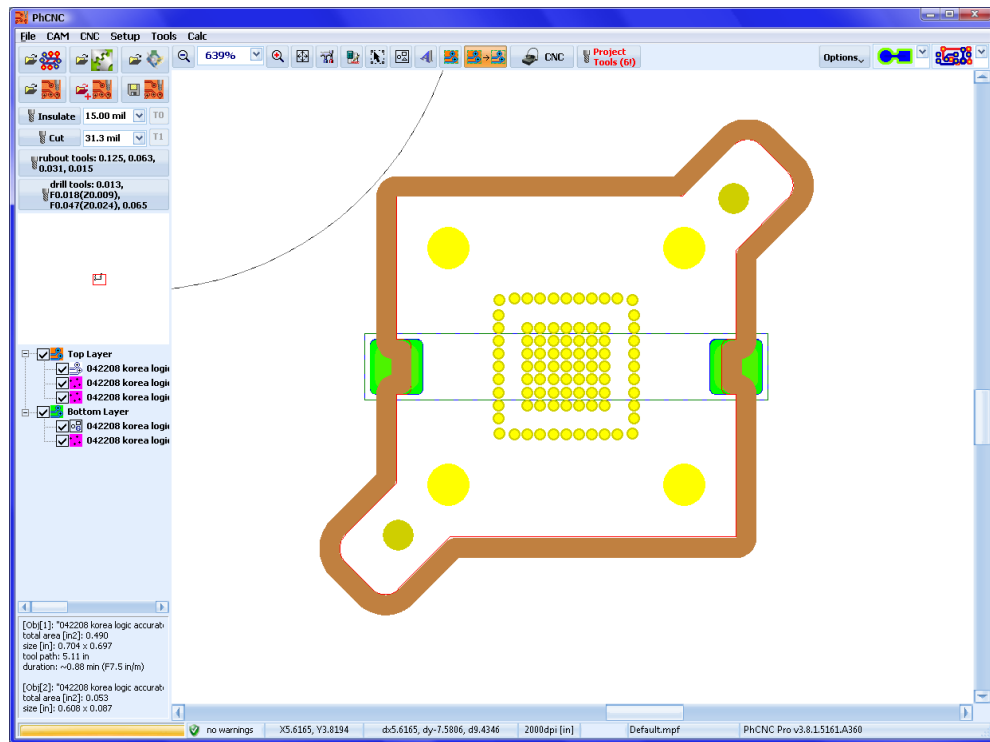


Each layer can be used in more than one program. After the import in PhCNC LSSE they will generate programs for layer insulation and rubout in CNC mode. The execution of the programs depends on the machine type. With a standard model equipped with depth limiter, PhCNC LSSE software will ask the operator to reset the depth limiter, to a new depth setting bringing the milling head to tool change position. If the machine has a program-controlled Z axis depth all operations (including "peck" mode) do not need operator intervention (except tool changes). If the machine is one of our new ATC models everything is automatic.

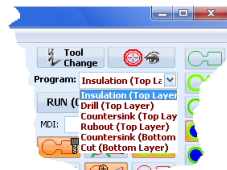
Important:

If your applications are using different materials (like FR4, plastics, Rogers, Aluminum, Brass, Bronze etc.) it is advisable to create a number of profiles corresponding to material properties (actually you may have more than one for each material if you need to). You can set and create profiles in the "Setup" menu (see 3.4 CNC process control settings). All the profiles you create are listed in the "Profile" list and can be selected from it.

Example of an imported DXF file:



After the process of importing the DXF files and all arrangements, you will enter the CNC mode. PhCNC LSSE will generate all machining programs needed, according to your settings.



The machining programs will be one of the following:

- Drill (Top Layer) is for drilling holes on the top side.
- Drill (Bottom Layer) is for drilling holes on the bottom side.
- Countersink (Top Layer) is for countersink machining on the top.
- Countersink (Bottom Layer) is for countersink machining on the bottom
- Insulation (Top Layer) will create the contours of the machined region on the top.
- Insulation (Bottom Layer) will create the contours of the machined region on the bottom.
- Rubout (Top Layer) will machine the internal part of the regions on the top.
- Rubout (Bottom Layer) will machine the internal part of the regions on the bottom.

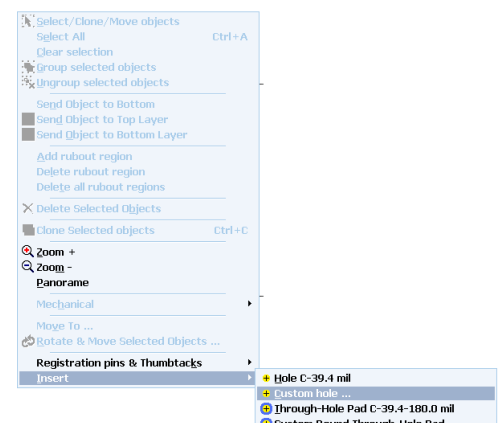
MTC models will require the operator to change tools, flip the part over, but the settings of the Z axis depth will be done automatically from the information included in the program. The new ATC models will require no intervention by the operator other than flipping the part over to be machined.

2.18 Holes and pads insertion

CAM mode of PhCNC LSSE allows you to insert additional holes and pads in order to meet the needs of fiducial registration process. This option can also be used for many other purposes like for instance when you need an additional hole or pad in your project. All inserted holes or pads are treated as separated objects and can be moved, grouped, selected...etc. as any object in CAM mode of PhCNC LSSE.

2.18.1 Holes insertion

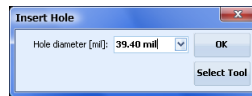
The submenu of "Insert" (from main popup menu) will provide



you with two options

- Hole with current diameter (visible in the menu)
- Hole with custom diameter (it will become current)

Below is the window for custom diameter.



The centers of inserted holes will be placed on the mouse cursor position at the time when you call the popup menu (right click). If you need to determine them accurately use the high zoom feature. Actual position of the holes used for references in fiducial registration is not important. Place them anywhere convenient for your design (we recommended along the X axis). Placing them at the maximum distance apart will improve the accuracy of your fiducial registration. Using the same Y coordinates will make finding the second fiducial easier.

Note:

There is no limitation where to place your holes. They can be placed outside of your project or inside the existing objects from your project.

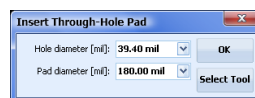
Important:

Holes that will be used for fiducial registration (existing in the project or added as dedicated for the registration) MUST be in the camera access frame (The smaller internal frame in your CAM screen).

2.18.2 Through-hole pads insertion

The process of inserting through-hole pads is the same as in 2.18.1

You have two choices, a current size or customized (that becomes current after usage).

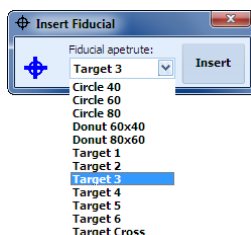


Notes:

- The pads are insulated circles that are entered in both the top and bottom layers.
- The drill diameter for the hole and through-hole (with pad) is the same if you change one, the other will follow
- Pad diameter can not be smaller than hole diameter. If pad diameter is entered smaller then the hole diameter it will be equal to the hole diameter.

2.18.3 Insert Target ...

Menu option "Insert Target ...", allows inserting of different "target" apertures at the mouse cursor position.

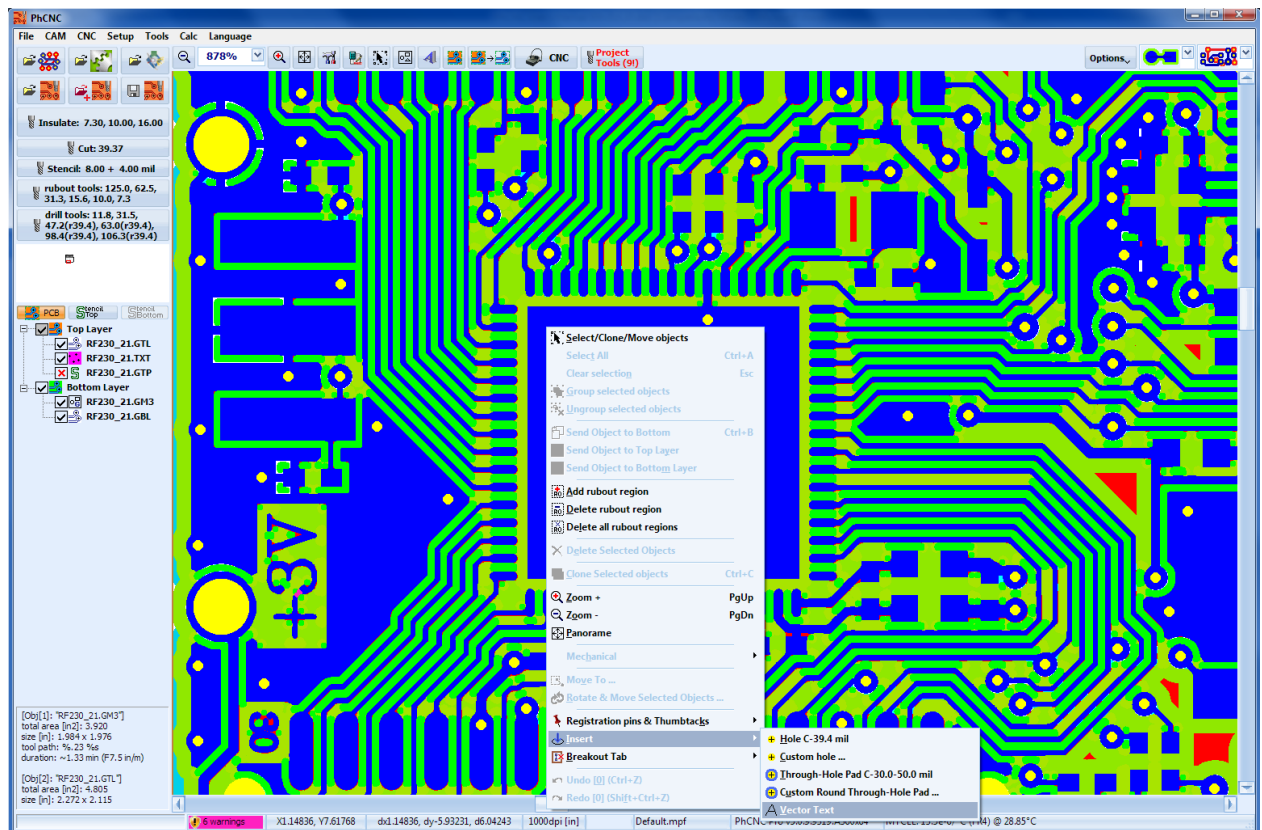


All apertures are predefined but adding new ones can be done by request.

2.19 Vector Text insertion function

PhCNC LSSE allows you to insert simple vector text into your design or as a separate project. The execution of the text is in "Insulation top" or "Insulation bottom" programs in CNC mode. This function offers a single line of text which is different from standard text (see 2.12.3) where the symbols are engraved as an outline of their shape. The thickness of the line in this mode is equal to the size of your insulation tool. The vector text function will allow you to insert very small size symbols in your project. For example, 32 mills (0.8mm) high symbols look good when you are using a typical 8 mil (0.2 mm) insulation tool.

To use this function, you have to select Insert from the popup menu:



You will then be presented with the vector text window as seen below:



The upper portion of the window is a simple text editor. The lower portion shows how the text will actually appear according to your settings.

The text size will be the same as the standard text sizes found in other text editor programs.

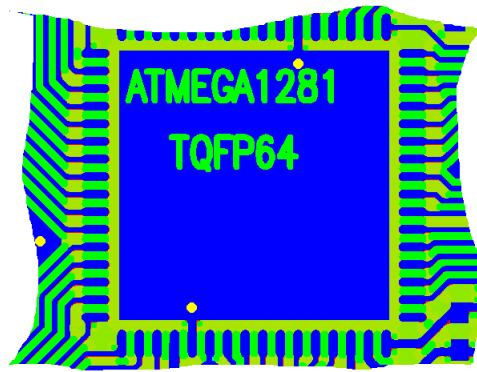
Buttons are for text slant (left or right).

Buttons are to change symbols width.

Buttons are to change the spacing between the symbols.

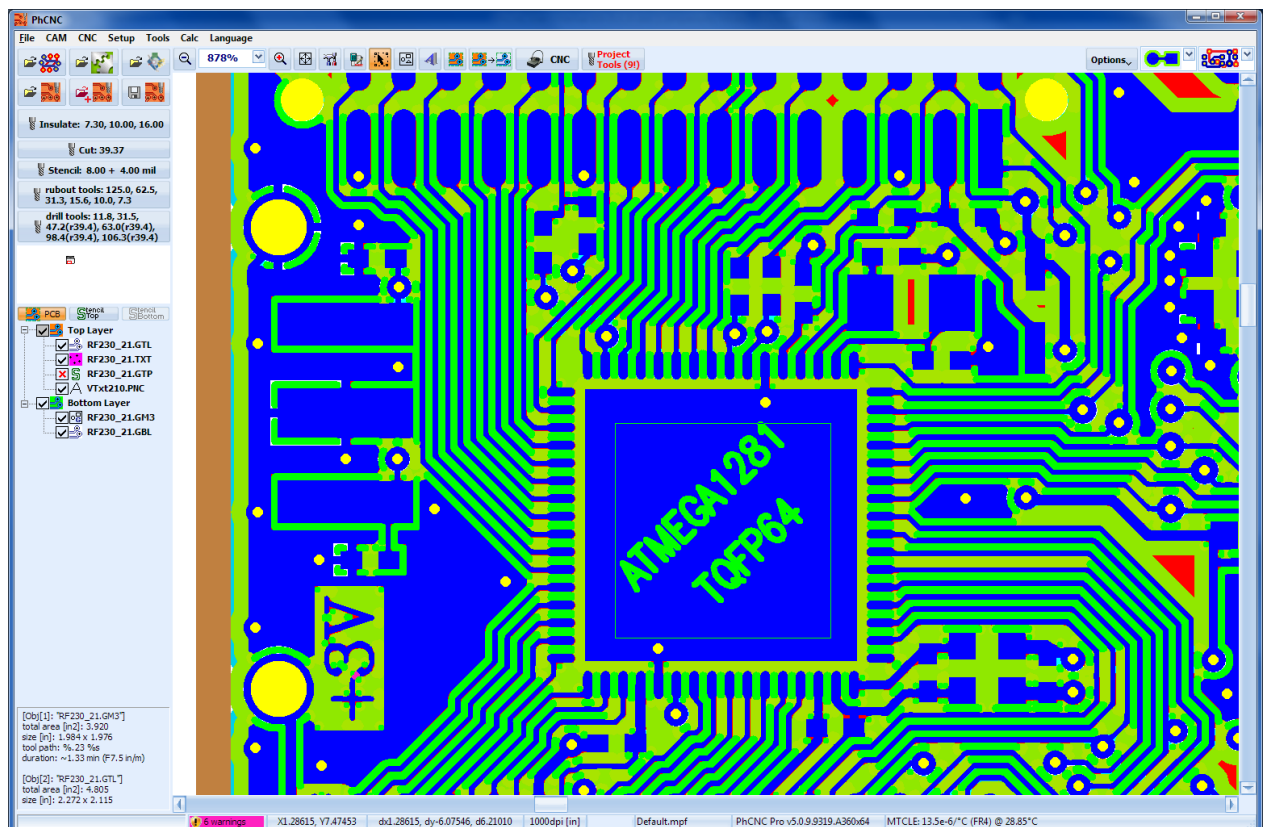
Buttons are to change the spacing between the lines (if there is more than one).

After you confirm your text is ready with "OK" button the text will be inserted in the position of the mouse cursor that you initially called the popup menu from.



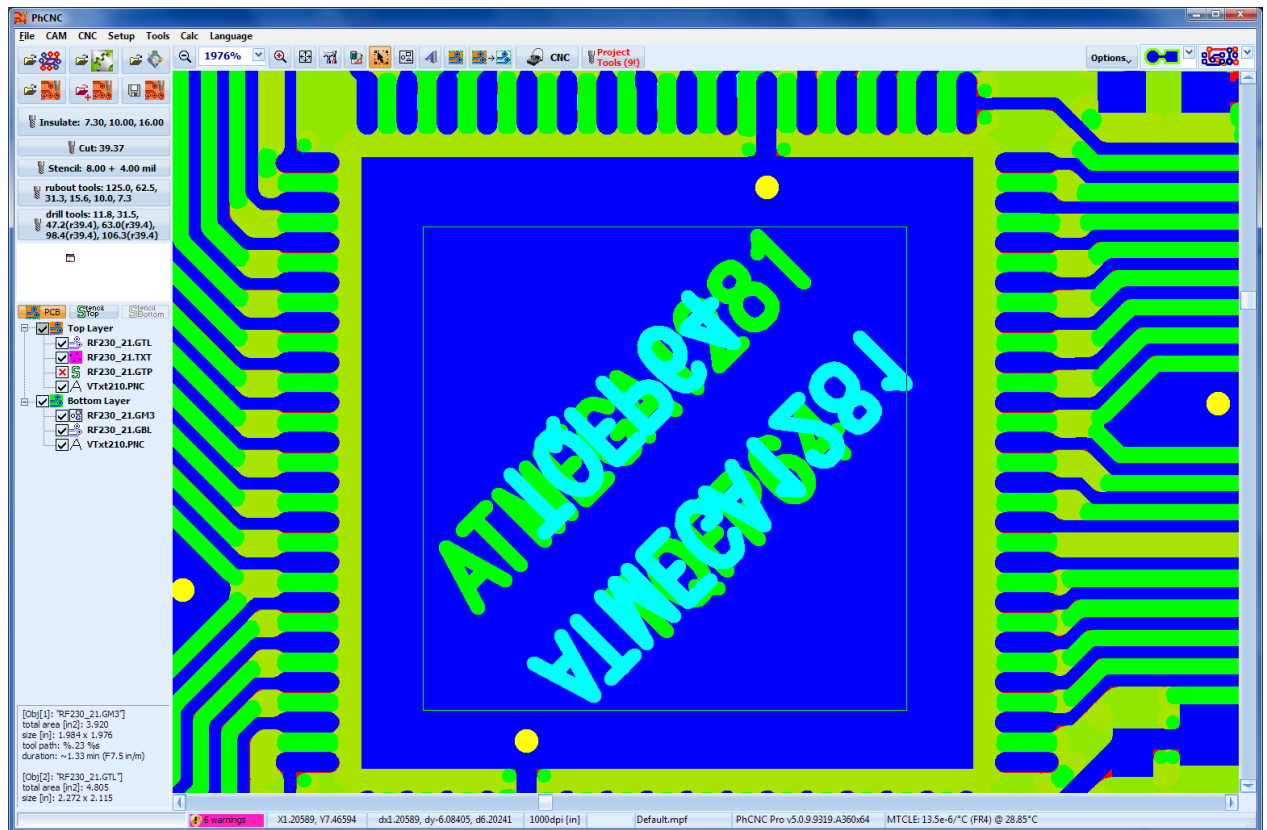
Inserted text objects can be manipulated just like other objects can they can be cloned, rotated, moved, grouped etc.

In the sample below the text is rotated at 45 degrees:



If you need to insert text in bottom layer, you have to use the (drag & drop) function.

In the sample below text from the sample above has been cloned and moved to bottom layer.

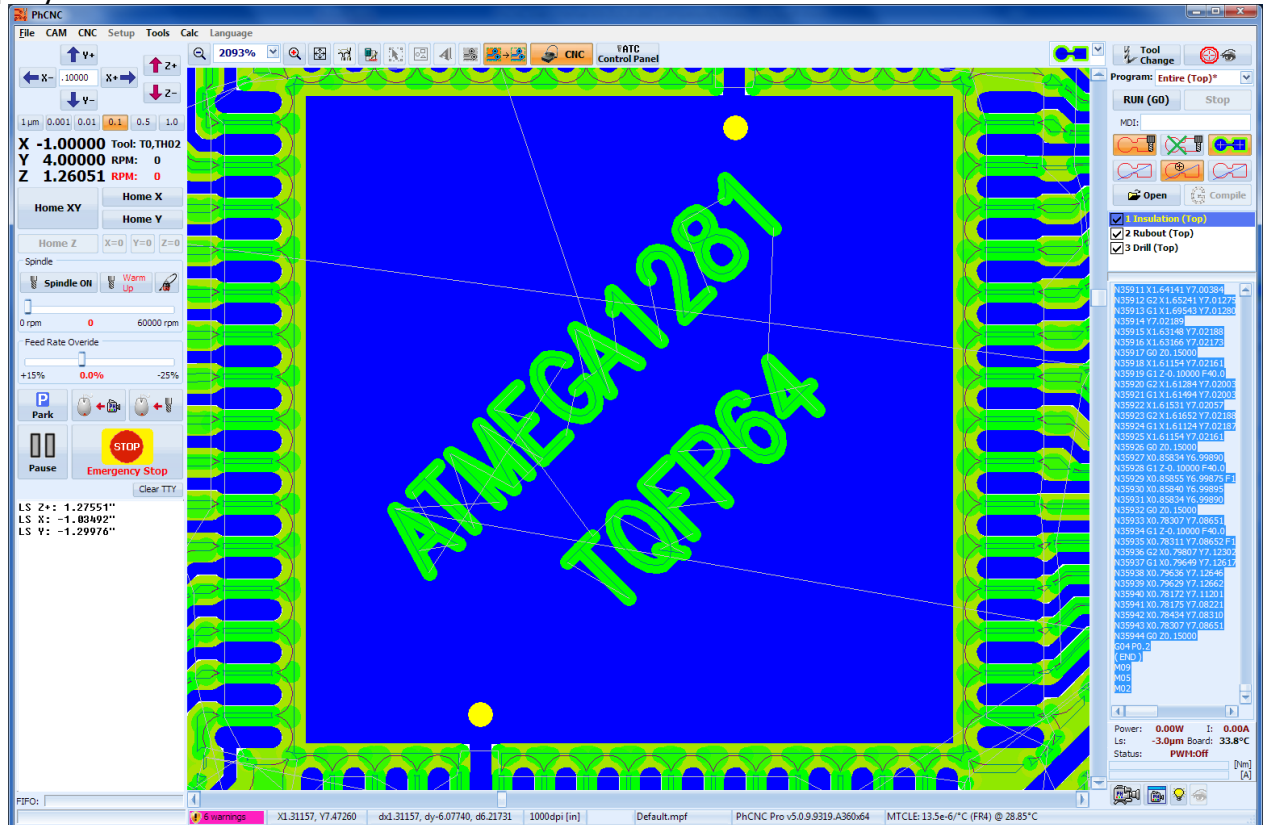


Note:

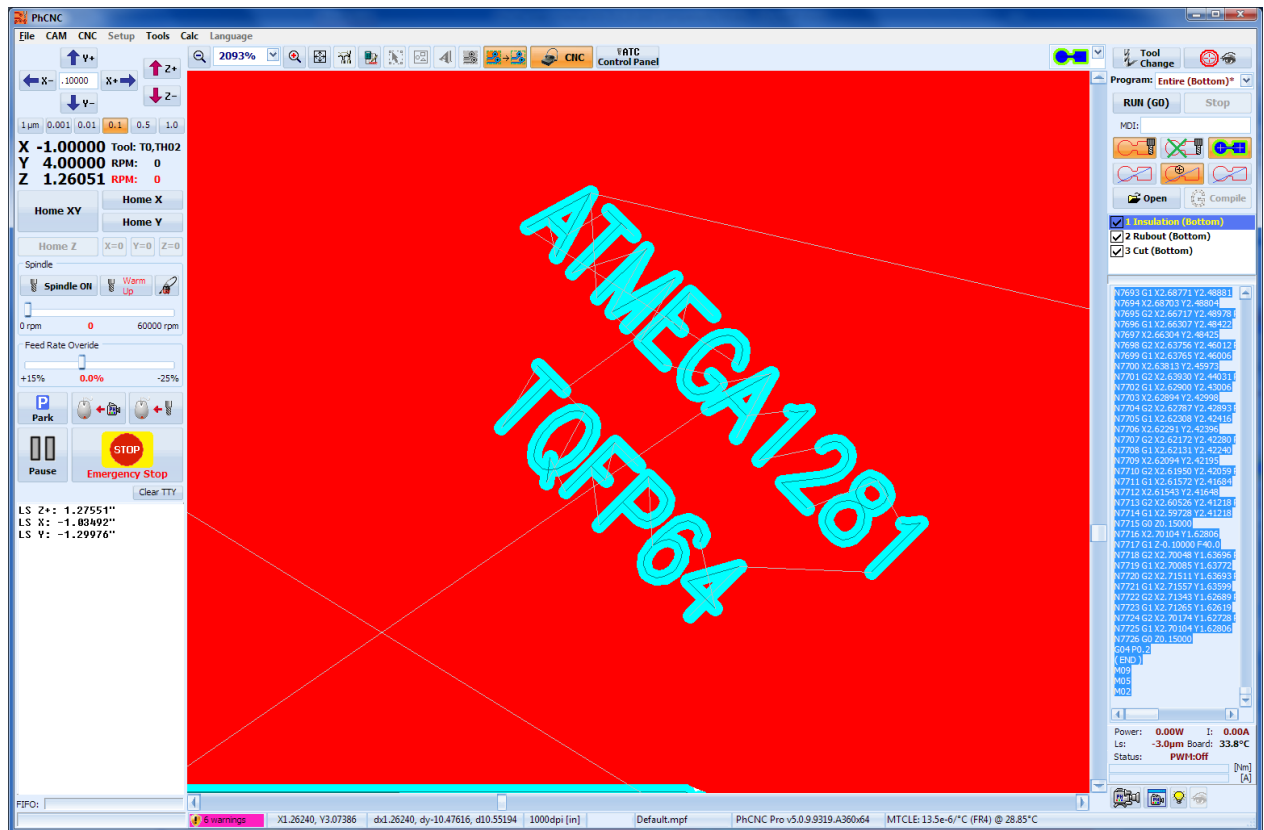
When the text is moved to bottom layer it is automatically mirrored along the Y axis. It will be mirrored again when you machine the bottom layer and will become readable.

Machining programs in CNC mode will automatically include the text in the machining process.

Top layer:



Bottom layer:



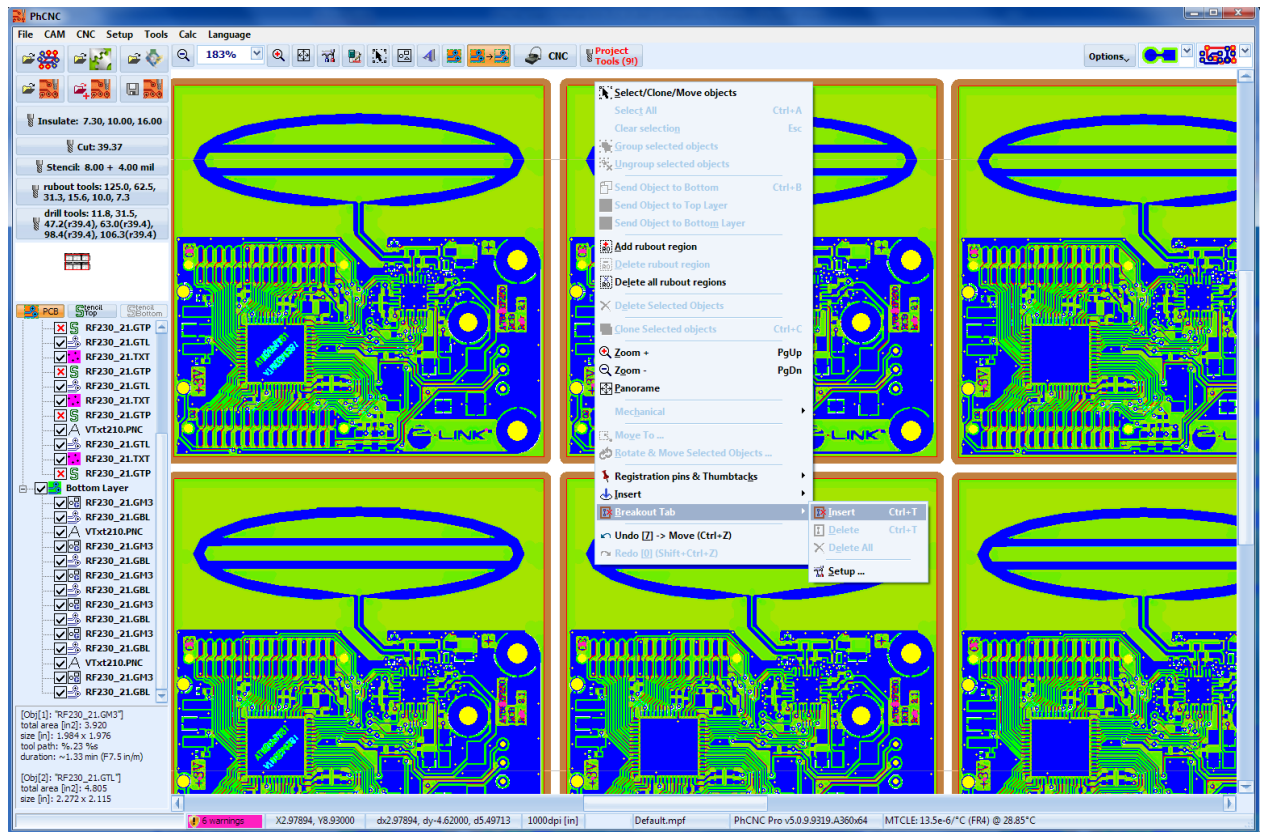
2.20 Breakout Tabs

PhCNC LSSE allows you to insert breakout tabs in the cutout contour.

Breakout tabs are small interruptions of the cutout contour that allow your ready board to remain firmly attached to the main work piece until you decide to remove it by breaking them. Using breakout tabs will allow your finished board to remain on its relative position to the work piece and any registration holes that may be in the work piece. This will allow you to use the registration holes for further operations like solder dispensing or automatic component placement. It will also prevent your board from being picked up and dragged by the vacuum (debris collection) system.

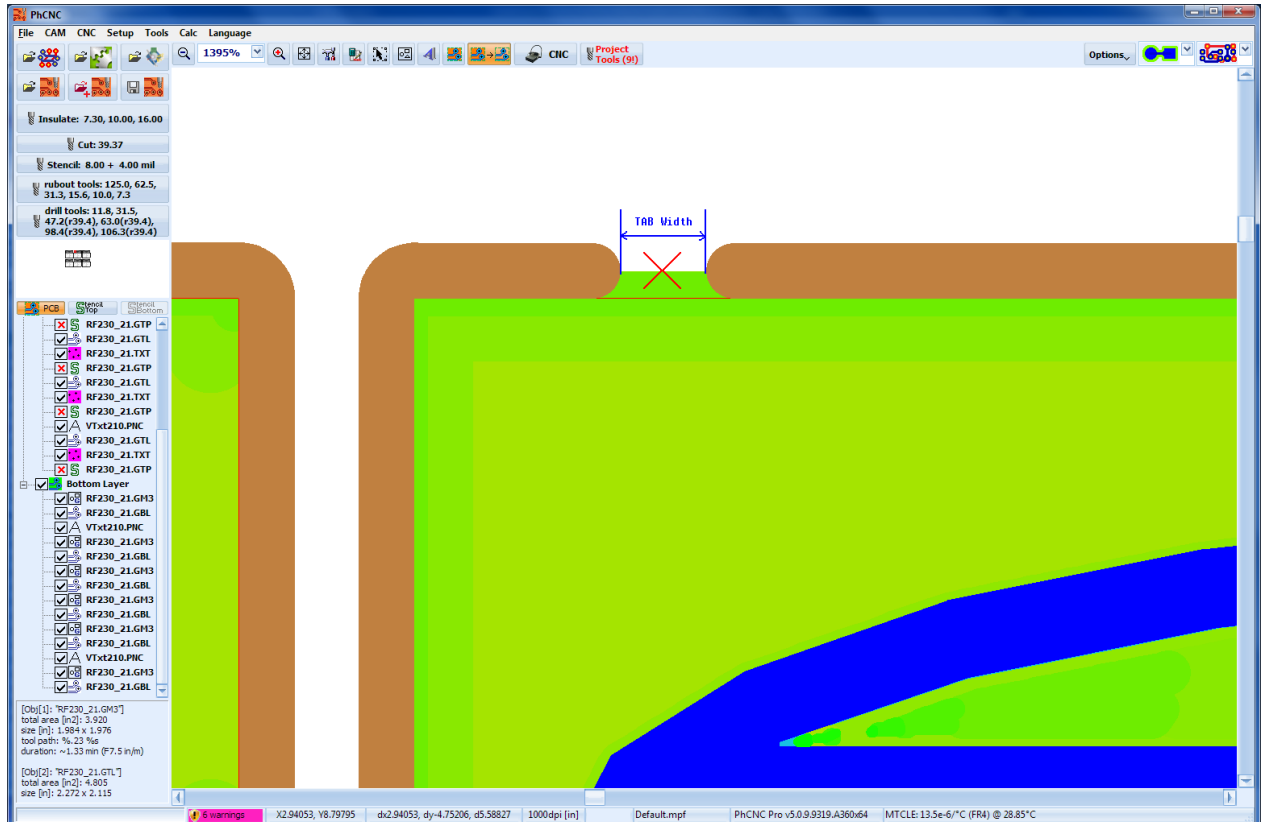
2.20.1 Breakout tabs insertion

Breakout tab insertions are possible in PhCNC LSSE in the CAM mode. Place the mouse pointer at desired position of cutout contour and make a right click. Select "Breakout tab" menu then select the "Insert" option from the list as seen in the sample below:



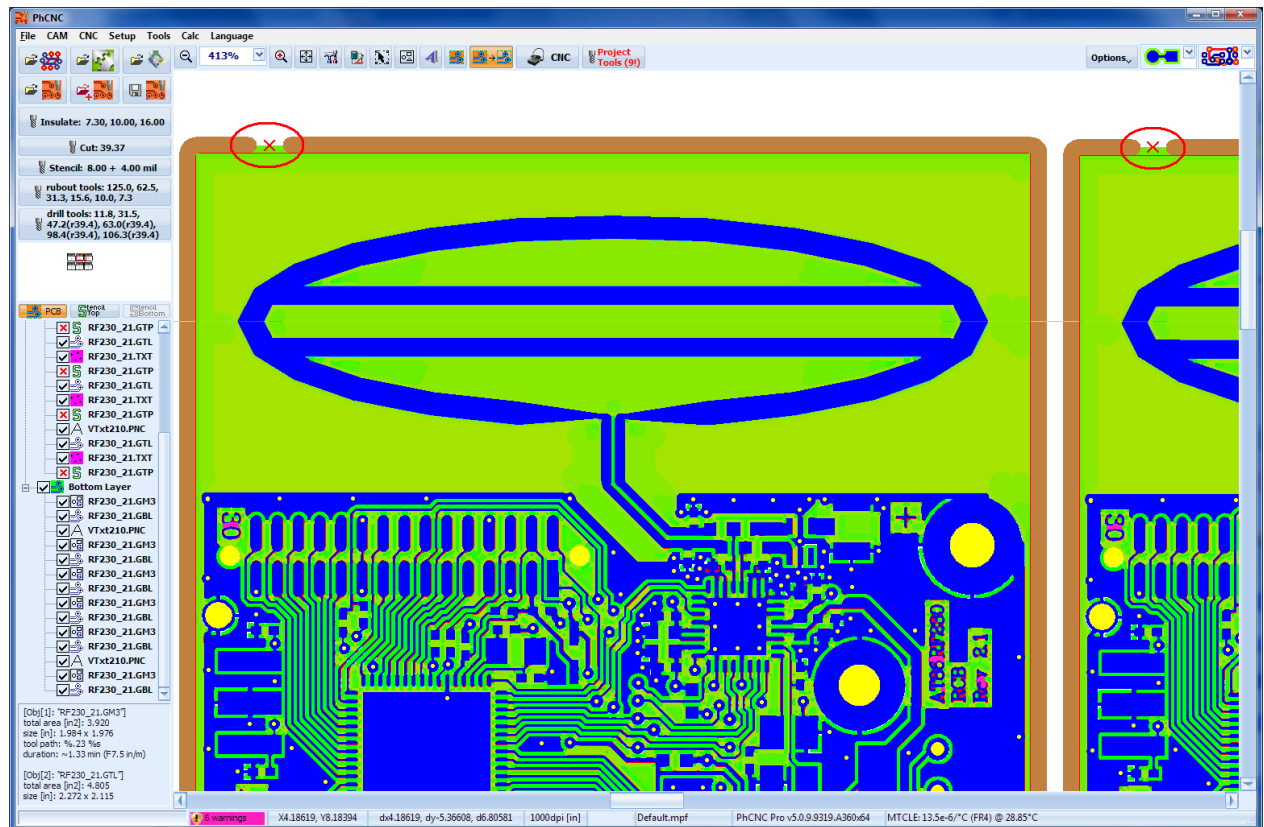
The "Insert" option is allowed only when the mouse pointer is on a cutout contour (mechanical layer).

Shown below is the inserted breakout tab:



Tab width is calculated by PhCNC LSSE and is independent from other contour parameters e.g. if you set 20 mils, the remaining material (at the closest point) will be 20 mils.

When you have cloned objects, inserting breakout tabs in one object will spread to all copies. Thus, inserting tabs before or after cloning will have the same result.



Shortcut: Place the mouse pointer on the desired point of the cutout contour. Use **Ctrl+T** to insert tab.

Note:

Tabs cannot be inserted over another existing tab, or placed where there is not enough space for the tab.

Note:

- Tabs positions and size are stored in the .PHJ project files so when you load a project file it comes with the individual settings of the tabs already in place.
- When you Import a project file it uses the current settings of the tabs. The imported files tabs will be recalculated in order to use current cutout tool.

2.20.2 Breakout tabs removal

To remove a tab place the mouse pointer over the small "x" (tab center) and make a right click. A popup menu will then appear. Select option "Breakout tabs" from the menu then select option "Delete" from sub menu.

Shortcut: Place the mouse pointer on the desired point of the cutout contour. Use **Ctrl+T** to remove tab.

Note:

Removing a tab from a cloned object will remove the tabs from all copies.

2.20.3 Breakout tab width setting

Select "Breakout tabs" then select "Setup".



By "default" tab width is 20mil (~0.5mm) but can be changed to meet certain material requirements. When the width is changed PhCNC LSSE recalculates all tabs to the new setting. If the width is increased and there is not enough space for a particular tab it will be removed.

Important:

If you change the cutout tool diameter, PhCNC LSSE will remove all tabs. Therefore it is recommended to insert tabs after the selection of a cutout tool.

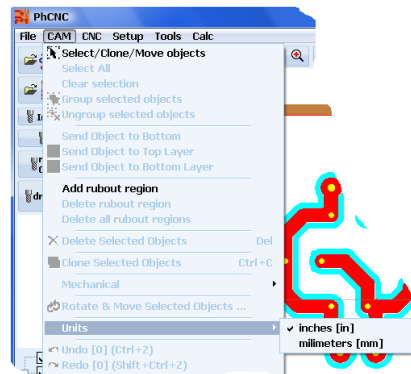
2.20.4 Breakout tabs complete removal.

Select "Breakout tabs" and then select option "Delete All" from sub menu.

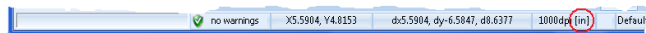
2.21 Units of measurement [mm]/[in]

PhCNC LSSE (1.15.2+) now handles imperial [inch] and metric [mm] units. By default, the active measurement units are set to inch and this will allow for the use of mils [mil], (1/1000 of an inch).

Switching between measurement units can be done from the Cam menu by selecting the Units menu option.



Selected units are shown in the status bar:

**Note:**

Tool profile settings now are using different units for feed & plunge rate. Imperial units are using [in/m] (inch per minute), metric units are using - [mm/s] (millimeter per second).

2.22 UNDO/REDO

PhCNC LSSE has an UNDO/REDO function. The size of UNDO/REDO buffer is 32 commands deep.

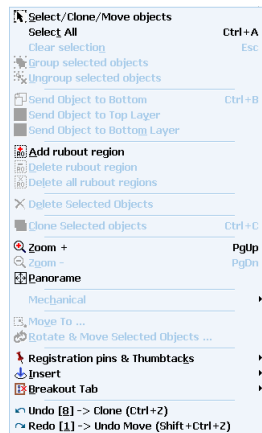
The following operations can use the UNDO/REDO function:

- MOVE (objects move)
- IMPORT (import of project or gerber/drill files)
- CLONE (copy/clone of objects)
- GROUP (group of objects)
- UNGROUP (ungroup of objects)
- SEND_TO_BOTTOM (Send to Bottom)
- ROTATE (rotation of objects)
- DELETE (removing of objects)
- INSERT_HOLE (insertion of drill holes)
- INSERT_PAD (insertion of pads)
- INSERT_TTF_TEXT (insertion of TTF text)
- INSERT_VECTOR_TEXT (insertion of vector text)
- NEW_PROJECT (new project)
- FIDUCIAL_MOVE (positioning project - CNC mode - point one) available in PhCNC LSSE only
- FIDUCIAL_ROTATE (rotating project - CNC mode – point two) available in PhCNC LSSE only

If you start a New Project by mistake you can "UNDO -> New Project" This will take you back to the old project as it was when you were there before the New Project was opened. But there is no REDO for New Project you simply need to select New Project for the file menu again.

Note:

- Undo/Redo commands are located in the CAM menu and can only be used with CAM functions.



Each Undo/Redo command contains the information for operation that is reversed, and the digit in square brackets shows the number of commands in the corresponding buffer.

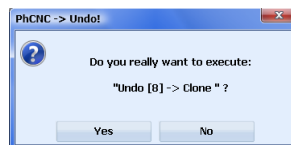
Undo and Redo have short cuts on the keyboard Ctrl+Z (Undo) and Shift+Ctrl+Z (Redo).

Most of Undo commands activated by Ctrl+Z need additional conformation.

Commands that are executed without conformation are:

MOVE, GROUP, UNGROUP, ROTATE & SEND_TO_BOTTOM

The rest of the commands when called by Ctrl+Z, will display the window bellow:



Note:

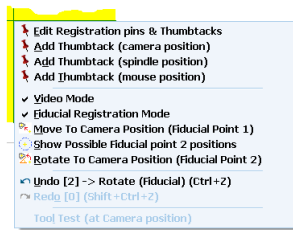
- All UNDO commands activated from menu are executed without conformation.
- All REDO commands despite how they are activated DO NOT require conformation!

REDO reverses the last UNDO command, but if you have another command that is in the UNDO list, the REDO buffer will be cleared.

Important:

REDO/UNDO buffers are cleared after execution of the following commands:

- Load Project
- Save Project
- Entering CNC mode
- Tool size change (insulation; cut; stencil) or project dpi
- Execution of the G-Code program (CNC mode)
- UNDO/REDO in the CNC mode is only for FIDUCIAL_MOVE and FIDUCIAL_ROTATE (PhCNC LSSE):



2.23 Forced Insulation

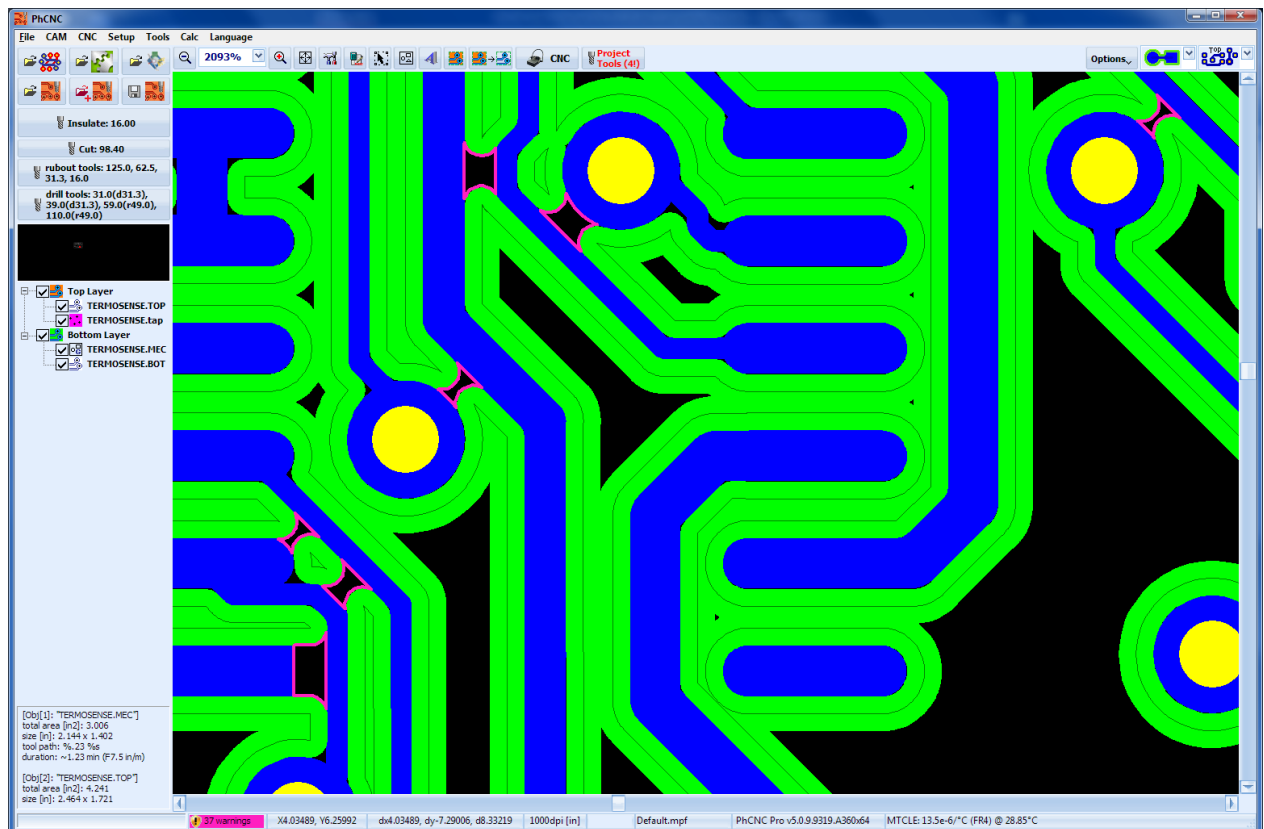
PhCNC LSSE has a function "Forced Insulation" that allows the system to handle sections of the design marked by DRV (See 2.13 Design Rules Violation) as impossible to insulate. The standard response of PhCNC LSSE tool path generation doesn't allow the tools to penetrate inside the traces/pads contours in this situation. When the distance between the traces/pads is smaller than selected insulation tool diameter it leaves them not insulated. Those sections are shown in different color (pink-red default settings) when the DRV is on. The "Forced Insulation" algorithm creates a practical compromise, reducing the size of both (all) features affected and will generate a single tool pass that reduces the size of them equally.

Button  is for "Forced Insulation" and has the following functions:

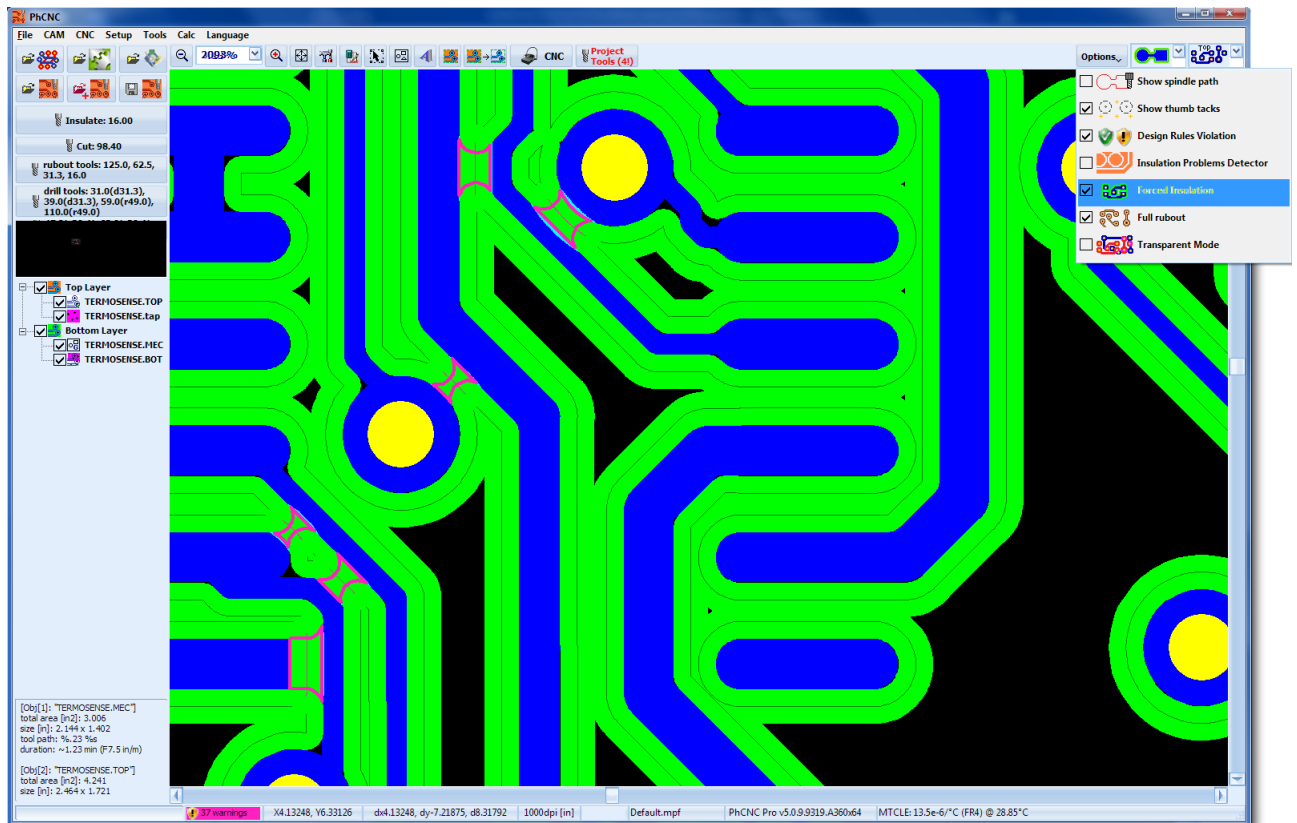
- Calculates tool paths used in "Forced Insulation". If the tool paths are never calculated the first activation will do so. If DRV function wasn't activated it will start it.
- Shows/hides tool paths related to "Forced Insulation".

This option can be activated also from Insulation Tools Window!

Shown below are sections of a design detected by DRV:



Shown below are the same sections insulated by "Forced Insulation" function:

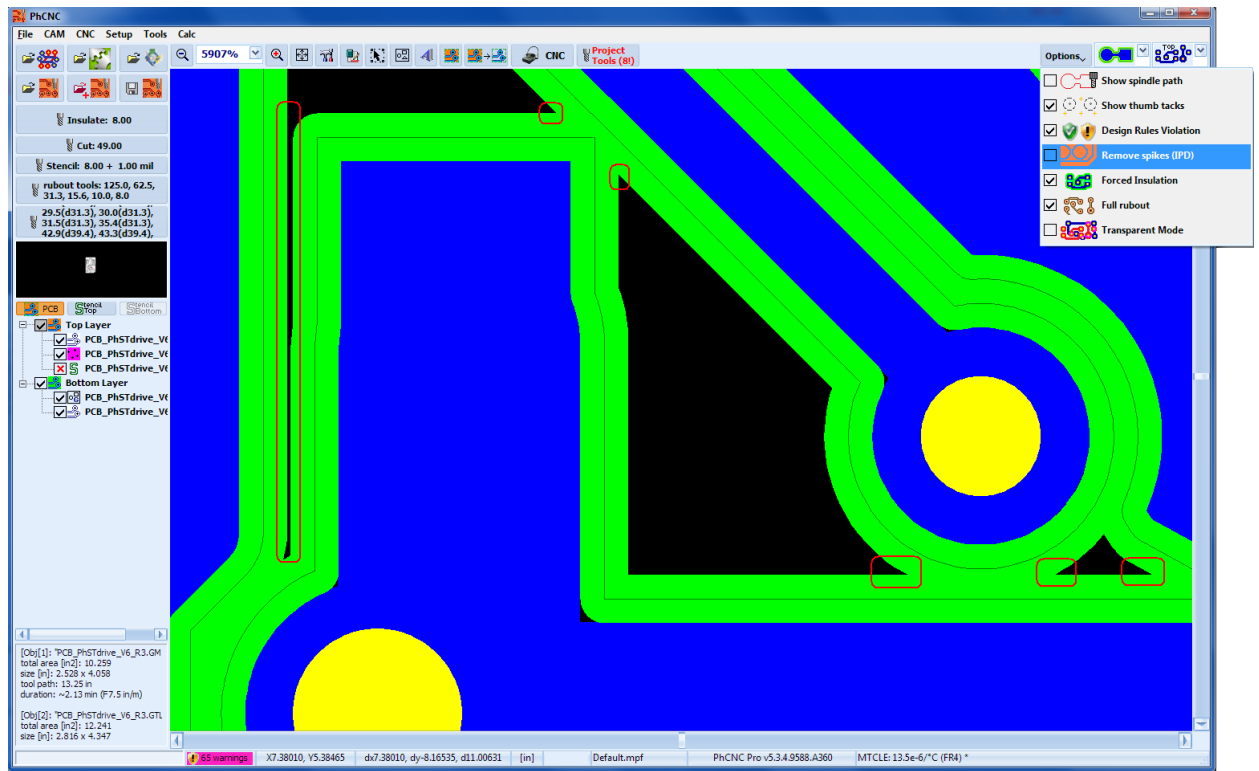


Light blue (default) strips shows additional copper removed from the design features when zoomed in to view the "Forced Insulation" paths.

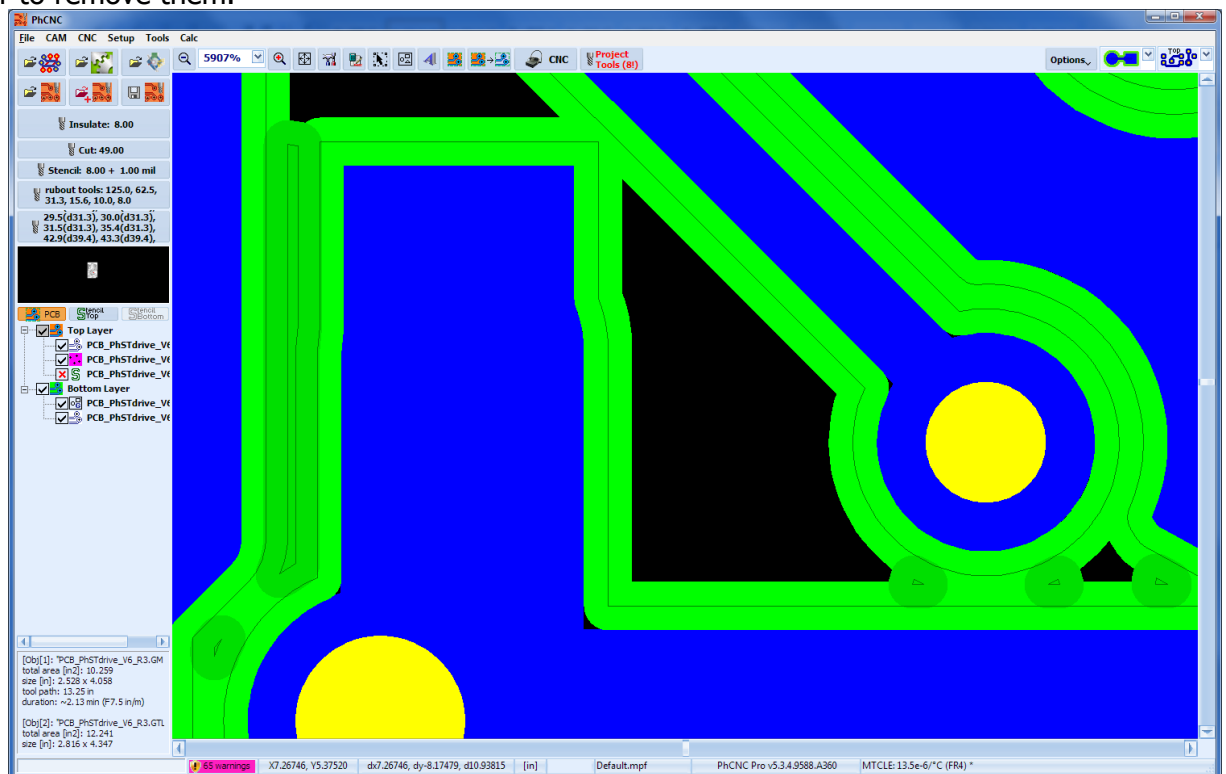
Button  will also serve as a switch to include/exclude the problematic sections in the machining process. "Forced insulation" tool paths are not automatically recalculated when you change the insulation tool diameter. If you change the insulation tool diameter you need to run the DRV function again to see if the new tool diameter will correct the DRV(s) or leave an acceptable amount of copper on the affected areas when using "Forced insulation".

2.24 Remove spikes (IPD)

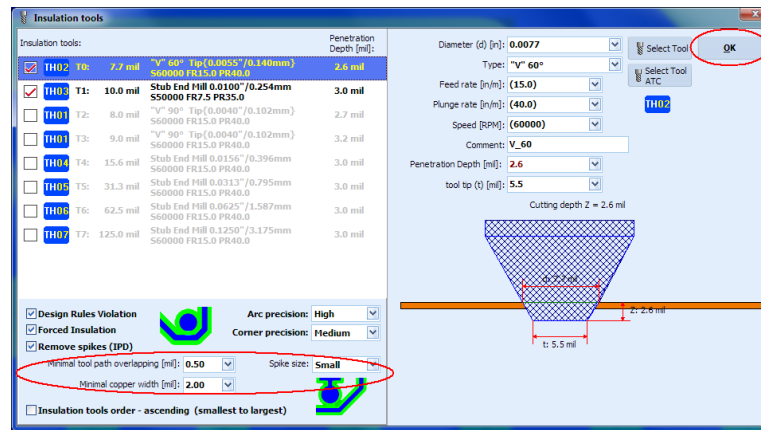
When the insulation only is used (not full copper rubout) very tiny copper strips may be left over from machining (hair sized). They do not have enough foundation to the board substrate and usually become free moving copper hairs that can create a condition for fine (hard to see) shorts on the board. It is relatively easy to remove them using "scotch bright". In some cases (extremely fine board pitch) the use of "scotch bright" is not recommended just because it may damage the original circuit. PhCNC LSSE software offers a secure option "Remove Spikes (IPD)" (Insulation Problems Detector) that locates and removes this potential source of problems. Shown below are some examples.



With the activation of IPD, the zones that have potential problems will be additionally machined in order to remove them.



Remove spikes (IPD) has optional parameter settings in the Insulation Tools window:



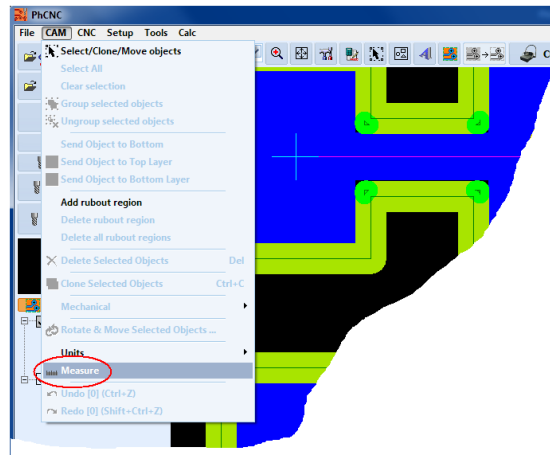
The parameters you may set are:

- minimal distance between tool paths that can be considered problematic (usually 0.5 mil).
- minimal residual copper width that can be considered source of potential problems (usually 2 mil).

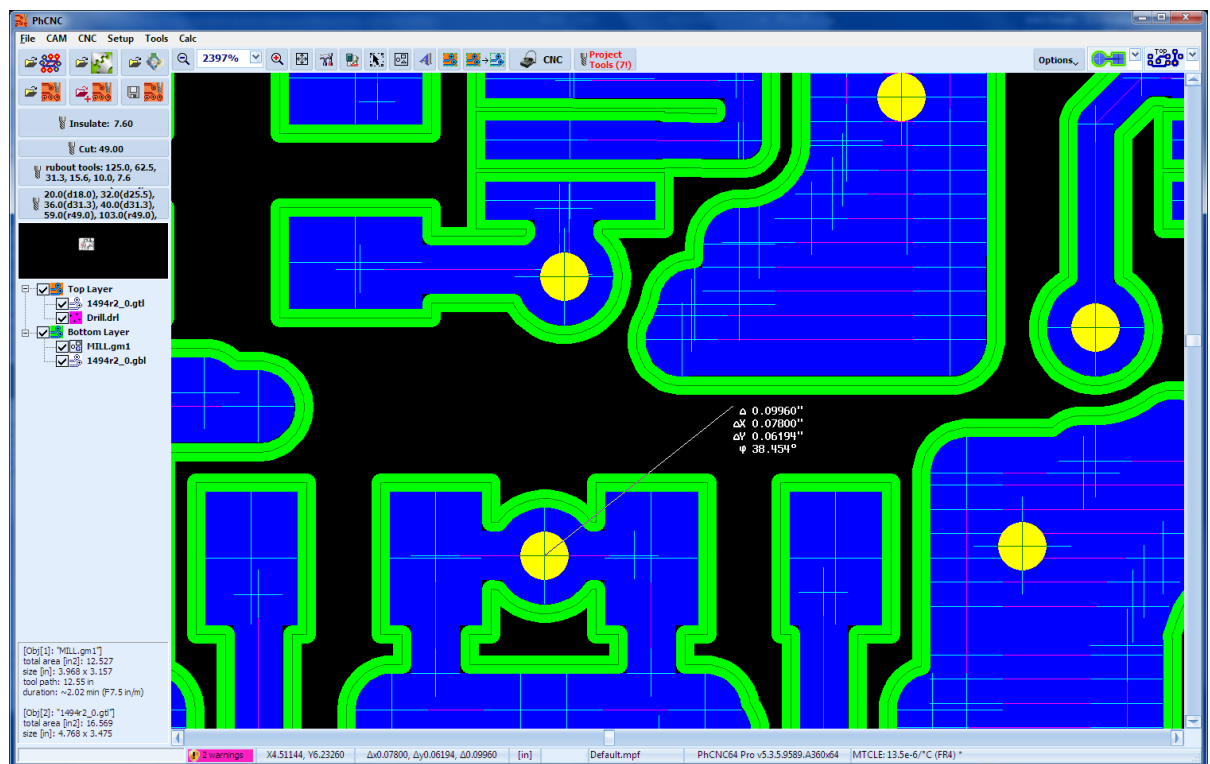
As a bottom line, using IPD when you do not use full copper rubout is very recommended.

2.25 Measure

This is a very convenient function that allows the user to measure distances on the project. After activation it shows the distance to the current mouse position. To measure distance place mouse cursor on the first point and make a left click. Move the mouse to the second point and read the measurement results. In order to have maximum accuracy use high zooms.



Canceling the Measure function can be done from the menu or simply by pressing space bar. Another convenient way to use it (for single measurement) is to place mouse on the first point, press and hold the space bar, move the mouse to the second point and read the results.

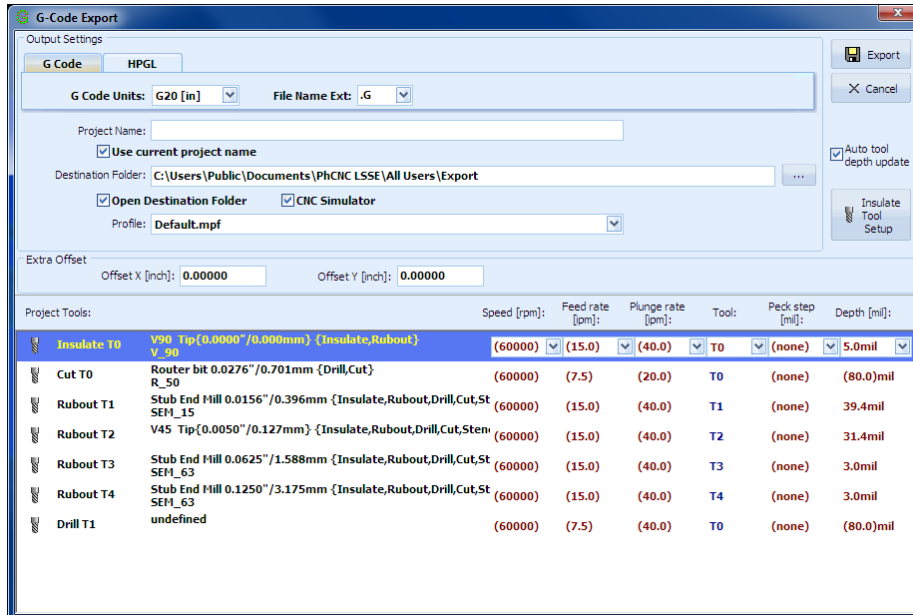


On screen information contains the distance, X difference, Y difference and the angle related to X axis. It is shown in the measurement units selected (inch/mm).

2.26 G-Code Export from PhCNC LSSE

PhCNC LSSE generates standard G-Code for a wide range of CNC controllers. The generated syntax can be adapted to the target controller using the post-processor and machine compatibility settings described in section 3.3.

The button  in PhCNC LSSE calls a window with two tabs for G-Code / HPGL export:



PhCNC LSSE, generates a G-Code / HPGL program for each machining process needed in your design:

- Insulation (Top)
- Insulation (Bottom)
- Drill
- Rubout (Top)
- Rubout (Bottom)
- Cut (Bottom)
- Stencil (Top)
- Stencil (Bottom)

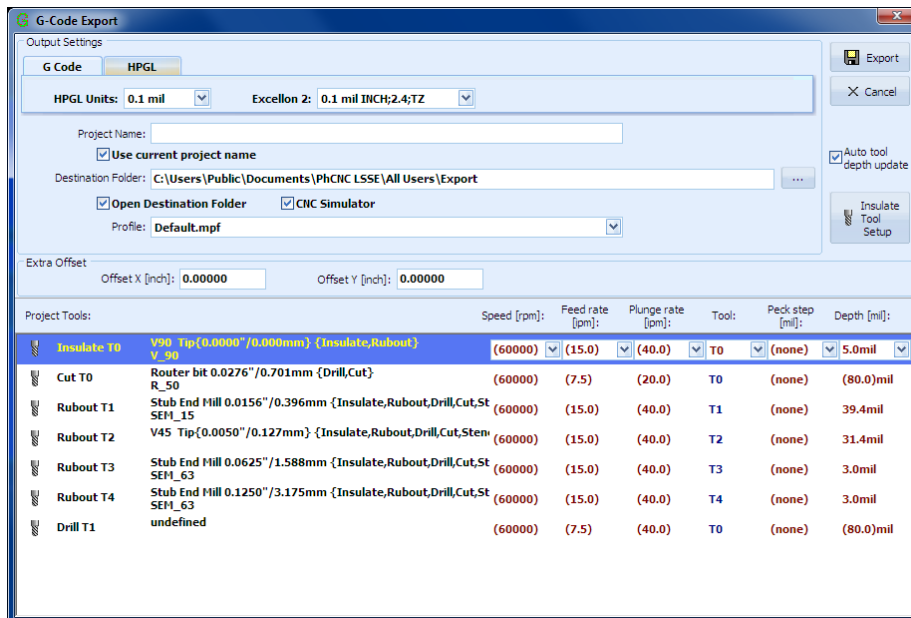
You have two options to export: G-Code or HPGL.

Basic parameters that you must determinate are:

G-Code Units:

G-Code output units are normally initialized from the units currently selected in CAM mode: inch selects G20 and mm selects G21. The units can be changed temporarily in Setup -> G-code. See section 3.3.4 for details.

HPGL Units:



Units of measurement that will be generated in case of HPGL export. They can be as follows:

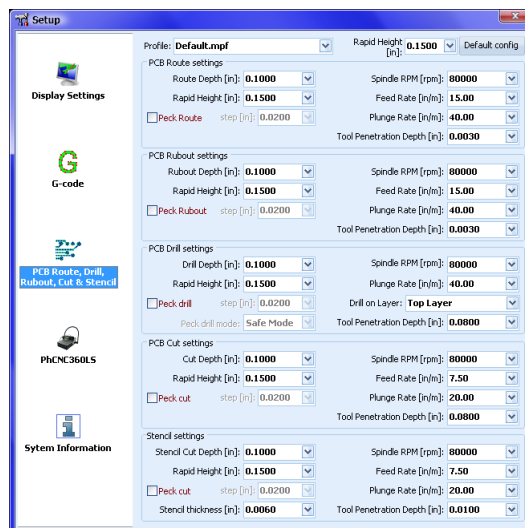
- HPGL units: 0.010mm, 0.025mm, 1.0mil, 0.1mil
- Excellon2: 0.010mm, 0.025mm, 1.0mil, 0.1mil

Tool penetration for each operation:

For systems that are using a depth-limiter, penetration is set in Setup menu.

Note:

the actual tool penetration depends on your depth limiter settings.



If you are using a system without a depth limiter (we expect this to be so in most cases), the penetration depth should be set according to the calculations for V shaped tools. The rest of tools used such as End mills, Drill bits and routers will use the values set in the Setup table (Tool Penetration Depth).

The operator will be able to make changes (if needed) in the selected tools penetration depth (see paragraph 2.4.6 Project Tools)

Insulation tool Z depth: Systems without depth limiter:

When the V tool is chosen, the tool penetration is calculated by PhCNC LSSE, it calculates the penetration depth based on the tool geometry.

When an End mill is in use, the penetration is set in the Insulation block of the Setup.

Rubout tools Z depth:

This sets the penetration depth of rubout tools.

Important:

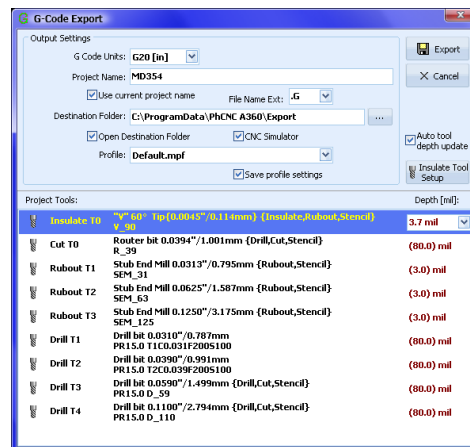
Our rubout program always uses the insulation tool as the first tool for rubout. The following tools are normally stub end mills and have to have the same penetration settings.

Project Name:

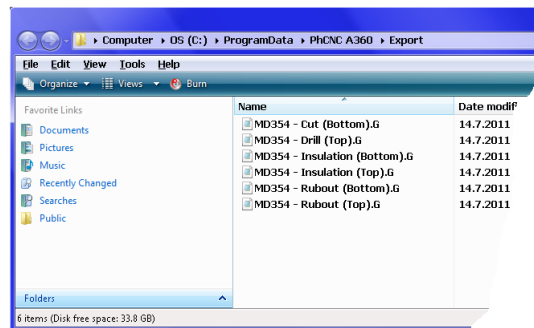
This is the name of the project and will be a prefix for all G programs generated.

Example:

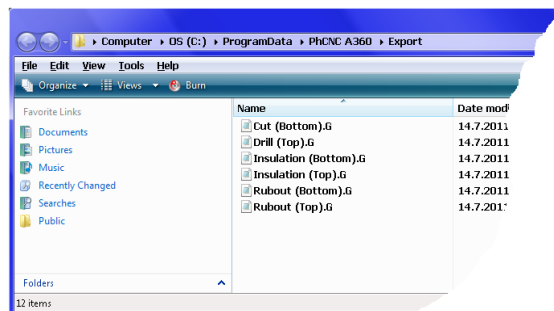
Project name: MD354,



Generated files are:



If there is no project name – generated files will have names related to the operations that they represent only.



If "Use Project Name" is selected, the project name will be the name of current project (if any). The files extension name is .GNC (by default) but can be changed if needed.

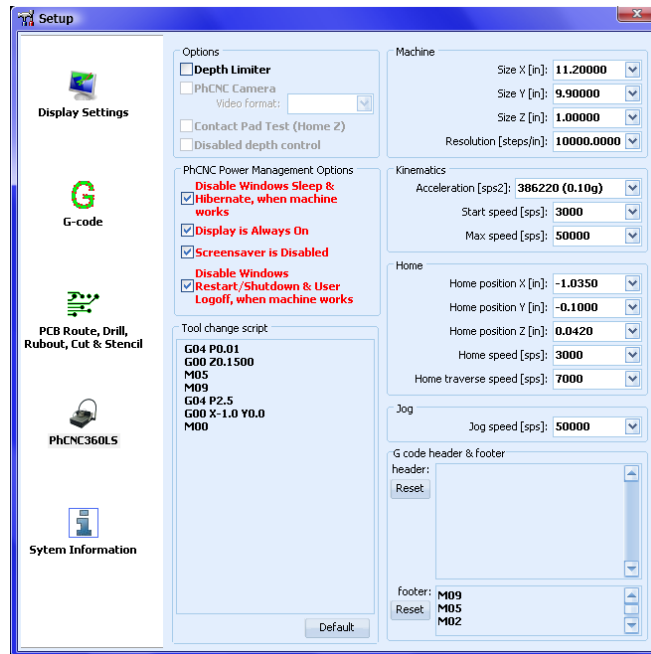
"Open destination folder"

If this option is selected, the folder that contains the G programs will be open (see examples above).

"CNC Simulator"

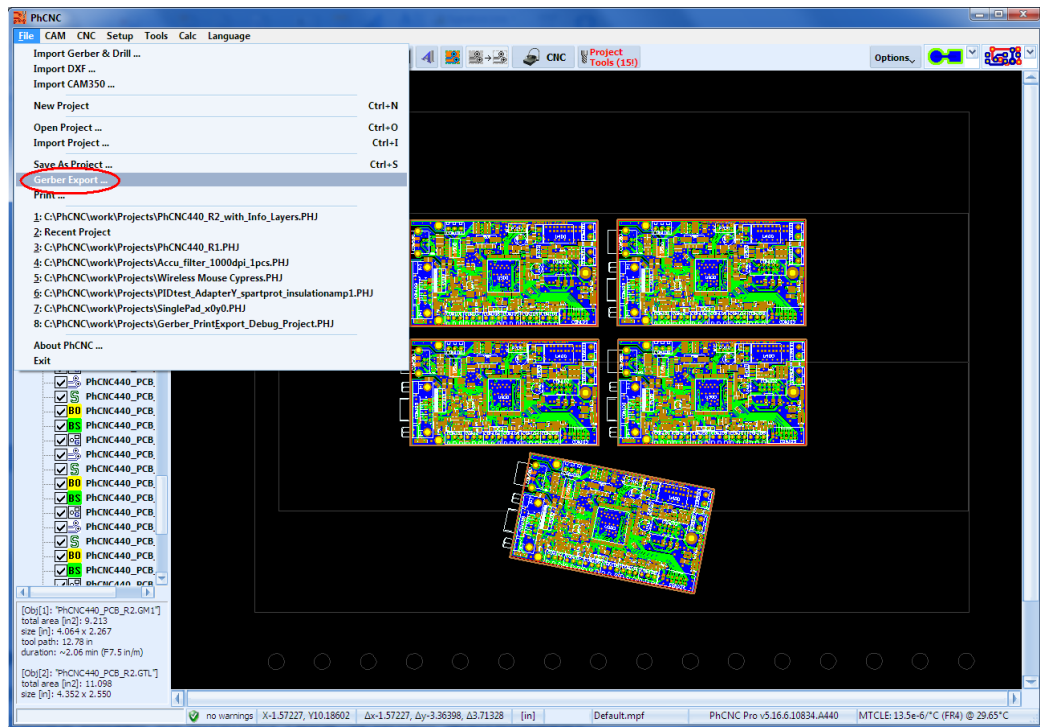
If this option is selected the system goes into CNC simulation mode after the G-Code programs are generated. In this mode all programs can be started.

All parameters of the simulated machine (dimensions, kinematics, etc.) can be set in the Setup menu.

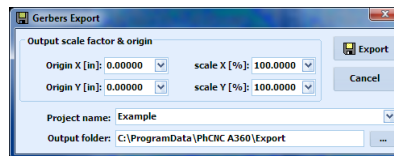


2.27 Gerber Export ...

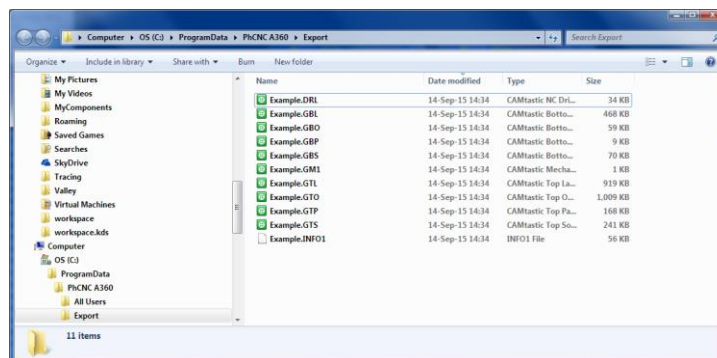
PhCNC LSSE allows an export of all Gerber and Excellon drill files in the current project . All visible files are exported including the info files that are not involved in the machining process.



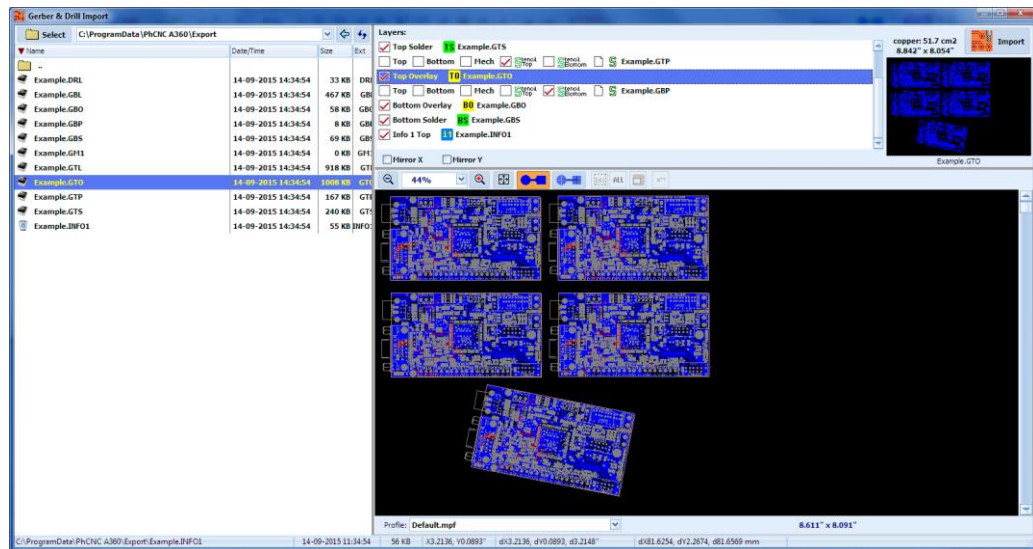
Gerber Export window allows to scale the files exported in Gerber and Excellon format (if needed).



After the export PhCNC LSSE automatically opens the folder that contain the exported files.



Exported Gerber and drill files can be imported back in PhCNC LSSE (for example after scaling on X/Y axes) or used in another CAD/CAM software.



Import & Export Gerbers & Info layers - video:

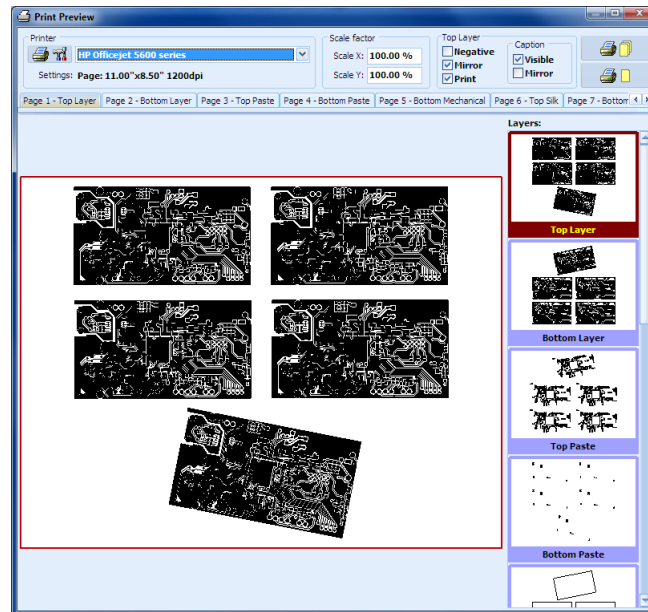
https://youtu.be/huY_sy04yi4

2.28 Print

PhCNC LSSE has an additional capability to print the layers of the project.

Print Preview window has the following options:

- printer selection and its settings
- scaling the print with individual scale factors for X and Y (common for all layers)
- individual settings for each layer
 - Positive/Negative
 - Mirror
 - Print - allows the layer to be printed
 - Caption - includes the name of the layer in the print
 - Mirror - prints the name in mirror



Layers' list includes all visible layers in the project. It is advisable to disable (hide) all layers that you don't have an intention to print. In the center window you will see the selected layer.

Using left double click on the layers name is the fastest way to enable/disable the layer for printing.

This button  serves to set the printer, page orientation and other printer specific settings.

Individual settings for each layer (Negative, Mirror, Caption, Caption mirror) and the printer used for PhCNC LSSE are saved in order to shorten the time needed in the next usage of Print option.

Button  sends all selected layers for printing and closes the Print Preview window.

Button  sends the current page only and leaves the Print Preview window open.

Note:

In the field for printer information, you may have a message "Invalid printer". This can happen in the cases when there are problems with the printer driver and its settings, missing DLL file or non-existing network printer or other problems related with the printer - PC connection.

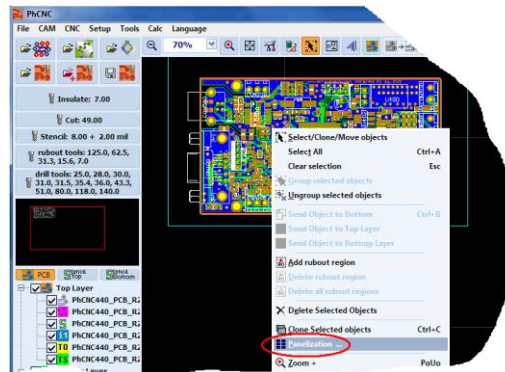
Print preview video:

https://youtu.be/zFNuUN_GkVo

2.29 Panelization

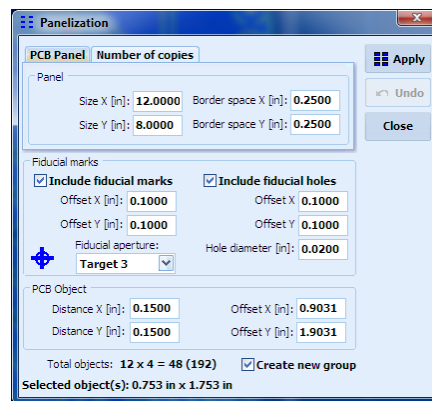
Panelization means placing multiple smaller objects/projects on a large work piece (panel) in order to be machined/manufactured together.

This PhCNC LSSE function allows fast and accurate multiplication of objects/projects or a group of them.



The first step is to select the object (or group of objects) using "Select/Clone/Move objects" from the popup menu.

Shown below is the Panelization window:



Panelization function has two modes of placement:

- PCB Panel
- Number of copies

2.29.1 "PCB Panel" mode

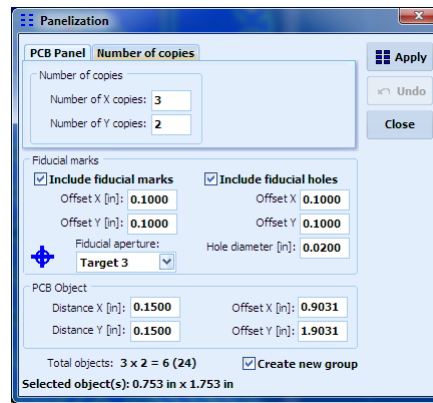
"PCB Panel" mode allows placing the object(s) in a panel with predefined size (Panel->Size X/Y) and on certain space from the physical edges of the panel (Panel->Border space X/Y). The number of copies is calculated based on:

- size of the object(s) (Selected objects(s))
- required distance between them (PCB Object->Distance X/Y)

The offset of the objects (PCB Object->Offset X/Y) is calculated automatically regarding the parameters above. It can be entered manually, but any changes in the parameters will return it to the automatically calculated values and new manual entry will be needed. The array of objects will be centered on the declared panel.

2.29.2 "Number of copies" mode

This is the easiest way to create the desired number of copies of your project entering the repeats on X and Y axes.

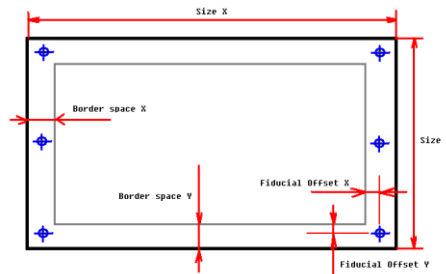


2.29.3 Fiducial marks

PhCNC LSSE allows inserting of targets & holes as fiducial marks around the panelization objects. Settings include: Offset (Fiducial marks->Offset X/Y); holes diameter (Fiducial marks->Hole diameter); aperture type (selected from the list).

The offset (Fiducial marks->Offset X/Y) depend of the mode selected.

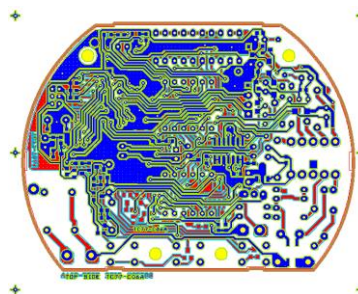
In "PCB Panel" mode, the offsets are defined in relation to Border space:



If negative values are used the marks will enter the machining space.

In "Number of copies" mode, the offsets of the fiducial marks are defined related to the area of placement of the objects. Again, if negative values are used the marks will enter the machining space. "Number of copies" mode allows easy adding of fiducial marks even on a single object. To do so the Number of copies X/Y should be set to 1.

Example:



In this case "Apply" button will not add additional copies, just will add fiducial marks.

2.29.4 Buttons "Apply" & "Undo" (Ctrl+Z)

Button "Apply", execute the placement of the projects/objects according to entered parameters and adds the fiducial marks.

Button "Undo" reverses the changes made after "Apply" and thus allows the operator to make changes in the parameters before next use of "Apply" with the corrected parameters.

2.29.5 Create new Group

This check box allows the result of panelization to be treated as a single object and moved without any changes in the relative position of the projects in the panel. This is useful function in many practical cases. If the individual object need to be moved operator can use "Ungroup selected objects" (popup menu).

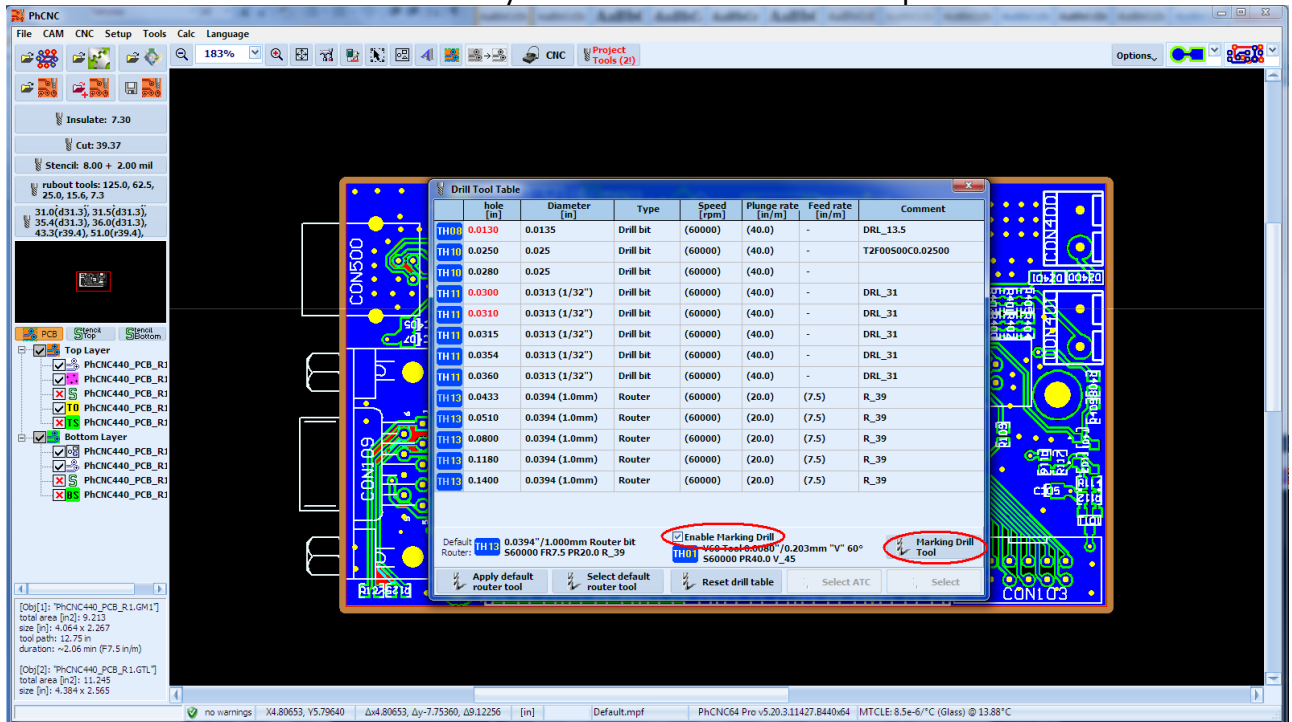
Panelization video:

<https://youtu.be/O2ziZLAYVwg>

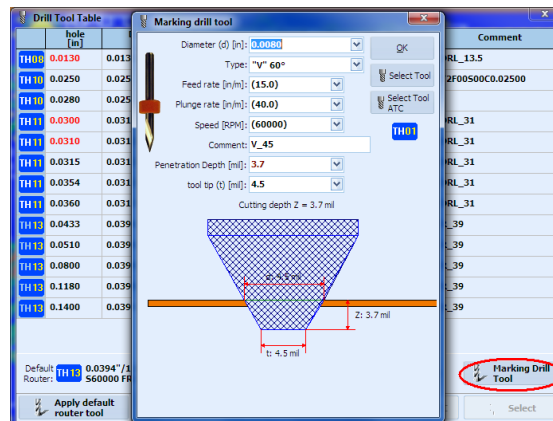
2.30 Marking Drill

PhCNC LSSE allows generation of "Marking Drill" program (some call it "Spot drill") that marks the centers of the holes to be drilled. The best tool for the marking process is a V type tool (V30, V45 or V60) small diameter stub end mill will do the job too.

In order to have "Marking drill" program generated you have to select "Enable Marking Drill" box as shown below. This also will allow you to select the tool for this operation.



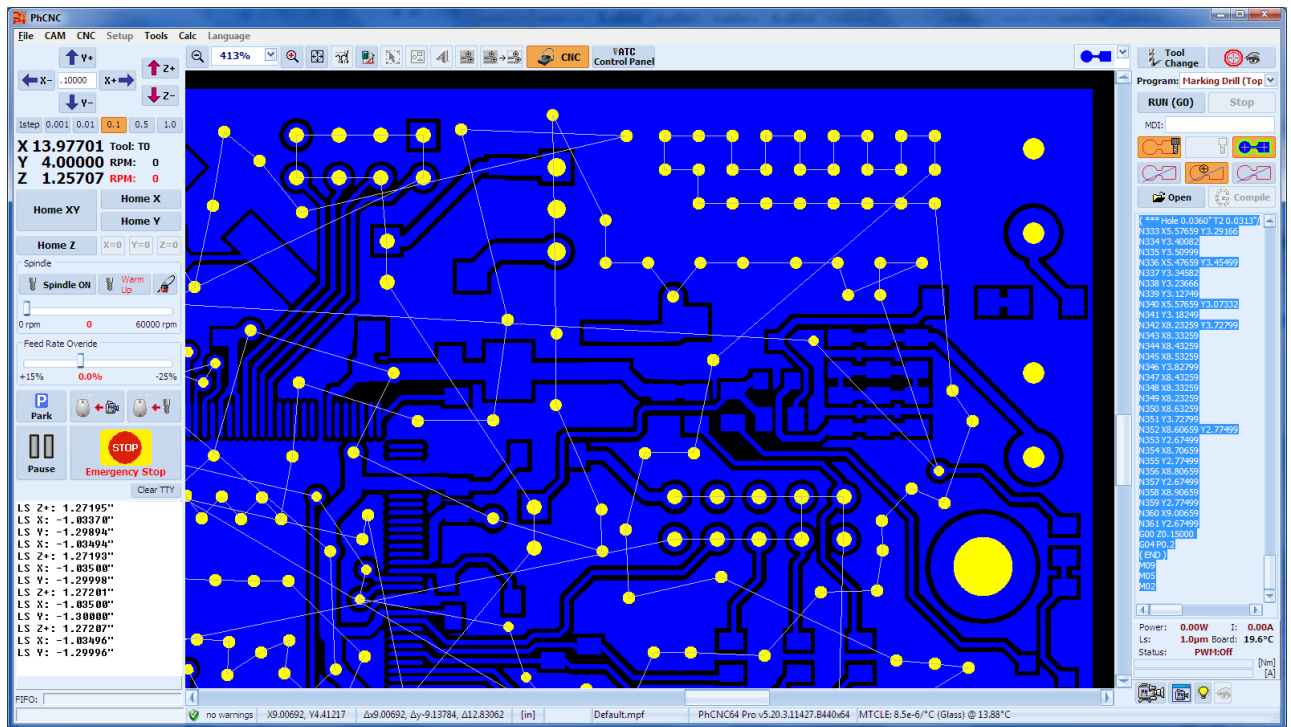
Marking Drill (Marking Drill Tool Setup):



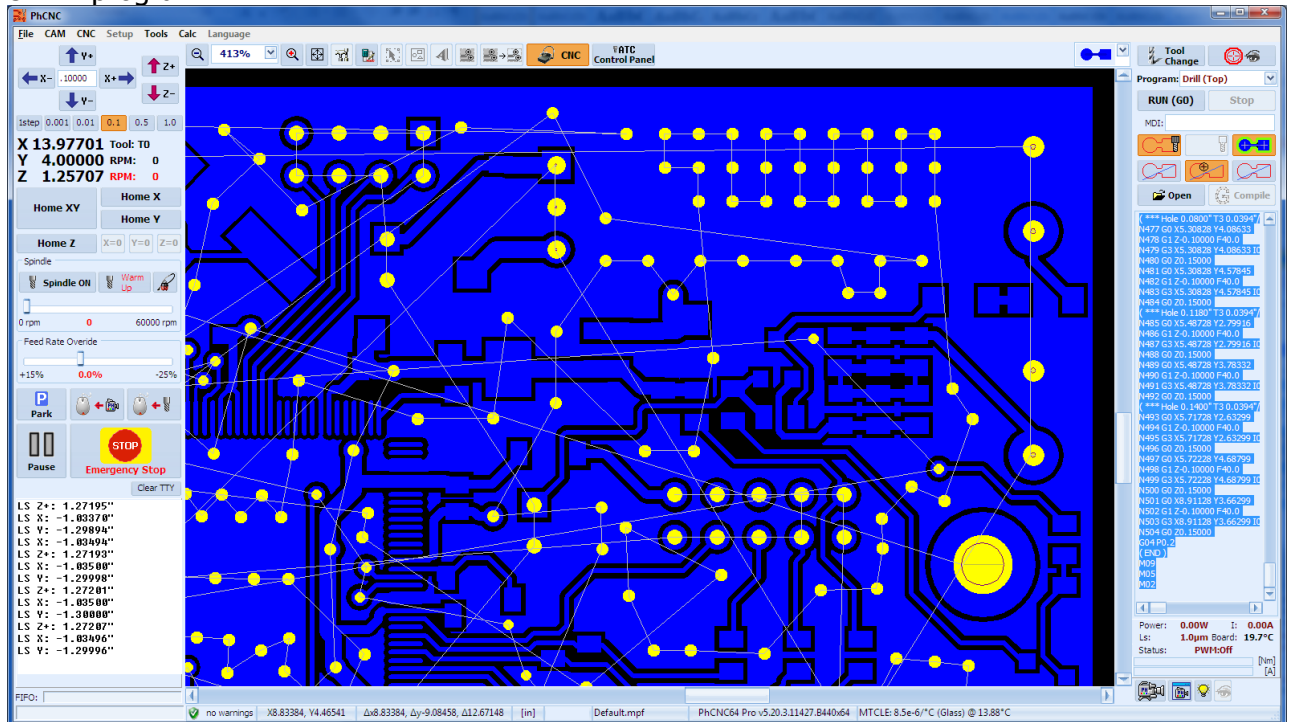
Notes:

The Marking Drill program will be applied only for the holes, for which drill bit will be used. The (usually larger) holes for which router bit is selected and the slot holes are excluded from the Marking Drill program.

In the example below, the larger holes will be cut with router bit, so they are excluded from the Marking Drill program:



The Drill program:



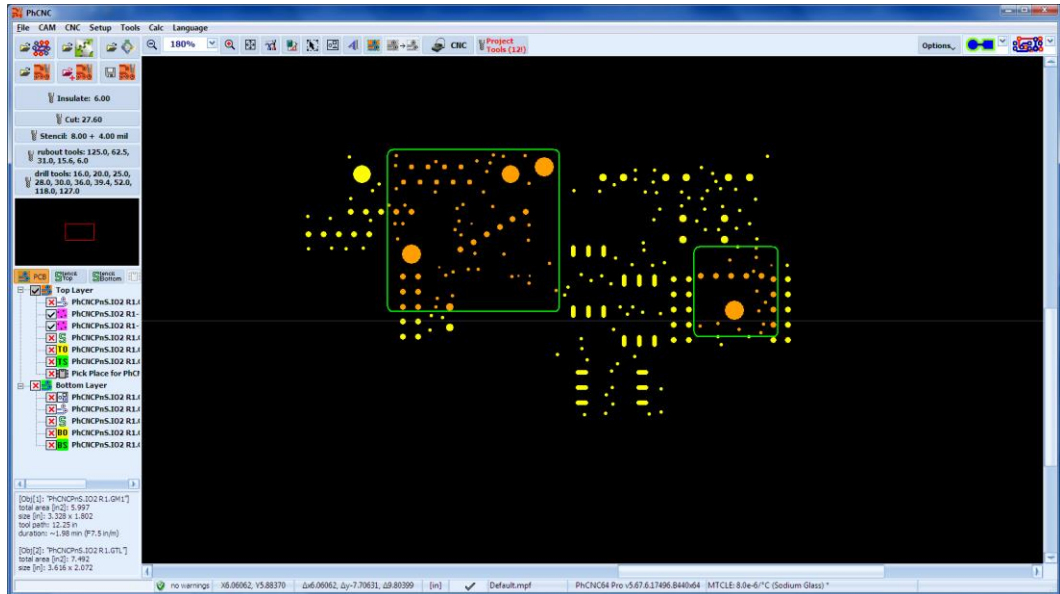
In the "Entire (Top)" & "Entire (Bottom)" ATC programs, the Marking Drill program will be executed before Drill program.

2.31 Holes Selection / Modifications / Removal

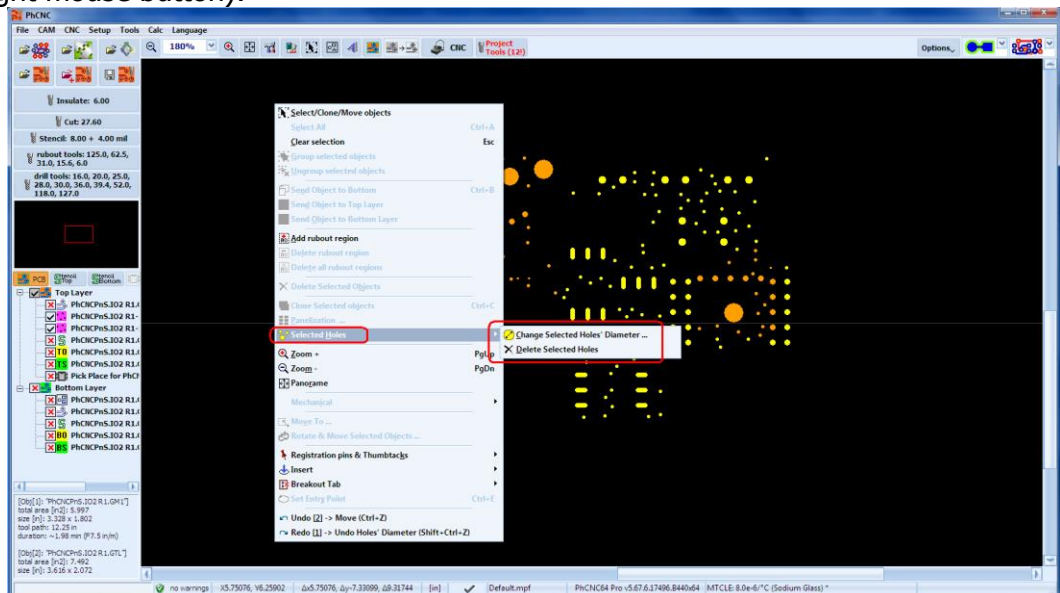
PhCNC allows to select holes from the project and modify/delete them.

For holes' selection, first press and hold "R" key. Then press and hold left mouse key and select the rectangular region. The holes inside this region will be selected (will change their color). You can select many regions. To remove the selection, you can use the "ESC" key or the option "Clear Selection" from the popup menu.

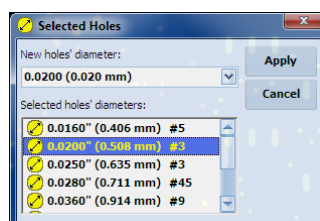
Example – on the screen below 2 rectangular regions are selected (the gerber files are hidden for clarity):



You can modify the diameter of the selected holes or delete them using the popup menu (when you click the right mouse button):



With the option "Change Selected Holes' Diameter ..." you can change the size of the diameter of any hole:



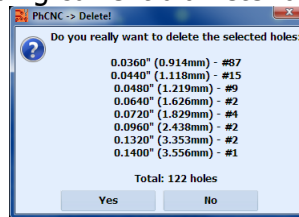
In the list you can find the holes' current diameters and the number of the selected holes with this diameter.

You can change the diameter for the holes with one and the same size simultaneously.

To delete selected holes, you can use the option "Delete Selected Holes" from the popup menu.

- If the holes removal is selected, PhCNC LSSE will require additional confirmation for the process to begin.

A list of selected holes will appear including current diameter and the number of them.



Notes:

- In case of mistakes related to the above two functionalities, you can use "Undo/Redo".
- If the project has multiple copies the changes will affect all of them, PhCNC LSSE does not allow the copies of the same project to be different.

Video #1 – holes manipulations:

<https://youtu.be/-U53MmXvnMA>

Video #2 - holes & regions selection:

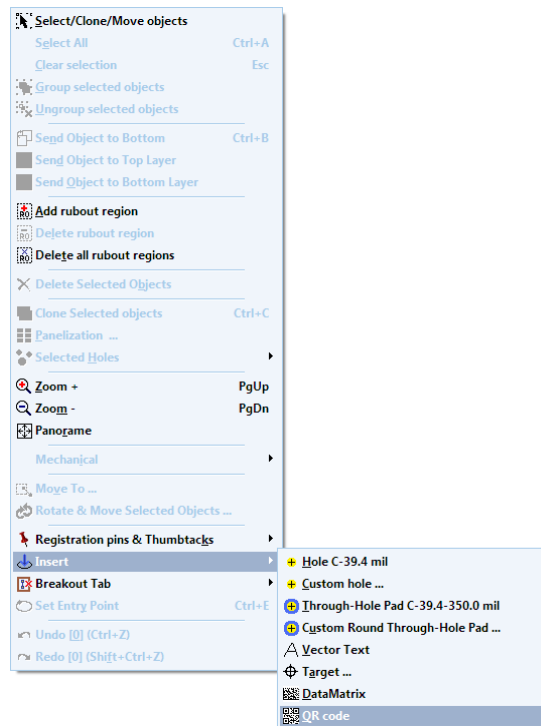
<https://youtu.be/j3JyJfE8SVU>

Video #3 - holes selection & panelization:

<https://youtu.be/nFny2D8eqBo>

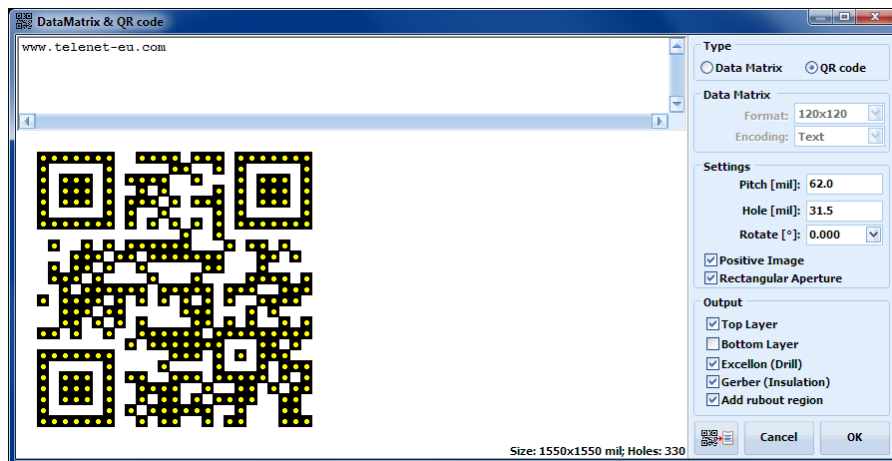
2.32 QR Code

PhCNC LSSE allows to insert QR Code at the mouse position from the Pop-up menu (right mouse click):



A lot of options are available depending on the methodology of manufacturing and the pitch size:

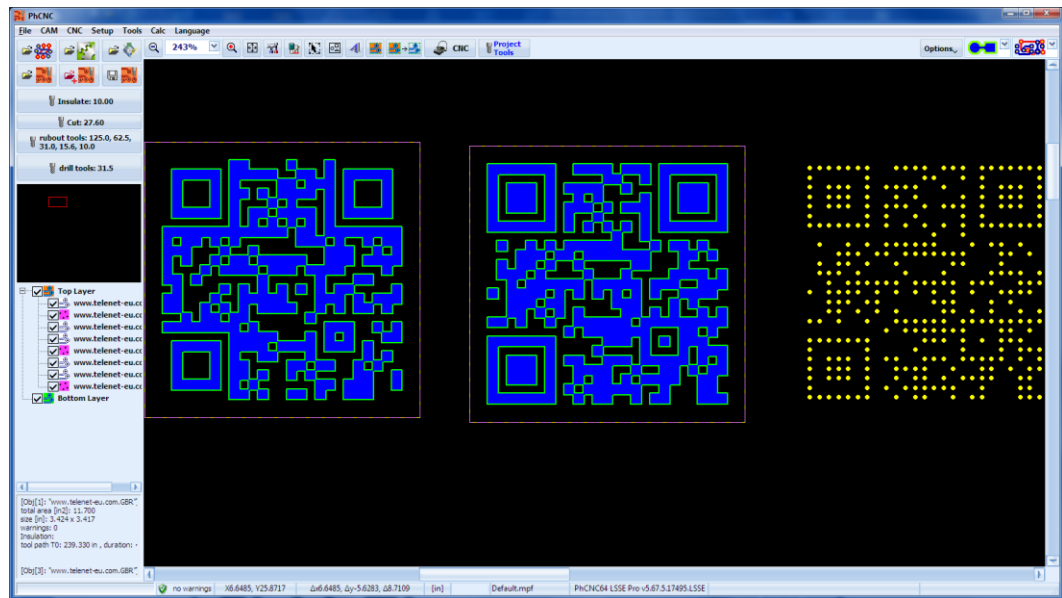
Positive image QR Code with Gerber & Drill:



Positive image QR Code with Gerber (engraving / insulation)

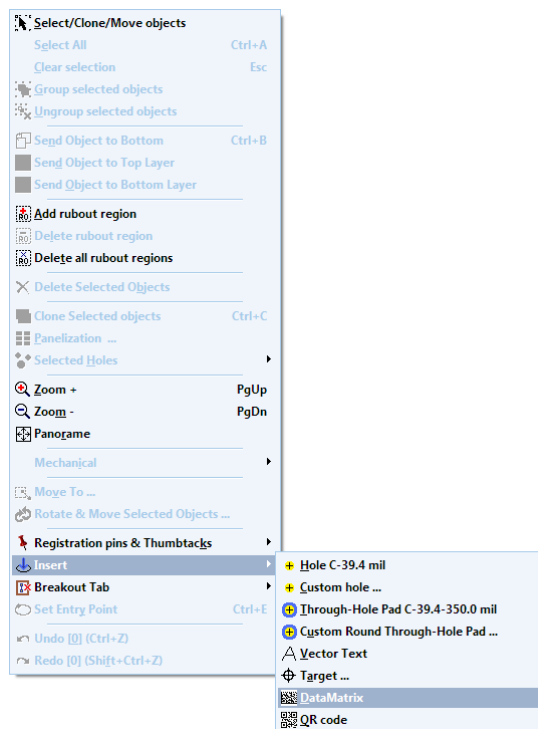


The result of several inserted QR Codes with negative, positive and drill only images:



2.33 DataMatrix

PhCNC LSSE allows to insert DataMatrix at the mouse position from the Pop-up menu (right mouse click):

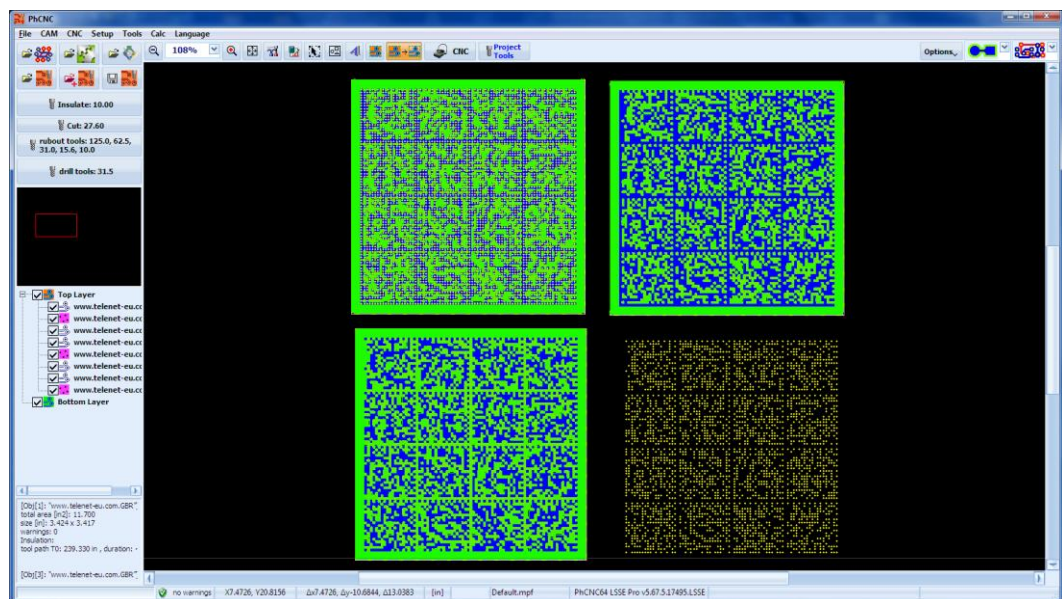


A lot of options are available depending on the methodology of manufacturing, the pitch size and the image size:

Positive DataMatrix with Gerber (engraving / insulation):



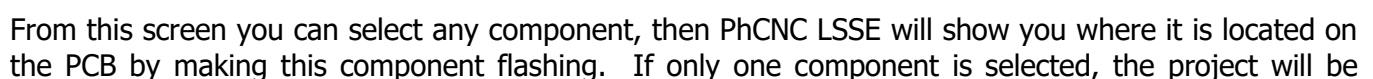
The result of several inserted DataMatrix with negative, positive and drill only images and with format 80x80:



2.34 Pick & Place Helper

PhCNC LSSE can import Pick & Place assembly files in CSV or IPC-D-355 formats (automatically detected by the software).

Here follows an example for project import with Pick & Place file:



moved on the screen and/or resized, so that the selected component becomes in the center. If several components are selected, the project will be moved on the screen and/or resized, so that all the selected components become visible.

Using Up or Down arrow you can select next/previous designator - it is very suitable for manual component placing.

PhCNC LSSE works in similar way when you have a panel:



From the Pick & Place window, you can select top or bottom layer. You can find component's position using its designator ...

video:

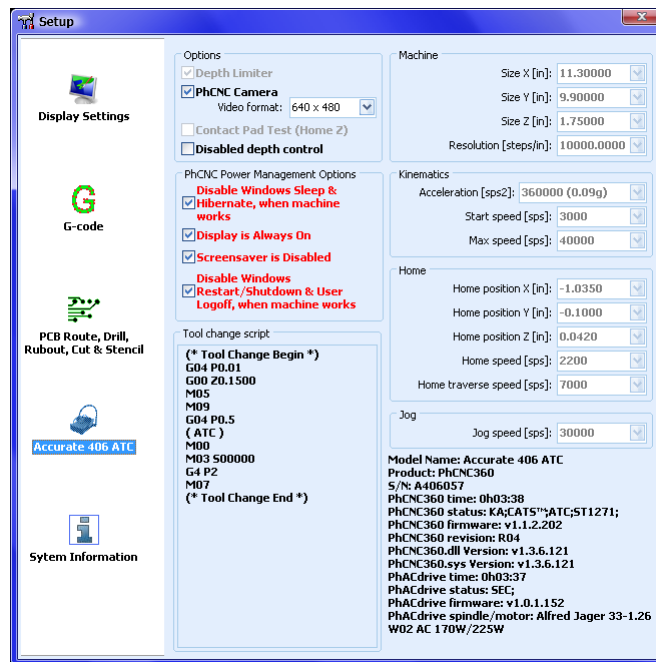
<https://youtu.be/Lm5CEI7WxWc>

3. Setup

All adjustments needed to run PhCNC LSSE are located in the window below



3.1 Depth Options Setup



Listed below are the settings for the depth limiter:

The "Depth Limiter" option changes the behavior of the machine for many of operations like "Insulation" and "Rubout". If you use "Peck" mode it also affects the "Cut" operation. This is due to the difference between the machines. The default setting for "Depth Limiter" is checked. Standard models use closed loop feedback to limit the depth and should have this option checked

3.2 CNC process control settings

The Setup window displays the following settings:

- Profile:** Default.mpf
- Rapid Height [in]:** 0.1500
- Default config:**
- PCB Route settings:**
 - Route Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck Route: ☐ step [in]: 0.0200
 - Spindle RPM [rpm]: 60000
 - Feed Rate [in/in]: 15.00
 - Plunge Rate [in/in]: 40.00
 - Tool Penetration Depth [in]: 0.0030
- PCB Rubout settings:**
 - Rubout Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck Rubout: ☐ step [in]: 0.0200
 - Spindle RPM [rpm]: 60000
 - Feed Rate [in/in]: 15.00
 - Plunge Rate [in/in]: 40.00
 - Tool Penetration Depth [in]: 0.0030
- PCB Drill settings:**
 - Drill Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck drill: ☐ step [in]: 0.0200
 - Peck drill mode: Safe Mode
 - G82 DWELL[sec]: none
 - Spindle RPM [rpm]: 60000
 - Plunge Rate [in/in]: 40.00
 - Drill on Layer: Top Layer
 - Tool Penetration Depth [in]: 0.0800
- PCB Cut settings:**
 - Cut Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck cut: ☐ step [in]: 0.0200
 - Spindle RPM [rpm]: 60000
 - Feed Rate [in/in]: 7.50
 - Plunge Rate [in/in]: 20.00
 - Tool Penetration Depth [in]: 0.0800
- Stencil settings:**
 - Stencil Cut Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck cut: ☐ step [in]: 0.0200
 - Stencil thickness [in]: 0.0060
 - Spindle RPM [rpm]: 60000
 - Feed Rate [in/in]: 7.50
 - Plunge Rate [in/in]: 20.00
 - Tool Penetration Depth [in]: 0.0100

Important:

The parameters included in this table are used in creating the G-Code programs. Therefore, changes made after you enter CNC mode will not be effective. In the latest revisions of PhCNC LSSE, the access of "Setup" window is disabled in CNC mode.

The Profile box is used to select the profiles you create to store your settings. You can create many different profiles for different materials and job conditions. This allows you to do fast and complete settings changes. The current profile is always stored in your .PHJ file, so if you have to repeat your project all tool and mode settings will be the same.

"Peck" options are used to machine nonferrous metals and some special plastics and laminates. They are also very useful in drilling holes with an extremely high aspect ratio. For example you can drill 12 mil holes in a 120 mil brass work-piece. The drill penetrates 15 mils deeper at every cycle and goes up out of the hole to clear the chips each time. It takes 8 cycles to do the hole, but this is a very high achievement for drilling small holes in metal.

"Tool Penetration Depth" this parameter represents the default value for each operation. It will be used if no other settings are made (as a tool setting or process/operation setting). These settings apply to all tools except the "V" type tools. Their default settings come from geometric calculations in the "Insulation tool" window (CAM mode).

Drill option G82 - Dwell sets the time (in seconds) that the drill bit will remain in the maximum penetration position before it is pulled up. By default this option is disabled.

The Setup window displays the following settings:

- Profile:** Default.mpf
- Rapid Height [in]:** 0.1500
- Default config:**
- PCB Route settings:**
 - Route Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck Route: ☐ step [in]: 0.0200
 - Spindle RPM [rpm]: 60000
 - Feed Rate [in/in]: 15.00
 - Plunge Rate [in/in]: 40.00
 - Tool Penetration Depth [in]: 0.0030
- PCB Rubout settings:**
 - Rubout Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck Rubout: ☐ step [in]: 0.0200
 - Spindle RPM [rpm]: 60000
 - Feed Rate [in/in]: 15.00
 - Plunge Rate [in/in]: 40.00
 - Tool Penetration Depth [in]: 0.0030
- PCB Drill settings:**
 - Drill Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck drill: ☐ step [in]: 0.0200
 - Peck drill mode: Safe Mode
 - G82 DWELL[sec]: none
 - Spindle RPM [rpm]: 60000
 - Plunge Rate [in/in]: 40.00
 - Drill on Layer: Top Layer
 - Tool Penetration Depth [in]: 0.0800
- PCB Cut settings:**
 - Cut Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck cut: ☐ step [in]: 0.0200
 - Spindle RPM [rpm]: 60000
 - Feed Rate [in/in]: 7.50
 - Plunge Rate [in/in]: 20.00
 - Tool Penetration Depth [in]: 0.0800
- Stencil settings:**
 - Stencil Cut Depth [in]: 0.1000
 - Rapid Height [in]: 0.1500
 - Peck cut: ☐ step [in]: 0.0200
 - Stencil thickness [in]: 0.0060
 - Spindle RPM [rpm]: 60000
 - Feed Rate [in/in]: 7.50
 - Plunge Rate [in/in]: 20.00
 - Tool Penetration Depth [in]: 0.0100

If this option is selected it applies only for the use of drill bit. If a router bit is selected it is disabled automatically.

Important:

Parameters set in the CNC process control settings are used as “default” for the tools. If you have any particular setting for any tool in the tool tables it will override the setting in this table.

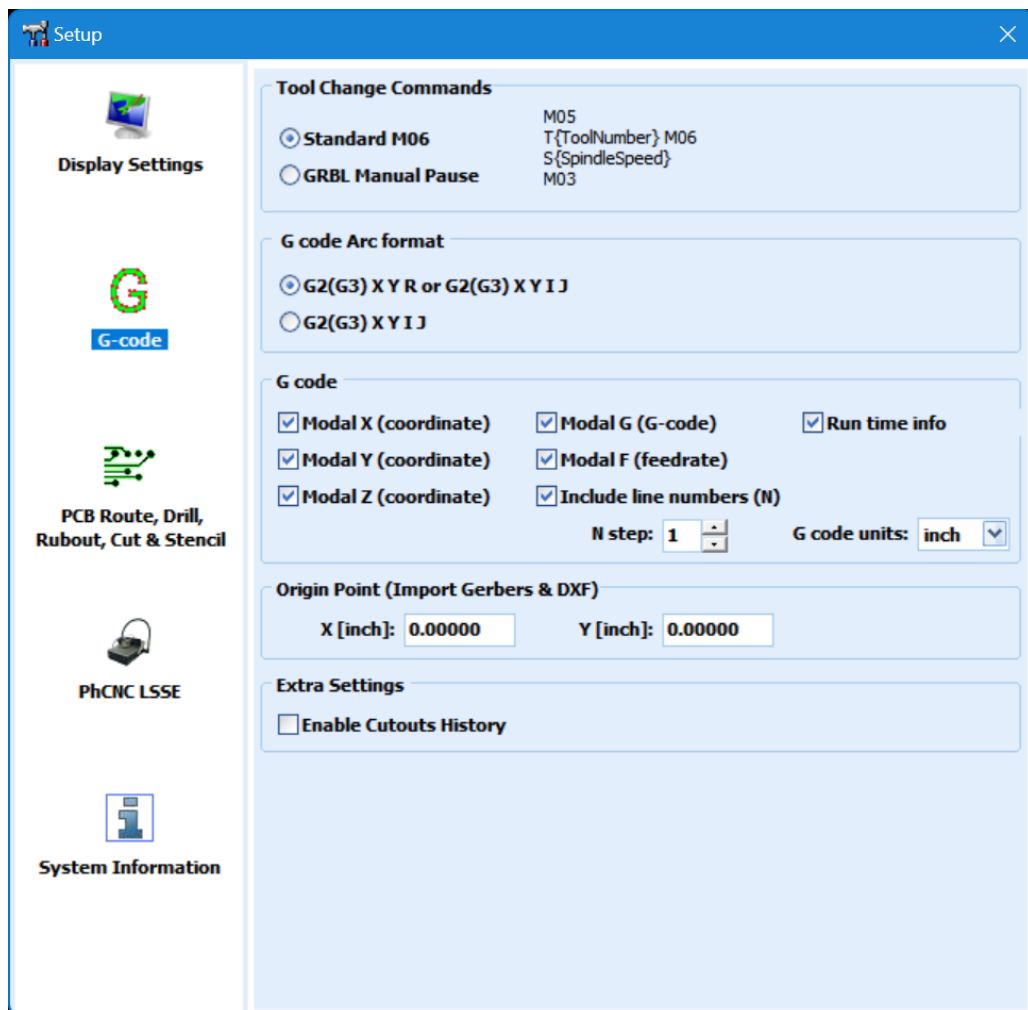
3.3 G-Code post-processor and machine compatibility settings

PhCNC LSSE can generate G-Code for a wide range of CNC controllers. Before exporting G-Code for a new machine, the post-processor settings should be checked and adjusted to match the requirements of the target CNC controller.

These settings affect the syntax and behavior of the generated G-Code. Incorrect settings may result in a program that cannot be interpreted correctly by the target CNC controller.

Once a suitable configuration has been established for a machine, these settings normally do not need to be changed.

Important: The default settings are suitable for most standard CNC controllers and should normally be left unchanged. Change the post-processor settings only when required for compatibility with the target CNC controller. After changing these settings, inspect and simulate the generated G-Code before running it on a machine.



3.3.1 Tool change commands

The Tool Change Commands setting determines how PhCNC LSSE generates tool-change operations.

Standard M06

This mode generates standard Tn M06 tool-change commands. It is suitable for CNC controllers that support the standard M06 command, including Mach3 and LinuxCNC. Depending on the machine configuration, M06 may request a manual tool change or operate an automatic tool changer.

A typical tool-change sequence generated by PhCNC LSSE is:

```
G0 Z0.15000
M05
T3 M06
S60000
M03
```

PhCNC LSSE moves the Z axis to a safe position and stops the spindle before requesting the tool change.

GRBL Manual Pause

This mode is intended primarily for classic GRBL controllers, where the standard M06 tool-change command is not implemented. Instead of requesting an M06 tool change, PhCNC LSSE generates an M00 program pause so the operator can change the tool manually and then resume execution.

This mode can also be useful with other simple CNC controllers that support program pause but do not provide a standard M06 tool-change mechanism.

3.3.2 Arc format

PhCNC LSSE supports the two commonly used methods of describing circular interpolation with G02 and G03.

G2(G3) X Y R or G2(G3) X Y I J

This setting allows arcs to be generated using either the radius parameter R or the arc-center offsets I and J, as appropriate.

G2(G3) X Y I J

This setting forces circular interpolation to be generated using I and J center offsets.

Example:

```
G02 X1.2500 Y0.7500 I0.1250 J0.0000
```

The appropriate setting depends on the CNC controller. If there is any uncertainty, the I/J format is generally preferable because it explicitly defines the arc center.

PhCNC LSSE divides circular motion into quadrant-safe arc segments where required. This improves compatibility with controllers that impose restrictions on arcs crossing quadrant boundaries.

Important: The default G-Code header contains G91.1, which specifies incremental I/J arc-center coordinates. If the header is modified, make sure that the arc-center mode expected by the CNC controller remains consistent with the generated G-Code.

3.3.3 Modal G-Code output

PhCNC LSSE can generate coordinates and other G-Code words in modal form. The available modal selections include X, Y and Z coordinates, the G command and feed rate F.

When a parameter is modal, PhCNC LSSE writes it only when its value changes.

For example:

```
G01 X1.0000 Y1.0000 F10.0
X1.5000
X2.0000
Y1.5000
```

is equivalent to:

```
G01 X1.0000 Y1.0000 F10.0
G01 X1.5000 Y1.0000 F10.0
G01 X2.0000 Y1.0000 F10.0
G01 X2.0000 Y1.5000 F10.0
```

Important: It is strongly recommended to keep all available modal options enabled. Modal output produces significantly smaller G-Code files and makes the programs easier for a human operator to read and inspect. Disable modal output only if it is required for compatibility with a particular CNC controller.

3.3.4 G-Code units and line numbering

G-Code units

The G-Code units setting determines the units used in the generated G-Code:

- inch - generates G20;
- mm - generates G21.

Normally, this setting automatically follows the measurement units currently selected by the user in CAM mode. If the user works in millimeters, G21 is selected when the G-Code setup window is opened. If the user works in inches, G20 is selected.

The user can override this selection temporarily. For example, while working in millimeters in CAM mode, the G-Code output can be changed manually from G21 to G20. This override does not change the CAM measurement-unit setting.

The next time the G-Code setup window is opened, the G-Code units are initialized again from the current CAM units:

```
CAM mode = mm    -> default G-Code units = G21
CAM mode = inch  -> default G-Code units = G20
```

This behavior allows the G-Code output units to be changed for an individual case without changing the user's normal CAM working units.

Line numbers (N)

When Include line numbers (N) is enabled, generated G-Code blocks are prefixed with sequence numbers. The N step parameter defines the increment between consecutive line numbers.

Example with N step = 10:

```
N10 ...
N20 ...
N30 ...
```

Line numbers are not required by most modern controllers, but may improve readability and may be required by some CNC systems.

3.3.5 G-Code coordinate offset

The X and Y fields in the Origin Point (Import Gerbers & DXF) group define a coordinate offset that is added to all X and Y coordinates when the G-Code programs are generated.

For example, with:

```
X offset = 10.0 mm
Y offset = 20.0 mm
```

a point that would normally be generated as:

```
X5.0 Y8.0
```

will be generated as:

```
X15.0 Y28.0
```

The geometry of the PhCNC LSSE project itself is not modified. The offset is applied only to the coordinates written to the generated G-Code programs.

This option can be useful when the coordinate origin expected by the target CNC machine differs from the coordinate origin used when preparing the project in PhCNC LSSE.

The X/Y offset is stored in the PhCNC LSSE INI configuration and remains active until it is changed or reset by the user.

Important: Normally both offsets should be set to zero. A non-zero offset affects all generated machining programs and should therefore be used intentionally.

3.3.6 CNC simulator options

Some options located on the G-Code setup page do not modify the generated G-Code. They control only the behavior of the integrated CNC simulator.

Run time info

When enabled, selecting a G-Code program in the CNC simulator displays a small information window with the estimated program execution time. The estimate is calculated using the current simulator settings, including machine acceleration and maximum movement speed.

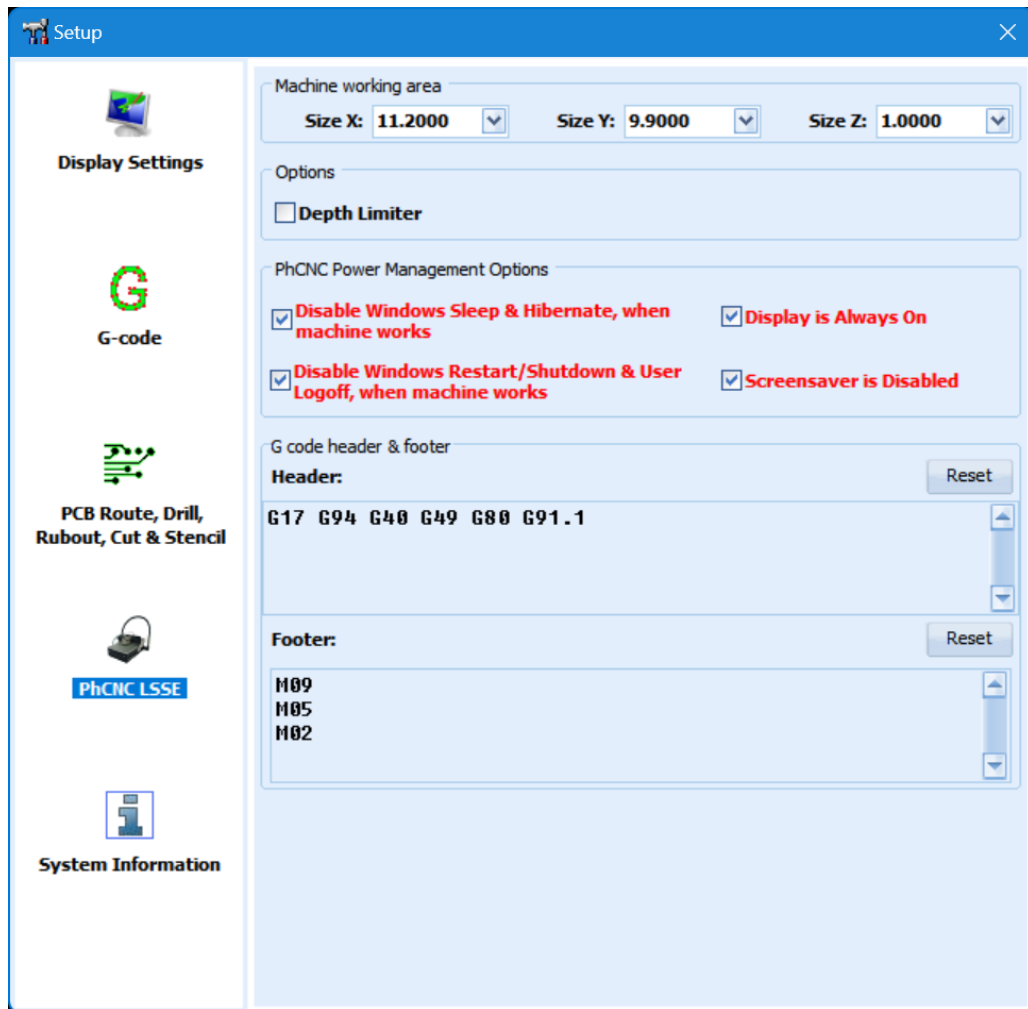
Enable Cutouts History

When this option is enabled, executing a board cutout program in the CNC simulator adds the resulting cutout contour to the cutout history. Previously machined contours are displayed as dashed outlines on the machine working area.

This is particularly useful when a partially used workpiece remains on the CNC machine. The operator can see the areas already used by previous jobs and position new PCB projects in the remaining free areas. This can reduce material waste and setup time.

3.3.7 G-Code header and footer

PhCNC LSSE allows user-defined G-Code commands to be inserted automatically at the beginning and at the end of every generated G-Code program. These fields are located in Setup -> PhCNC LSSE.



Header

The Header is inserted before the machining commands of every generated program. The default header is:

```
G17 G94 G40 G49 G80 G91.1
```

The default commands place the CNC controller into a known initial state before machining begins:

- G17 - XY working plane;
- G94 - feed rate in units per minute;
- G40 - cancel cutter-radius compensation;
- G49 - cancel tool-length compensation;
- G80 - cancel active canned cycles;
- G91.1 - incremental I/J arc-center coordinates.

Footer

The Footer is appended after the machining commands. The default footer is:

```
M09
M05
M02
```

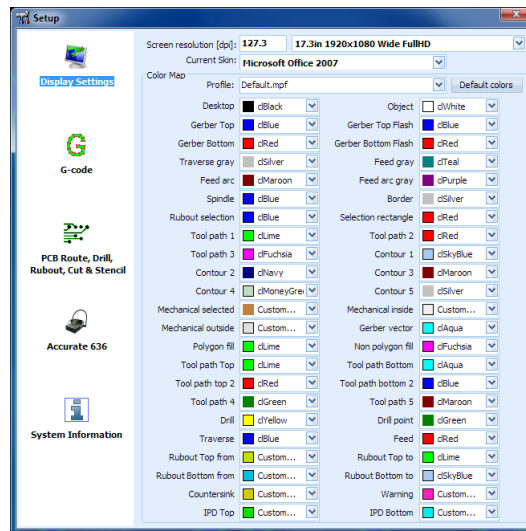
These commands turn off auxiliary/coolant output where applicable, stop the spindle and terminate the program.

Important: For most CNC controllers, the default Header and Footer should be suitable and should not be modified unless required by the target controller. The Reset buttons restore the default Header or Footer.

3.4 Display settings

A proper setting of screen resolution will allow you to have the actual size of objects displayed when the zoom is at 100%.

PhCNC LSSE has 7 built in skins, you are able to change them in this window.

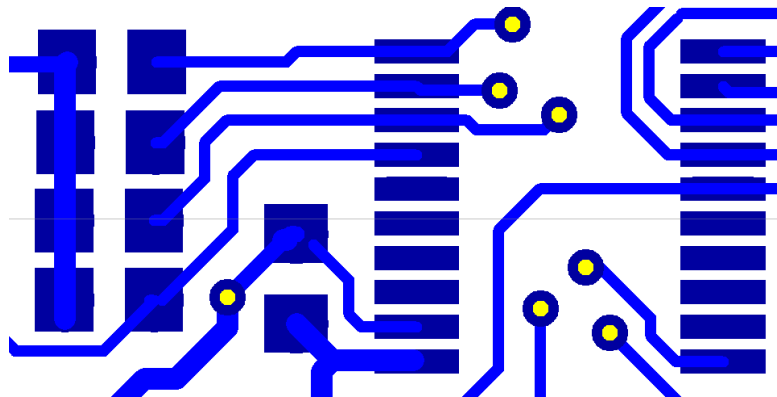
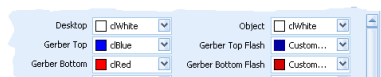


You can change the colors for drawing of the objects on the screen, as well as create different color schemes called profiles. You can select a profile by entering the name of the profile in the profile box or selecting it from the drop-down list.

Example:

If you want flash apertures to be in a different color:

- Select a different color from the "Gerber Top Flash" drop down list that is different than the "Gerber Top" color.
- Select a different color from the "Gerber Bottom Flash" drop down list that is different than the "Gerber Bottom" color.



Important:

In this manual white is the color we used for the Desktop & Background of screen shots for the purpose of saving ink/toner when it is printed. By default, the color of Desktop & Background is black in the actual program.

3.5 Multithreading & PhCNC LSSE64

PhCNC LSSE does extraordinary volume of calculations during the determination of tool paths in Insulation, Rubout, Remove spikes (IPD), Forced Insulation, ...

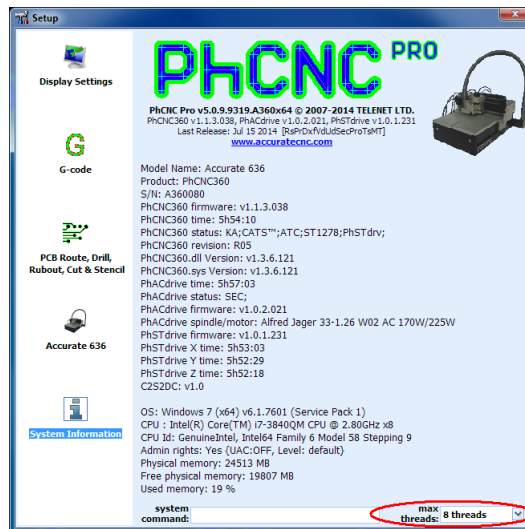
Beginning v4.4+, all calculations are optimised for multithread execution. This creates conditions to reduce calculation duration by 2 to 8 times (depend of the hardware in use).

When a parallel processing is in use, **PhCNC LSSE** will need larger size of RAM. This problem does not exist in **PhCNC64 LSSE** - the 64 bit version of **PhCNC LSSE**. This version is designed to run under **x64** OS (Vista/Win7/8/8.1) and is capable to address more than 4GB of RAM.

PhCNC64 LSSE is fully identical to **PhCNC LSSE**. The 64 bit version simply is capable of full utilization of the capabilities of the new x64 OSs.

PhCNC64 LSSE and **PhCNC LSSE** can be installed on the same computer, but only one can be run at a time.

There is an option for manual setting of the maximum number of parallel threads in the Setup Window.



By default the maximum number of parallel threads is equal to the number of logical cores in the CPU(s) of the computer in use.

PhCNC LSSE will utilize the maximum number of parallel threads when it is possible or needed.

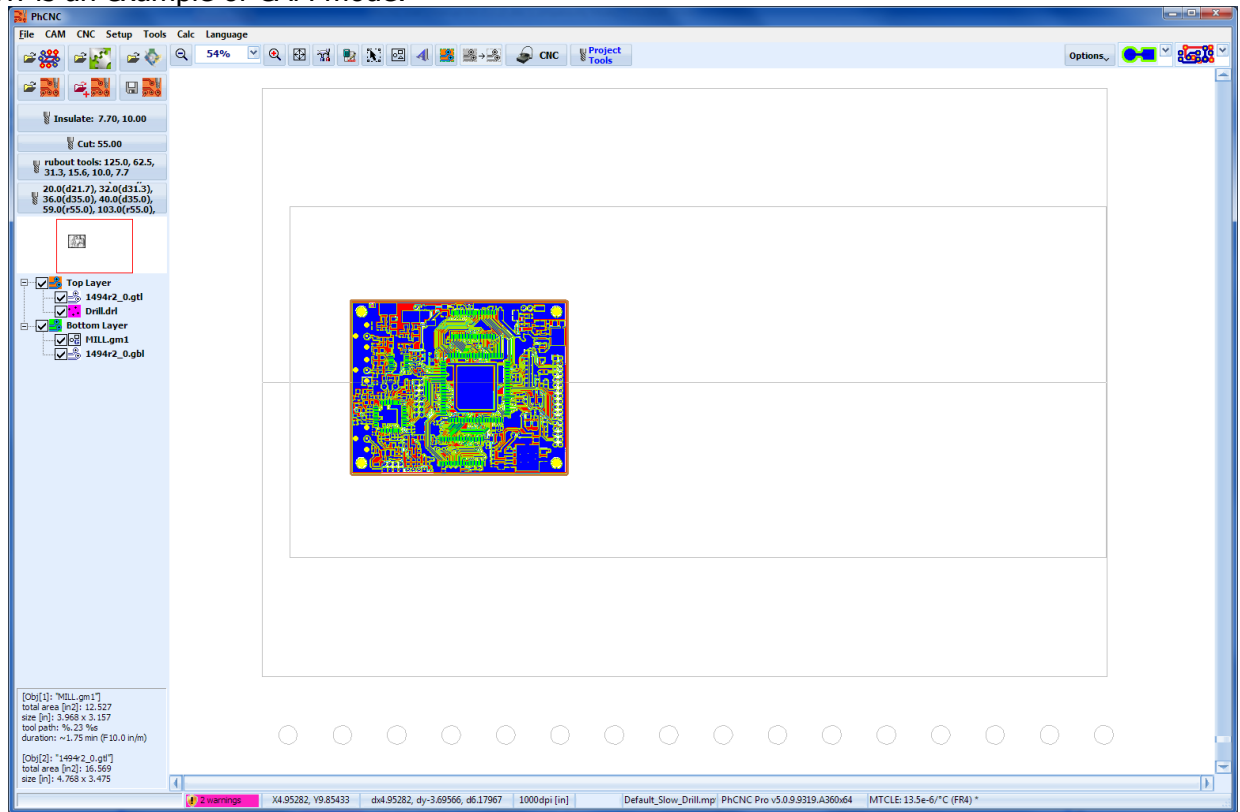
4. CNC mode

In CNC mode you will be able to produce the boards with your design, as edited in CAM mode. To enter CNC mode, click on the  CNC button.

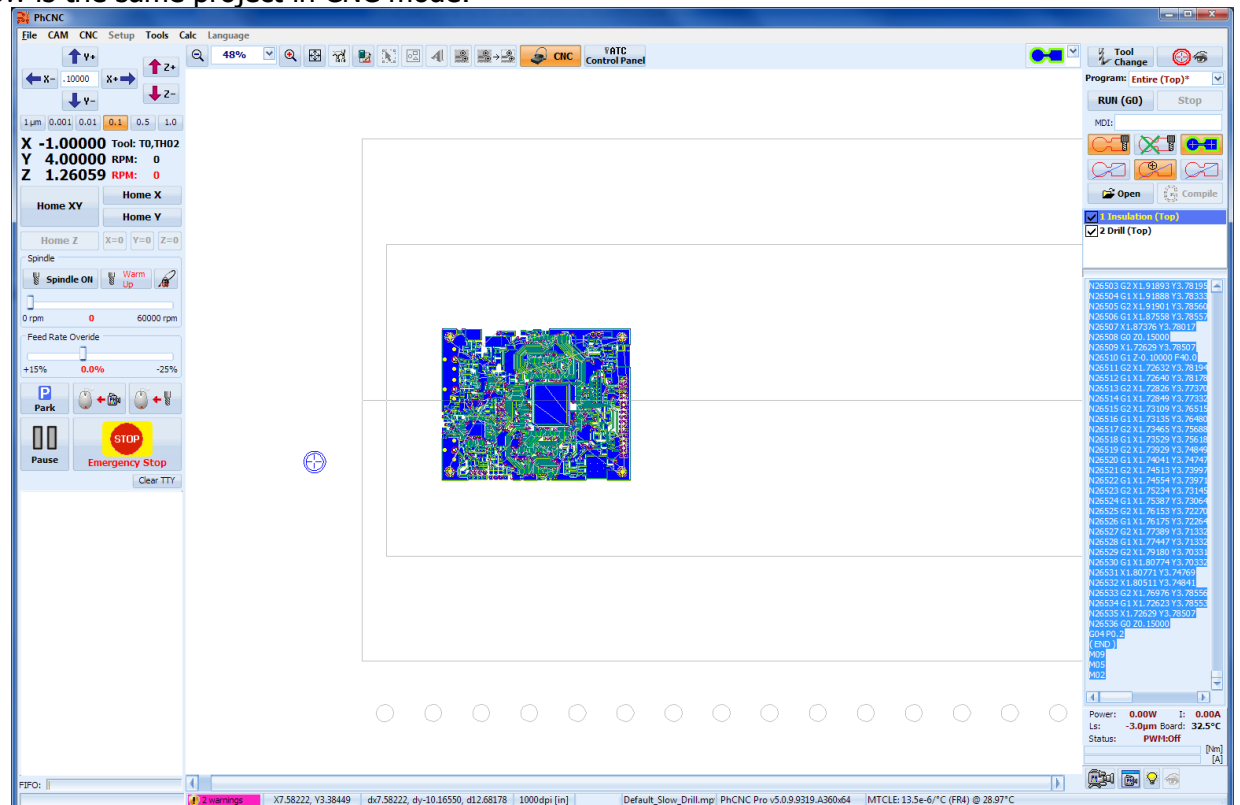
Important:

PhCNC LSSE will reject your request to enter the CNC mode if a machine is not connected to the computer or not powered up.

Below is an example of CAM mode:



Below is the same project in CNC mode:

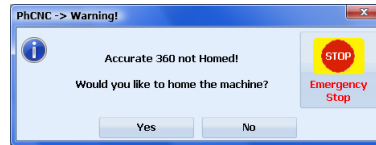


On the left side of your screen are all the manual controls, and on the right side are all the program execution controls.

Every time PhCNC LSSE is started, the first transition to CNC mode will put the machine in the Emergency Stop condition.

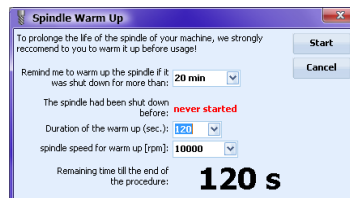
To proceed further you must click on the  button.

The next window will ask you to Home the machine. The Initial Home procedure is **essential and must be executed**. To continue left click on the yes button.



Now the machine will move the Z axis first until it reaches the upper limit switch then it will move the X & Y axes until they reach their home switches.

The next window will prompt you to warm up the spindle. PhCNC LSSE keeps track of spindle usage and will issue this prompt depending on the settings specified in this window. It is recommended to accept and push the Start button. This will prolong the life of your spindle.

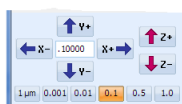


After this procedure the window above will close automatically.

If you select Cancel, this prompt will appear again when you start any of the G-Code programs.

4.1 Manual controls

4.1.1 Spindle/camera positioning using Jog buttons or mouse



Pressing the arrow buttons shown below allows movements of all three axes using the specified step shown in the center box. One step is executed with every mouse click. If you hold down any of the buttons for more than 1 sec., the respective axis will move constantly until you release the button.

Another convenient option to move the machine head to certain coordinates is to use the "Chase Mouse (Spindle mode)" by pressing the  toggle button (alternative action button- ON/OFF). Point the mouse to the desired position and right-click. The machine will position to this point using its traverse speed.

Important note:

In order to improve operator safety "Chase Mouse" functions ("Spindle" or "Camera" mode) will require the "Ctrl" button on the keyboard to be activated (pushed) during the command. When you select "Chase Mouse (Spindle mode)" and push the control key, the cursor will look like this . Right clicking with the mouse will bring the spindle to the position of the cursor. Using the status bar (described earlier in this manual) and the powerful zoom (up to 20000%) you may position your work with the accuracy of 0.000039 inch (1µm).

The information fields below show current spindle coordinates for all three axes, current tool, spindle speed in RPM setting and the actual RPM.



4.1.2 Home X & Y

The buttons , , and  are for homing the X & Y axes.

4.1.3 Home Z

This button starts the "Home Z" process. To execute Home Z, you have to place (manually) the milling head over your work piece and click on "Home Z". During the process the milling head will lower until it touches the material and will go back to calibration station. It will touch the calibration station surface (precisely known height).

Then it will update your "stack"* height. After successful execution the PhCNC LSSE will record your new "stack" height. In every future tool tip calibration PhCNC LSSE will correct the Z coordinates so that the tool tip will touch the material at Z=0.

* "stack" means - the thickness of your backing material + the thickness of your work piece.

Important:

You will need to repeat "Home Z" procedure only if you change the thickness of the stack e.g. change the thickness of backing material or the thickness of your board material (work piece). The determination of the surface of material is designed to allow the machine to work in proper conditions but is not related to tool penetration accuracy.

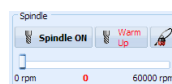
PhCNC LSSE v5.0 and above executes the Home Z procedure automatically at first start of the program using "Run G0" button. In this case the stack height are calculated based on first coordinate due for machining in your project.

4.1.4 "X=0", "Y=0", "Z=0" Buttons

Buttons , , and  allow you to move the coordinate system origin to the current spindle position. You may use them separately or in any combination. The usage is not recommended because it will change your tool change position. They are designed for tests and maintenance functions. Normally they are disabled.

4.1.5 Spindle & Vacuum control

The block below is for manual spindle and vacuum control:



During the execution of a G-Code program, it takes control of the spindle and vacuum.

Button  (alternative action button - ON/OFF) turns the spindle motor on and off. You may adjust the spindle RPM using the slide bar located in this block.

Button  starts the Spindle warm up procedure.

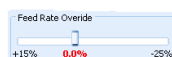
Important:

Spindle must not be started without a tool mounted in the collet.

 Button manually turns on and off your vacuum system. If a G-Code program is running it takes control of the vacuum system.

4.1.6 Feed Rate Override

The slide bar shown below can change the feed rate from +15% to -25% during G-Code execution.



4.1.7 Emergency Stop



This button stops the machine immediately and unconditionally.

Important:

Homing procedure is needed after using this button!

The machines maintain the coordinates after Emergency Stop.

The Homing procedure will be offered as a prompt, but you may skip it.

4.1.8 Pause



This button is Pause/Continue. It stops and resumes the machine operations. If you push pause the machine will not execute any commands - manual or program controlled, until you push continue.

4.1.9 Park



This button moves the machine head to the corner opposite to the machine zero. It is designed to provide access to the work piece and the tool magazine in ATC models.

4.2 Program execution in CNC mode

When you enter CNC mode, PhCNC LSSE generates all necessary programs to produce your board(s), based on data entered in CAM mode.

In the field "Program" (shown below) are listed all programs. Based on the last example we have:

- "Insulation (Top Layer)"
- "Drill (Top Layer)"
- "Rubout (Top Layer)"
- "Insulation (Bottom Layer)"
- "Rubout (Bottom Layer)"
- "Cut (Bottom Layer)"

The first 3 programs must be executed on the top layer and the last 3 on the bottom Layer, i.e. after you flip the board. PhCNC LSSE has generated these 6 programs:

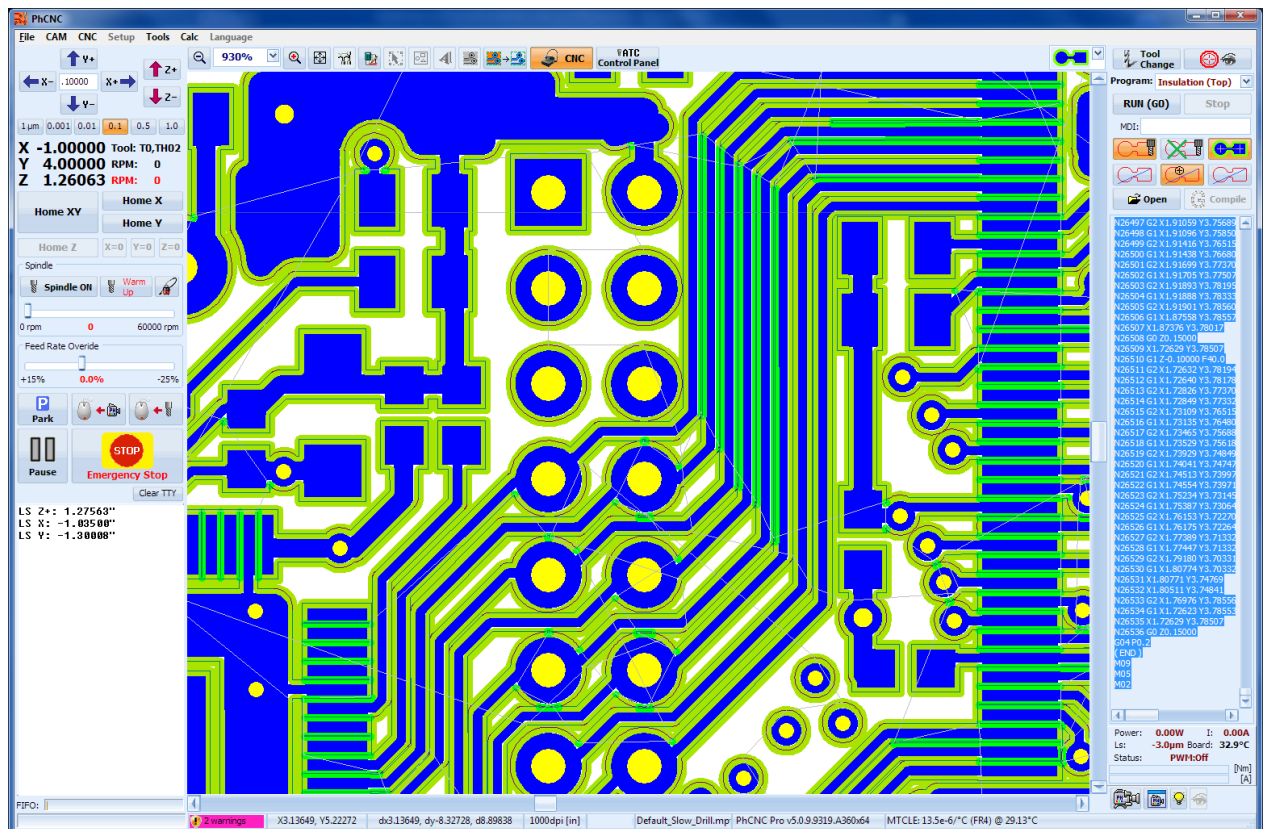


The operator sets the order of execution as desired.

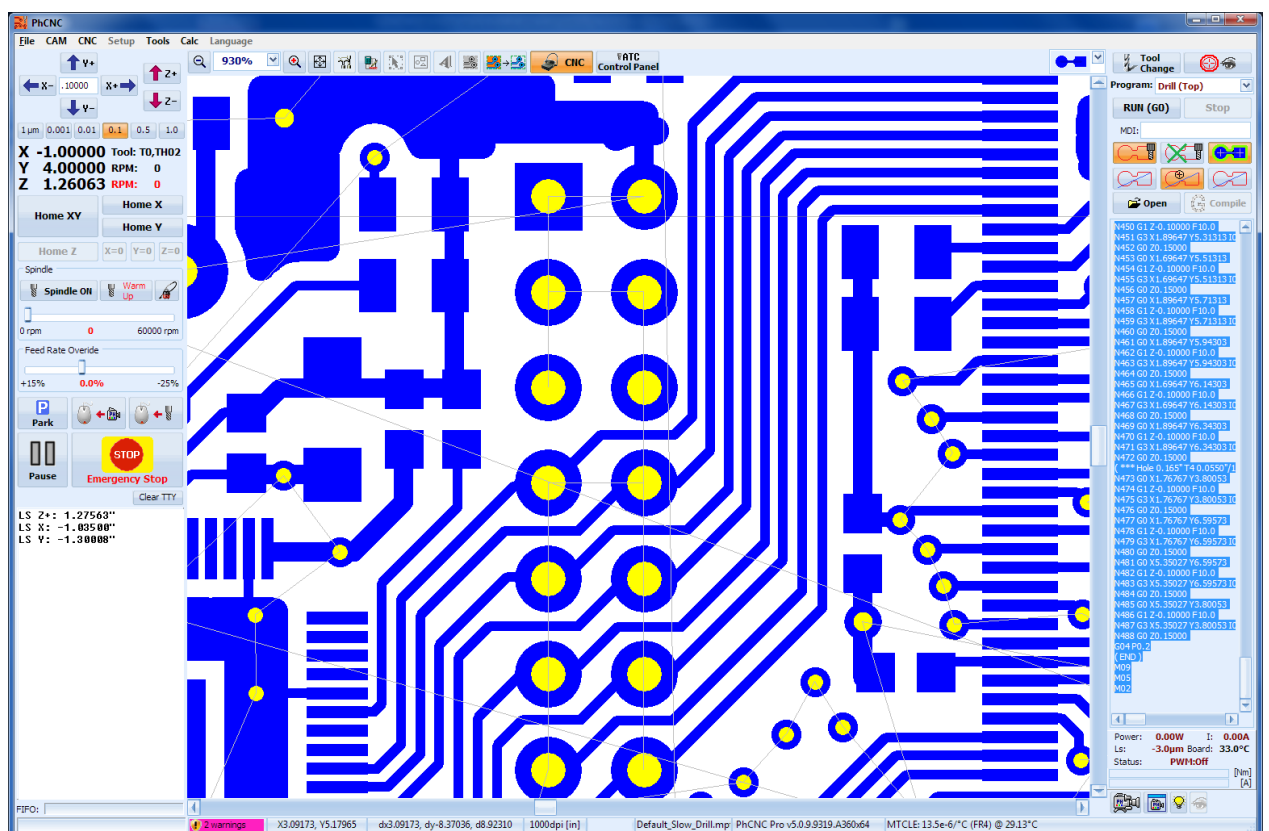
Note that the cutout must be the last!

Shown below are the tool paths of each program:

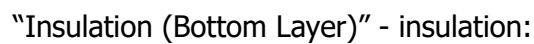
"Insulation (Top Layer)":

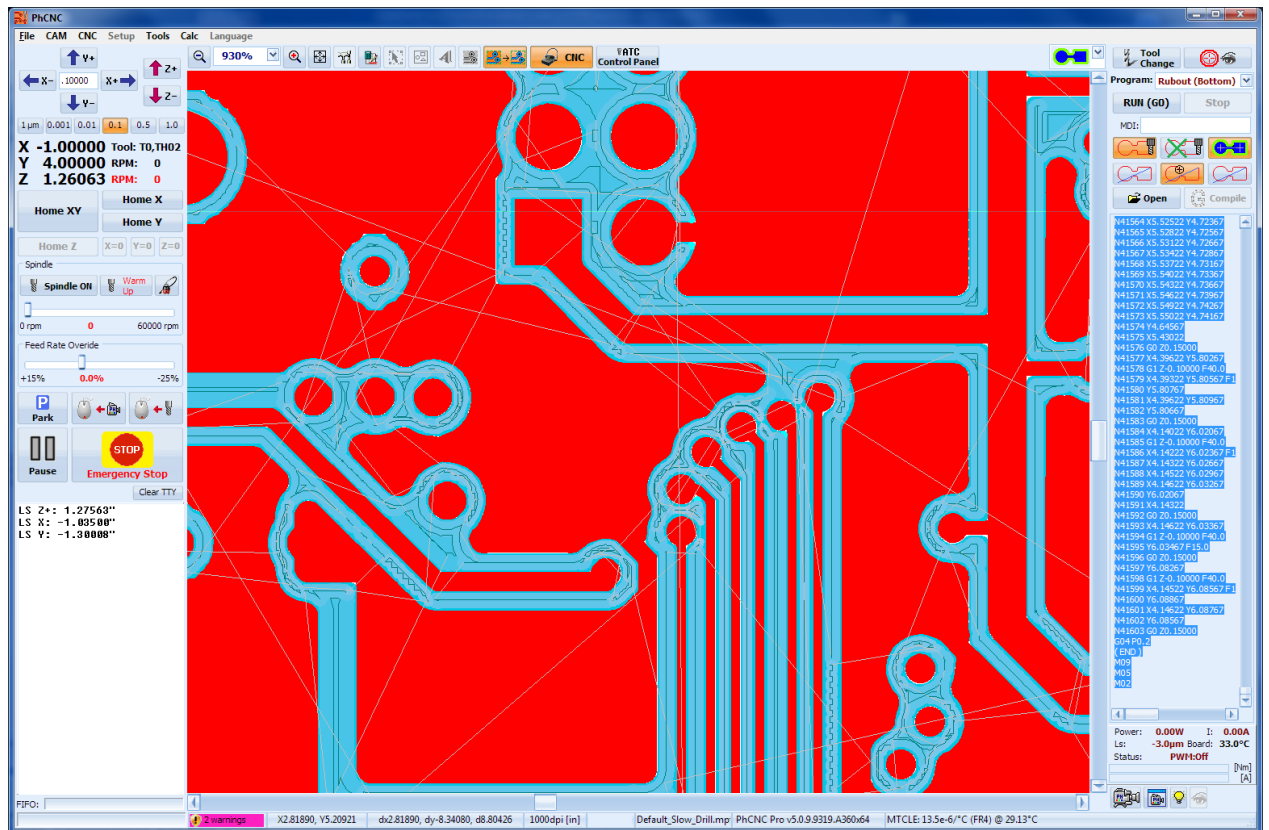


"Drill (Top Layer)":

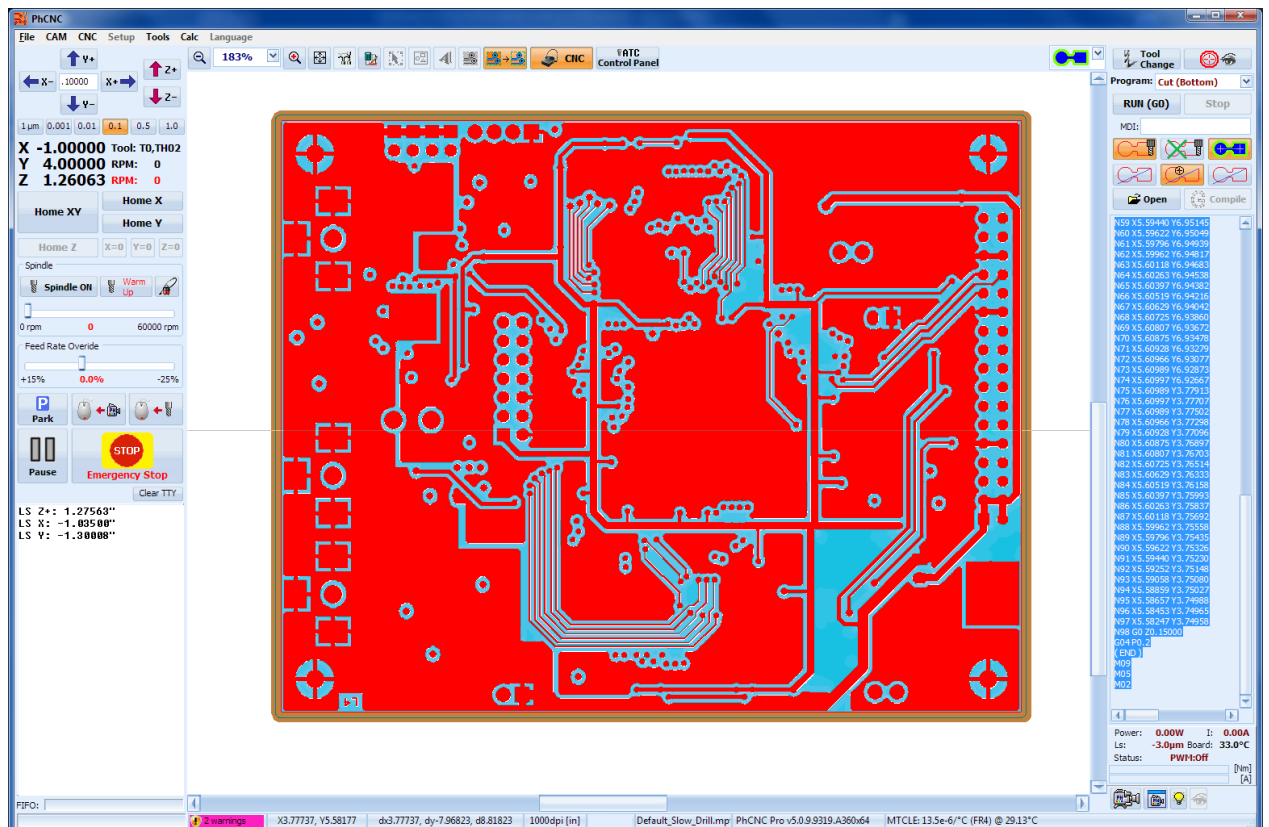


"Rubout (Top Layer)":

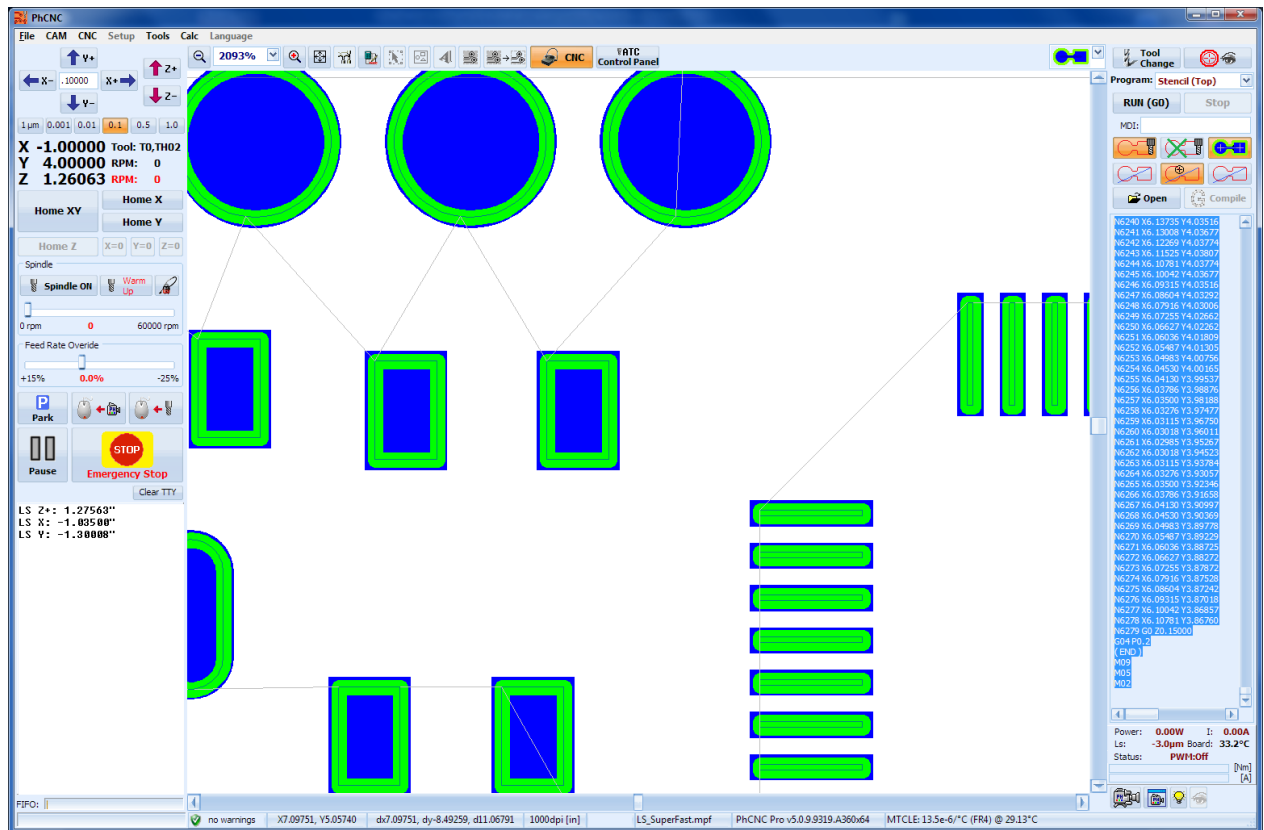




“Cut (Bottom Layer)”:



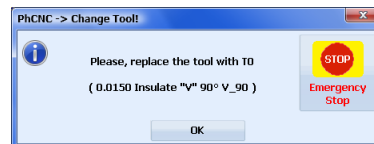
If during the import in CAM mode, you have layers declared as a stencil for solder paste (usually only on the top layer) these additional machining programs will appear - “Stencil (Top Layer)” and “Stencil (Bottom Layer)” (if any).



4.2.1 Control buttons for G-Code programs execution

4.2.1.1 Tool Changes

During program execution, PhCNC LSSE will ask you to insert the necessary tool for each segment of the program. When you are ready, confirm with the OK button:



Button  will allow you to initiate a tool change procedure at any time. If a program is in progress, it will be interrupted. The spindle will go to the tool change position and the window above will appear. After tool change is completed, you have to calibrate the tool if your machine does not have automatic calibration or computer assisted calibration. Confirm with the OK button if the machine has automatic tool calibration or computer assisted calibration it will go through this process first then the spindle will go back to the position, where it was before the interruption and the program execution resumes from that point. Tool change windows contain tool size information that was entered for that tool.

4.2.1.2

Starts the execution of the selected G-Code program.

Important:

PhCNC LSSE does a single "Home Z" at first start of program after it is activated. (see **Home Z**)

4.2.1.3

Stops the execution of the active G-Code program.

4.2.1.4 MDI

MDI is a command line for manual execution of G-Codes. If you use G0, G1, G2, G3, or other commands that require feed rate a feed rate value must be entered. Without it, the command will be ignored!

MDI:

During the execution of a G-Code program the MDI line is not functional.

Examples of MDI usage:

G0 X0 Y4.950

G1 X7.5 Y4.950 F15

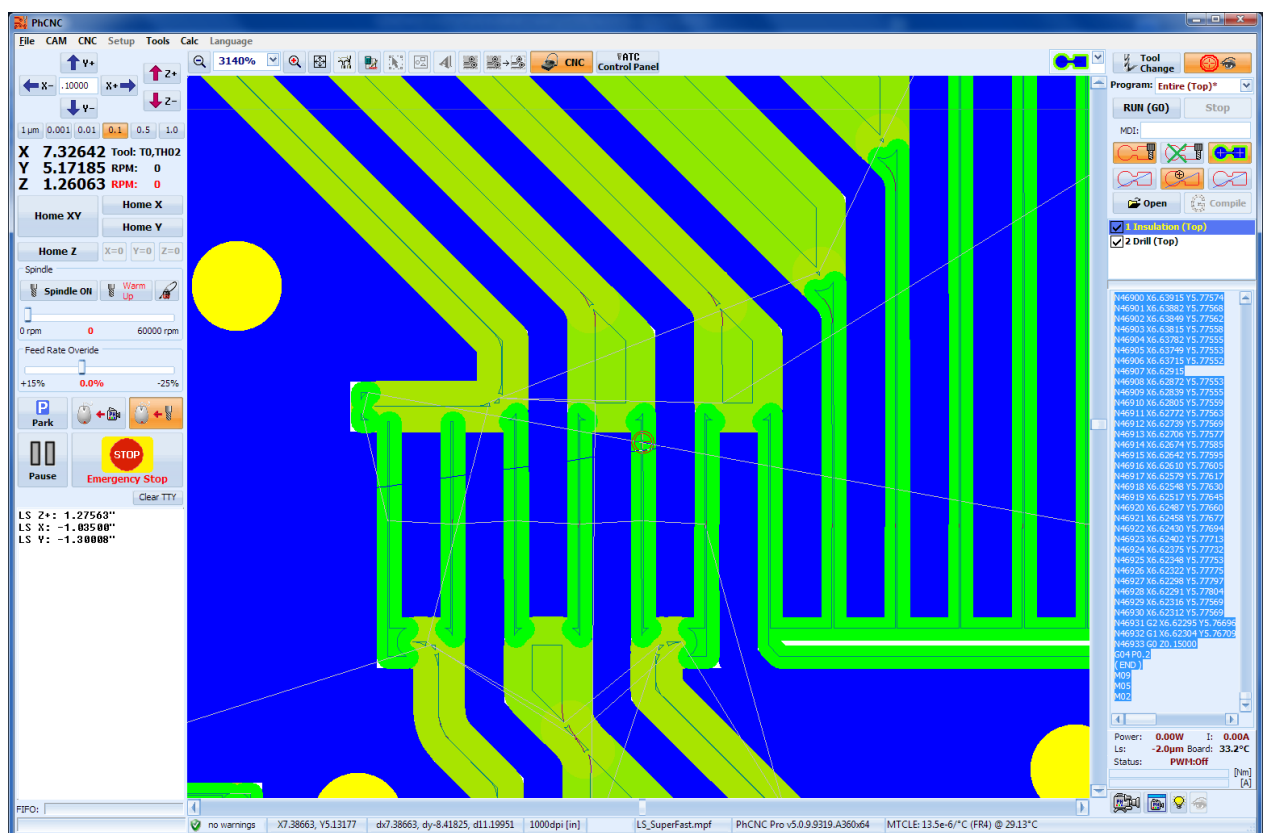
Important:

Execution of G/M commands from MDI line and G code programs generated by outside applications require high level of the operator experience. The results of the execution are in full responsibility of the operator.

4.2.1.4

This button activates "Scroll In View" screen mode. When active, "Scroll In View" keeps the spindle position visible by scrolling the window. This mode will be canceled automatically if the operator scrolls using the mouse.

Shown below is an active "Scroll In View" mode:



4.2.2 Drawing control buttons in CNC mode



4.2.2.1

This button shows/hides accumulated RAW information generated during the spindle movements. Red lines indicate negative Z coordinates (tool movements penetrated into the work piece), blue ones - positive or zero (usually movements above the surface of work piece).

4.2.2.2

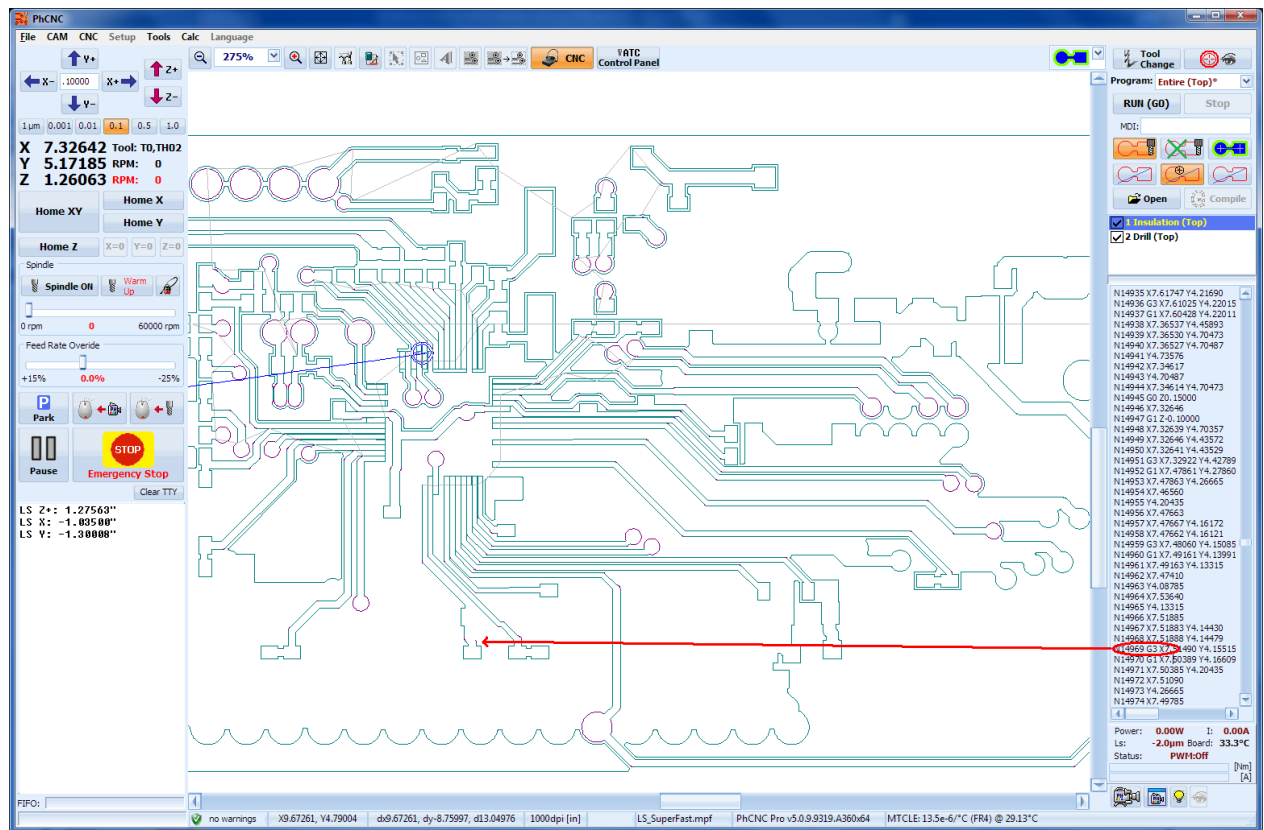
This button clears accumulated RAW information.

4.2.2.3

This button stops the drawing of the selected G-Code program, at the current cursor position. Consequently, you can match the position of the cursor with the line of code in the text field on the

right in order to be able to run the program to or from this point. You will need to switch the  Gerber hide/Show button to hide so you can see the G-Code drawing on the screen.

In the example below, the cursor is located at line N1233 and the screen shows all tool paths made up to the execution of this line of code. You can move the cursor to different lines using Up, Down, Page Up, Page Down or the mouse. The drawing on the screen will be updated immediately. This option helps to determine the function of each line in a G-Code program.



4.2.2.4

This button disables the  button and draws all tool paths generated by the entire G-Code. No matter where the cursor is.

4.2.2.5

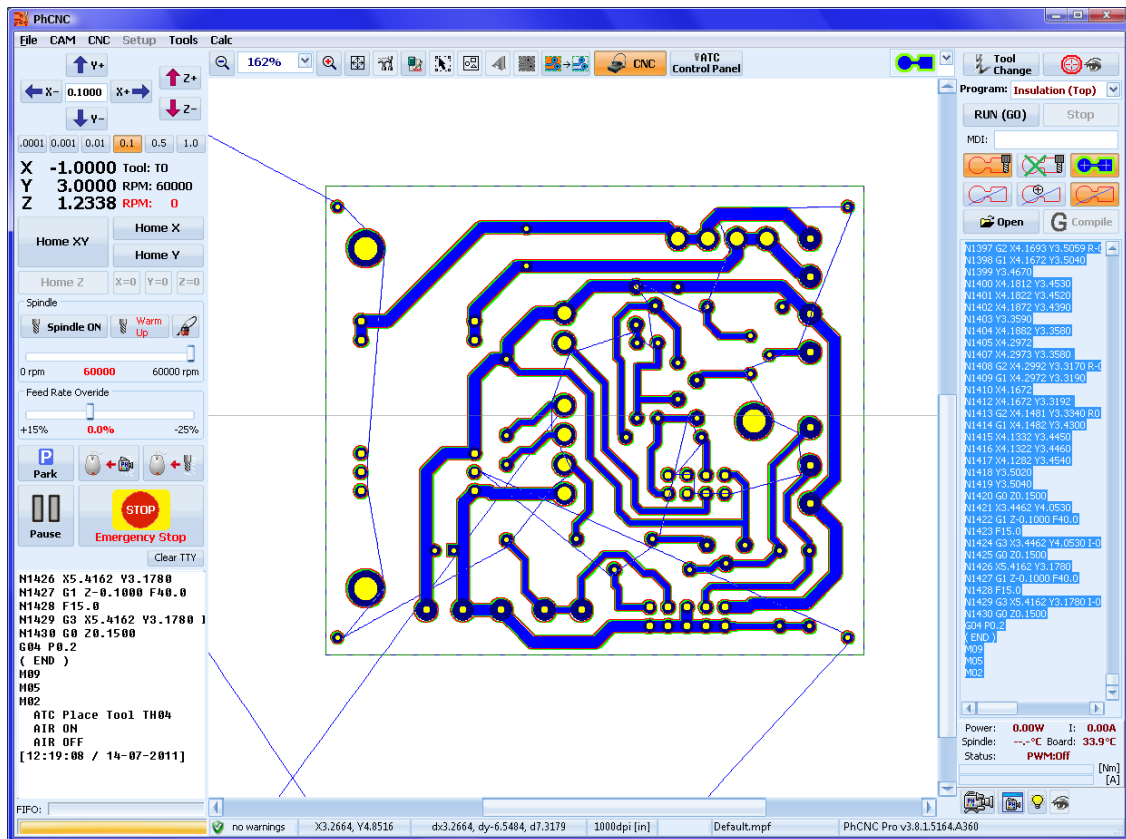
This button is used to recompile the G-Code and update the screen if changes were made by the operator. This option is only intended to allow viewing the changes.

When you use **RUN (GO)**, PhCNC LSSE automatically recompiles the G-Code.

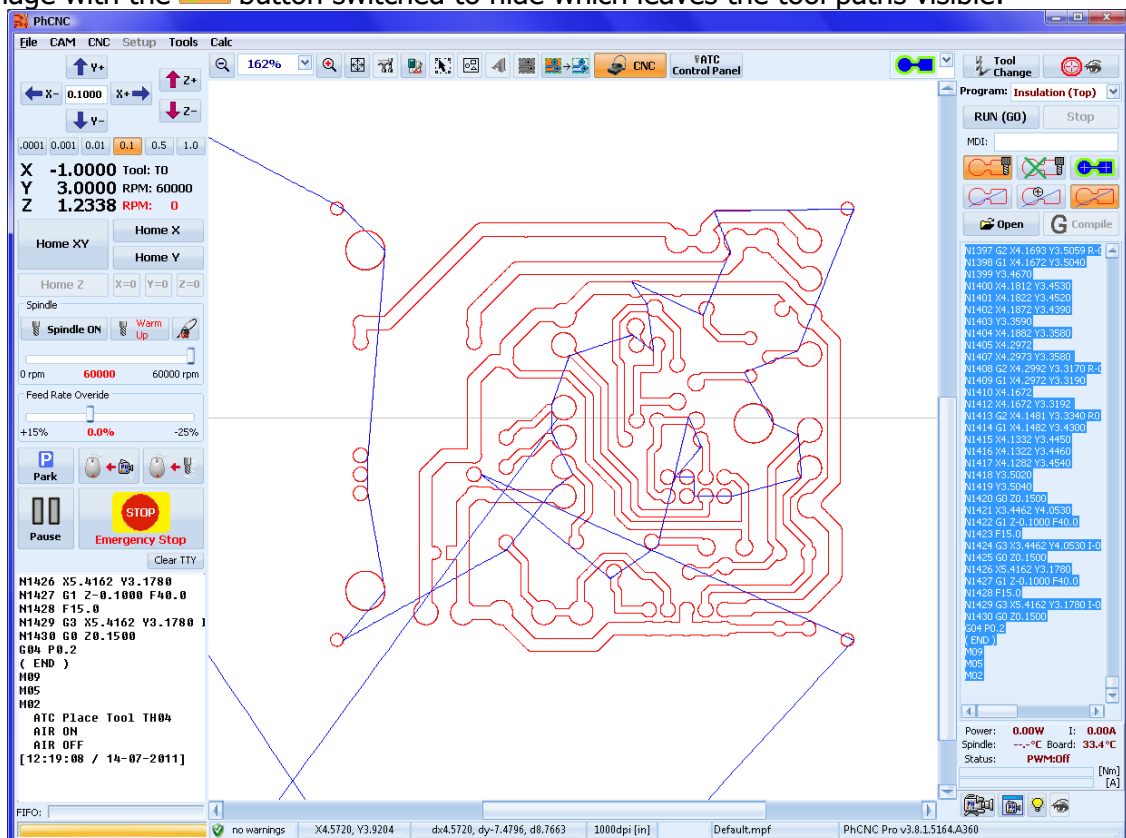
4.2.2.6

Show/Hide button for Gerber information from CAM mode. When its background is orange all drawing control buttons from CAM mode are available. Switching this button to hide is necessary to see the G-Code drawing on the screen.

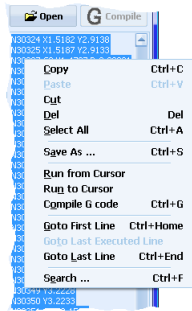
Shown below is a completed Top Layer insulation program the  button is set to show allowing the Gerber information to be visible:



Same image with the  button switched to hide which leaves the tool paths visible:



4.2.3 G-Code editor menu



This menu is available by right clicking inside the text window to the right. You can perform the following operations from this menu:

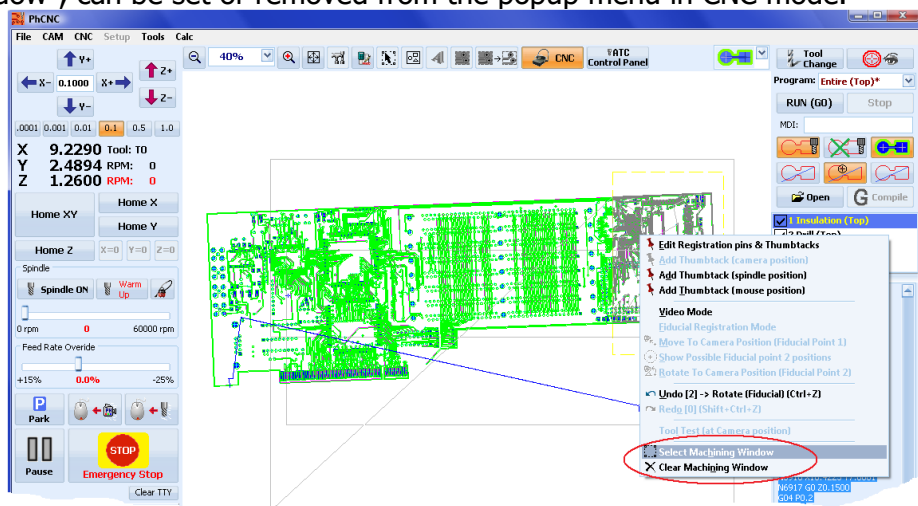
- **Copy:** this is the standard copy function used in most text editors and windows programs.
- **Paste:** this is the standard paste function used in most text editors and windows programs.
- **Cut:** this is the standard cut function used in most text editors and windows programs.
- **Del:** this is the standard Delete function used in most text editors and windows programs.
- **Select All:** This function selects all the G-Code at once.
- **Run from Cursor:** This function starts the machining process from the selected line.
- **Run to Cursor:** This function starts machining from the beginning to the selected line.
- **Compile G-Code:** This function is the same as the  button it compiles the G-Code after changes have been made with the editor (and refreshes the screen drawing). If not used in advance PhCNC LSSE executes this command automatically when the program is started.
- **Goto First Line:** This function positions the cursor at the first line.
- **Goto Last Executed Line:** This function positions the cursor to last executed line from your TTY window (when the machine has been stopped during machining program). You can use this command even during the machining progress.
- **Goto Last Line:** This function positions the cursor at the last line.
- **Search:** This function searches for a text match in the G-Code (Ctrl+F). Selects searched text and draws the G-Code to the selected line. Search can be started with "Enter" as well.

4.3. Machining Window

PhCNC LSSE has an option that we call "Machining Window". This option allows you to determine a square region of the board that will be machined in CNC mode (simulator only). Any parts of the project outside the selected region will not be machined.

"Machining Window" region is set automatically to cover entire work envelope in case that there is a project loaded that exceed the work envelope.

"Machining Window", can be set or removed from the popup menu in CNC mode.

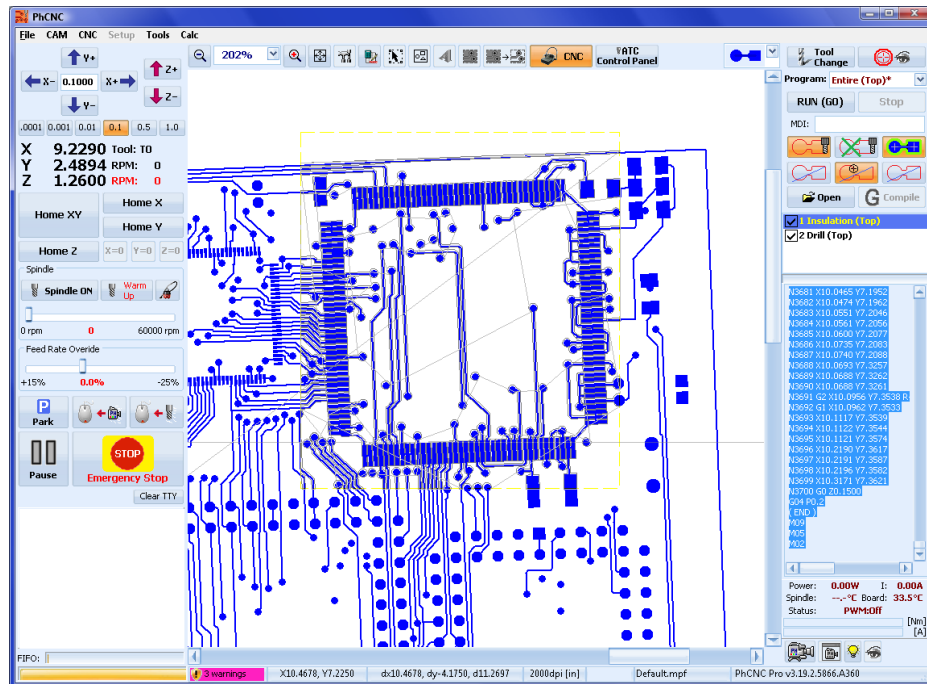


To select the "Machining Window" place the mouse cursor on the point that you like to be the lower left corner of the future "Machining Window" and make a right click. Then click on "Select Machining Window" line and move the mouse cursor to the desired right upper corner of the future window. Make a left mouse click that will complete the selection. Selected region must remain entirely in the work envelope.

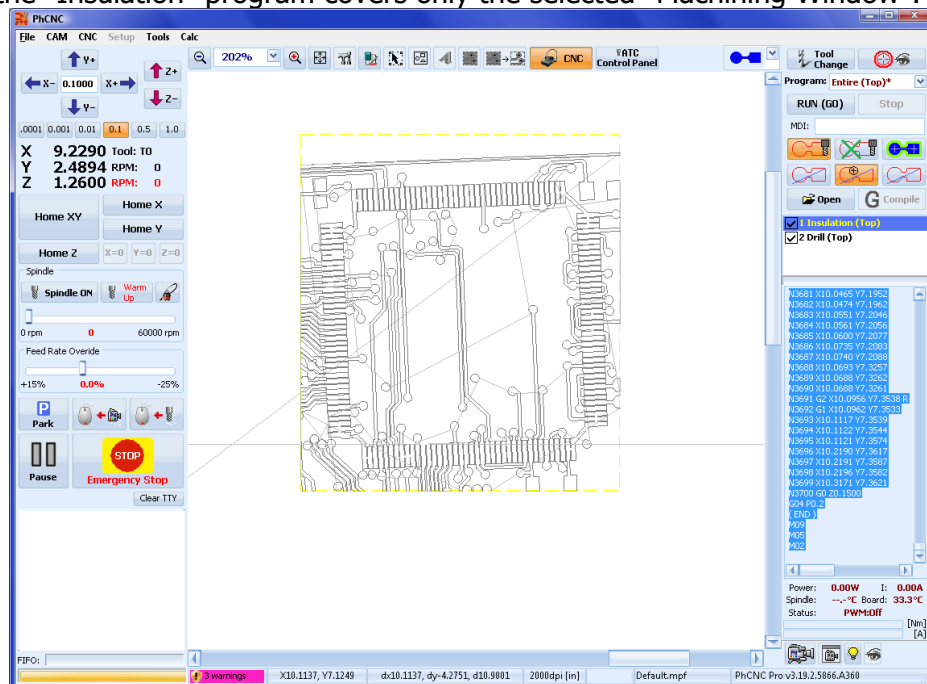
After selecting a new "Machining Window", PhCNC LSSE will automatically recalculate all the machining programs in order to machine only the selected region.

Example:

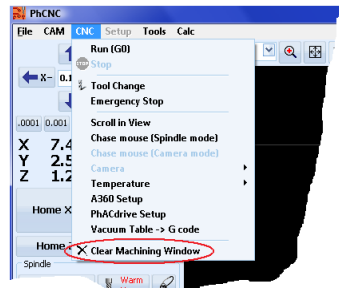
Shown below is a "Machining Window" that covers small part of the project.



As seen below the "Insulation" program covers only the selected "Machining Window":



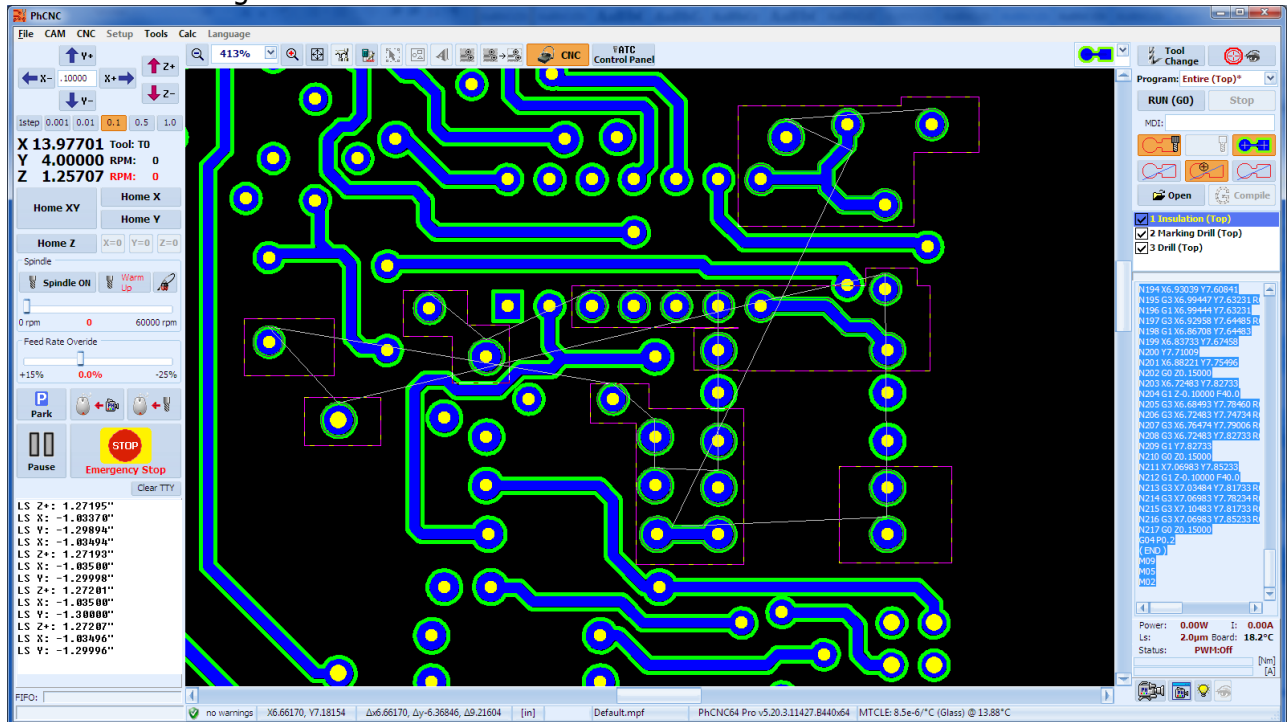
If you need to remove the "Machining Window" select "Clear Machining Window" from the popup menu. This function is also available in main menu CNC -> "Clear Machining Window" and in CAM mode of PhCNC LSSE.



Switching between CAM and CNC mode does not affect/remove the selected "Machining Window". Loading a new project will remove it.

4.4 Regions Selection

PhCNC LSSE allows selections of multiple regions for machining. The selection has to be made in CNC mode. Press the R key on your keyboard and use the mouse to select a region. Regions selected are united in a single G-code that is updated after any change in the region selection. All objects that are included in the regions will be machined.



Machining Window option has a higher priority and works independent of the regions selections. If there is a machining window already set and also regions selected the system will proceed only with the regions selected that are inside the machining window. All components outside the machining window will not be machined. The priority can be used in some cases, but require operators experience. In the beginning we don't recommend the usage of both region selections and machining window.

Clearing of the selected regions can be done using "Clear All Regions" of popup menu in CNC mode or in "CNC->Clear All Regions" (available in CAM mode too!)

5. In case of further questions

In case you were not able to find what you were looking for in the current manual, you can ask us a question in a dedicated email. Please, don't forget to provide screenshots, where applicable. Use the following address:

e-mail: support@telenet-eu.com

web: <https://www.telenet-eu.com>