PhCNC

PCB Prototyping Software

v5.75.1

(PhCNC & PhCNC LSSE)

© 2007-2021, TELENET LTD.

User's Manual

www.accuratecnc.com

Dec 6, 2021

Table of Contents

1. What is PhCNC? 6
1.1 How to use PhCNC 6
1.2 How to produce PCB prototypes using PhCNC? 7
2. CAM mode 8
2.1 Gerber/Drill Import 8
2.1.1 Window for Gerber/Drill files import 8
2.1.2 Gerber/Drill files import 9
2.1.3 "Gerber View" window 14
2.1.4 Profiles for automated Gerber & Excellon import 14
2.1.5 Gerber RS 274-D files import 17
2.1.6 Gerber files for Stencils Import 20
2.1.7 Convert Gerber to Drill 23
2.1.8 Info layers import 24
2.1.9 Gerber editor 25
2.1.10 Single layer PCB import 26
2.2 PhCNC Project Files 27
2.3 Buttons and fields used in CAM mode 27
2.4 Tools 32
2.4.1 "Insulate" Tools 34
2.4.2 "Cut" Tool 37
2.4.3 "Rubout" Tools 38
2.4.4 "Drill" Tool Table 44
2.4.5 "Stencil" Tool (Pro version only) 45
2.4.6 Project Tools 45
2.5 Object selection mode 47
2.6 Move Objects 48
2.6.1 Moving objects using the mouse 48
2.6.2 Move using "Rotate & Move Selected Objects …" 50
2.6.3 Move using "Move To …" 50
2.7 Clone (Copy) objects 51
2.8 Rotate objects 51
2.9 Group/ungroup objects 53
2.10 Delete objects 54
2.11 Rubout – excessive copper removal 54
2.12 Editor for the mechanical layer 57
2.12.1 Contour cutting selections 57
2.12.2 Contours selection in the mechanical layer 57
2.12.3 Contours generation with "PhCNC Printer" 57
2.12.4 Generation of contours with ("Insert Text") 61
2.13 Design Rules Violation (DRV) detector (Pro version only) 61
2.14 Thumbtacks & Registration pins (Pro version only) 63
2.14.1 Registration pins 64
2.14.2 Thumbtacks 66
2.15 Insulating copper rubout (Pro version only). 67
2.16 CAM350 import (Pro version only) 69

2.17 DXF import (Pro version only) ... 70
2.17.1 Selecting DXF layers for "Cut" operation ... 71
2.17.2 Selecting DXF layers for Drill & Countersink ... 71
2.17.3 Selecting DXF layers for milling (MILL) ... 71
2.18 Holes and pads insertion (Pro version only) ... 73
2.18.1 Holes insertion ... 74
2.18.2 Through-hole pads insertion ... 74
2.18.3 Insert Target 74
2.19 Vector Text insertion function (Pro version only) ... 75
2.20 Breakout Tabs (Pro version only) ... 78
2.20.1 Breakout tabs insertion ... 78
2.20.2 Breakout tabs removal ... 80
2.20.3 Breakout tab width setting ... 80
2.20.4 Breakout tabs complete removal. ... 81
2.21 Units of measurement [mm]/[in] ... 81
2.22 UNDO/REDO ... 81
2.23 Forced Insulation ... 83
2.24 Remove spikes (IPD) ... 84
2.25 Measure ... 87
2.26 G-Code Export (PhCNC LS/LSSE) ... 88
2.27 Gerber Export 92
2.28 Print ... 94
2.29 Panelization ... 95
2.30 Marking Drill ... 97
2.31 Holes Properties ... 99
2.32 QR Code ... 100
2.33 DataMatrix ... 101
2.34 Pick & Place Helper ... 102
3. Setup ... 104
3.1 Machine Setup ... 104
3.1.1 Depth Limiter ... 104
3.1.2 PhCNC Camera ... 104
3.1.3 Disabled depth control ... 105
3.1.4 PhCNC Vacuum Control Options ... 105
3.1.5 PhCNC Power Management Options ... 105
3.2 CNC process control settings ... 106
3.3 G-Code post processor adjustments and selections ... 107
3.4 Display settings ... 107
3.5 Multithreading & PhCNC64 ... 109
4. CNC mode ... 110
4.1 Manual controls ... 111
4.1.1 Spindle/camera positioning using Jog buttons or mouse ... 111
4.1.2 Home X & Y ... 112
4.1.3 Home Z ... 112
4.1.4 "X=0", "Y=0, "Z=0" Buttons ... 112
4.1.5 Spindle & Vacuum control ... 112
4.1.6 Feed Rate Override ... 113

4.1.7 Emergency Stop 113
4.1.8 Pause 113
4.1.9 Park 113
4.2 Program execution in CNC mode 113
4.2.1 Control buttons for G-Code programs execution 117
4.2.2 Drawing control buttons in CNC mode 118
4.2.3 G-Code editor menu 121
4.2.4 Programs and operations execution using a ATC system 121
4.3 Tool Test (Pro version only) 123
4.4 ATC Control Panel 124
4.4.1 "Tool Holders Check" 125
4.4.2 "Return Tool" 126
4.4.3 "Get Tool" 126
4.4.4 "Place Tool" 126
4.4.5 "Replace Tool" 126
4.4.6 "Tool Settings" 126
4.4.7 "Manual Tool Change" 127
4.4.8 "Open Spindle's collet" 127
4.4.9 "Park" 127
4.4.10 "EmStop" 127
4.4.11 "Check tool's ring" 127
4.4.12 "Auto Tool Depth Update" 127
4.4.13 "Manual Depth Control Mode" 128
4.5 ATC tool magazine 128
4.5.1 Tool placement using the "ATC Control Panel" 128
4.5.2 Manual tool placement 129
4.6. Machining Window (Pro version only) 129
4.6.1 Applications of the "Machining Window" option 129
4.6.2 Selection of "Machining Window" region 130
4.7 Regions Selection 133
4.8 Material Thermal Coefficient of Linear Expansion (MTCLE) 134
5. Video mode of PhCNC 135
5.1 Video mode activation 135
5.2 Popup window of "PhCNC Video" 136
5.2.1 "PhCNC Video" control buttons 136
5.2.2 Other functions of "PhCNC Video" window 136
5.3 Integrated window "CNC Video" 137
6. Fiducial Registration (Pro version only) 140
6.1 Auto Fiducial Registration 140
6.1.1 Execute AFR 140
6.1.2 Assisted AFR 141
6.1.3 Execute Fine AFR 141
6.2 Auto Hole Detector 142
6.2.1 "Move to Hole Center" 142
6.3 Fiducial Registration (Manual mode) 144
6.4 "2D Fiducial Transformation" Function 151
7. G code commands available in PhCNC 154

8. PhCNC Calculator 157
APPENDIX-A: How to calibrate PhCNC video camera (VC) 158
A.1 VC angle calibration 159
A.2 VC magnification calibration. 160
A.3 X offset calibration 160
A.4 Y offset calibration 161
APPENDIX-D Tool Penetration Depth 163
D.1 Tool change operation in ATC machines 163
D.2 Tool change operation in MTC machines (Manual tool change) 164
APPENDIX-E: Front panel LED indicators 165
APPENDIX-F: PhCNC controllers 166
F.1 PhCNC360 USB controller (2008-2015) 166
F.2 Firmware updates 166
F.3 PhACdrive firmware 166
F.4 PhSTdrive firmware 167
F.5 Additional status information for the machines equipped with PhACdrive 167
APPENDIX-G: USB drivers installation 168
APPENDIX-I: PhCNC control programs installation 169
I.1 First time PhCNC installation 169
I.2 Upgrade of an existing installation 170

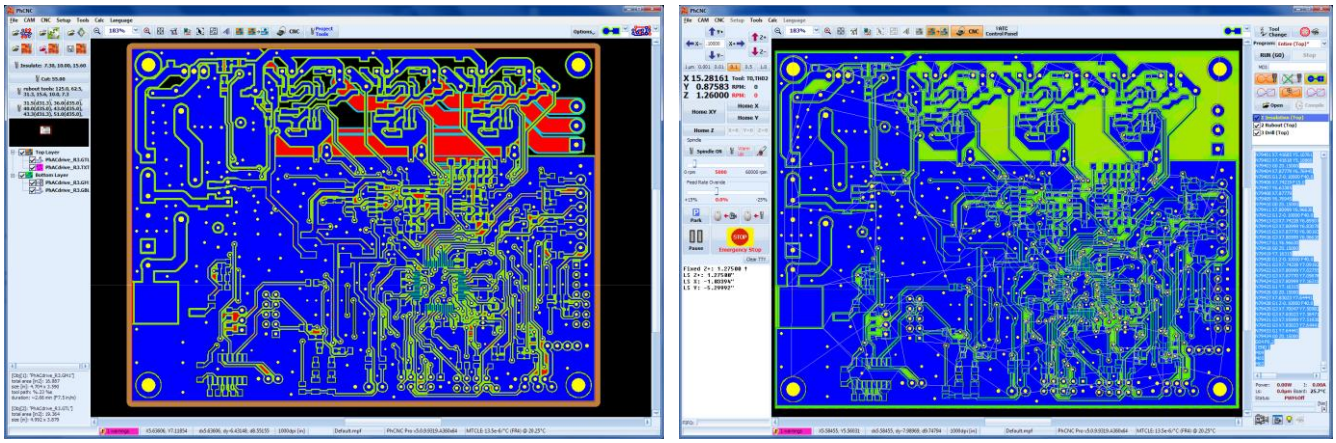
1. What is PhCNC?

PhCNC is an user friendly, extremely accurate, specialized software for PCB (Printed Circuit Board) prototyping for desktop CNC machines. The clear and simplified user interface utilizes pop-up menus, function buttons and high quality graphics. It offers many automatic selections and calculations for standard operations, which can be accessed and modified by the user for special and custom designs. PhCNC also offers a wide variety of control tools for panels/nameplates engraving, solder paste stencil production, and small parts production.

1.1 How to use PhCNC

PhCNC has two modes of operation, namely, the CAM and CNC modes. At start up, PhCNC enters the CAM mode, where you can open your files and edit the final layout of your prototype. Here, you can also specify the tools, which will be used. In the CNC Mode, the user can execute the programs that have been generated by PhCNC to control the machine. Note that you can switch between the two modes at any time. If you have our video camera/microscope you will be able to measure and inspect your project, and use precise optical registration to align your work piece. Below is a screenshot of each of them:

CAM mode **CNC mode**



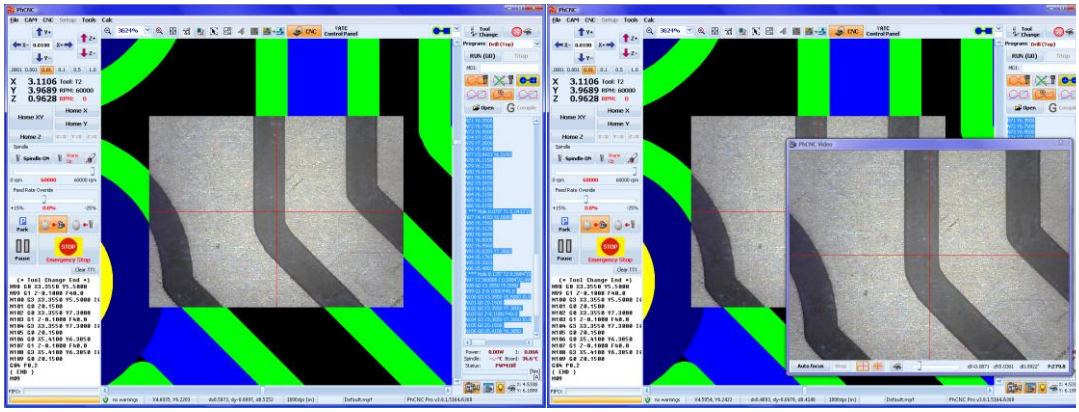
Below is a list of available operations in the CAM mode:

- Load one or more PCB designs (new visual interface helps Gerber/Drill/Stencil* files selection)
- Group/Ungroup multiple PCB designs and other separate objects
- Copy/Delete/Select/Deselect/Move objects or groups
- Rotate objects or groups by an arbitrary angle
- Visually check generated insulation tool paths.
- Insulation with up to **8** insulation tools.
- Automatic detection of Design Rules Violation (DRV).
- Forced Insulation.
- Remove spikes (IPD).
- Assign regions for excessive copper rubout and choose the tools to be used in the rubout process
- Additional traces insulation. A mode that can replace "copper rubout" in many cases, it is faster and saves tools.
- Rubout with up to **16** tools (**8** insulation + **8** rubout tools).
- Select inner/outer tool diameter compensation for all cutting operations.
- Breakout tabs insertion.
- Vector text insertion.
- Holes and pads insertion.
- Universal input "PhCNC printer" driver allows you to import your design for front panels and nameplates from any CAD software (not available in the demo version).
- Load/Save/Import PhCNC project files (PHJ). Excellent function for repeated projects manufacturing.
- Panelization
- Gerber Export
- Print
- Import of CAM350 CAM files (includes visual interface for CAM layers).
- Import of 2D DXF files for parts machining.

Below is a list of available operations in the CNC mode:

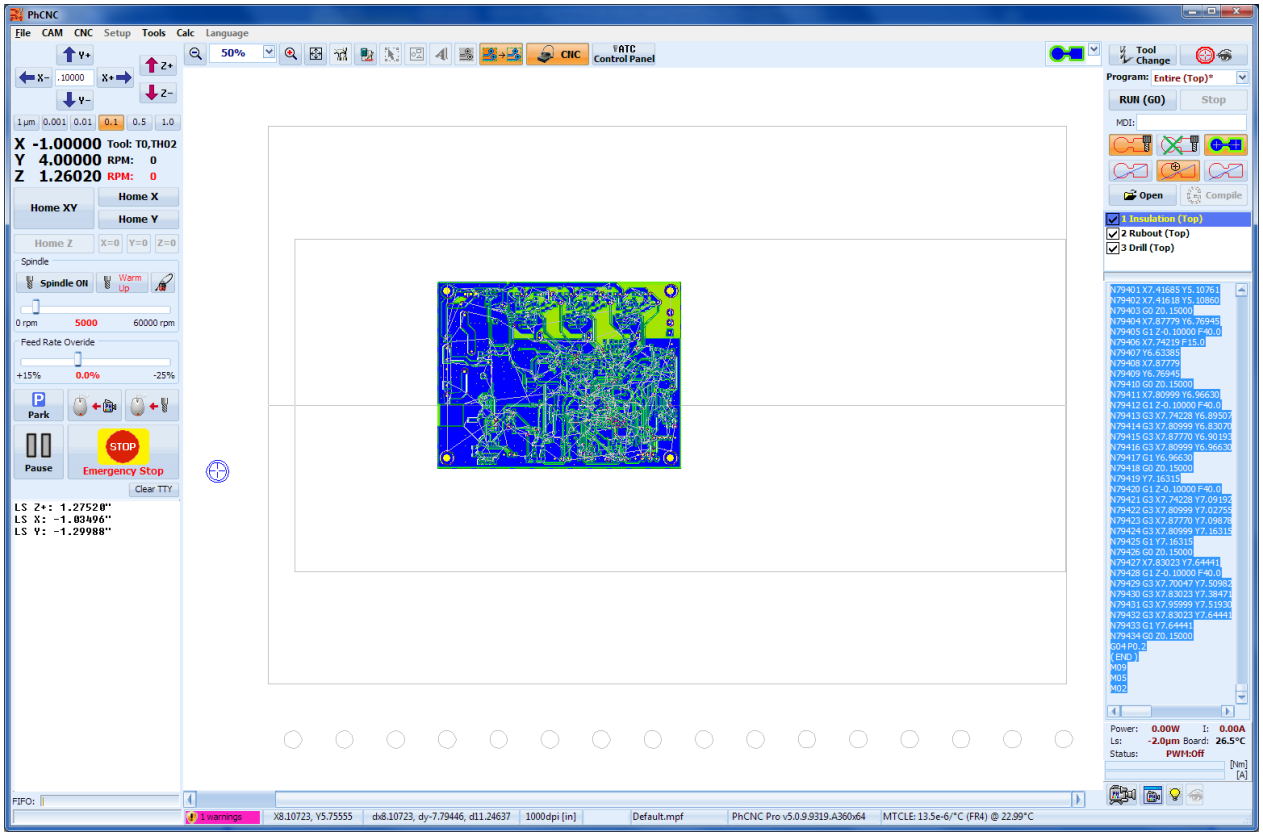
- Insulation (top and bottom layer)
- Rubout (top and bottom layer)
- Drill (top or bottom layer)
- Cutout (top or bottom layer)
- Stencil (top and bottom layer)
- Countersink (top or bottom layer)
- "Peck" mode is now available in PhCNC Pro version. It is used for high aspect ratio machining.
- Auto Fiducial Registration.
- Quality control. Positioning and measurements with optional camera/microscope

CNC Video mode #1 **CNC Video mode #2**



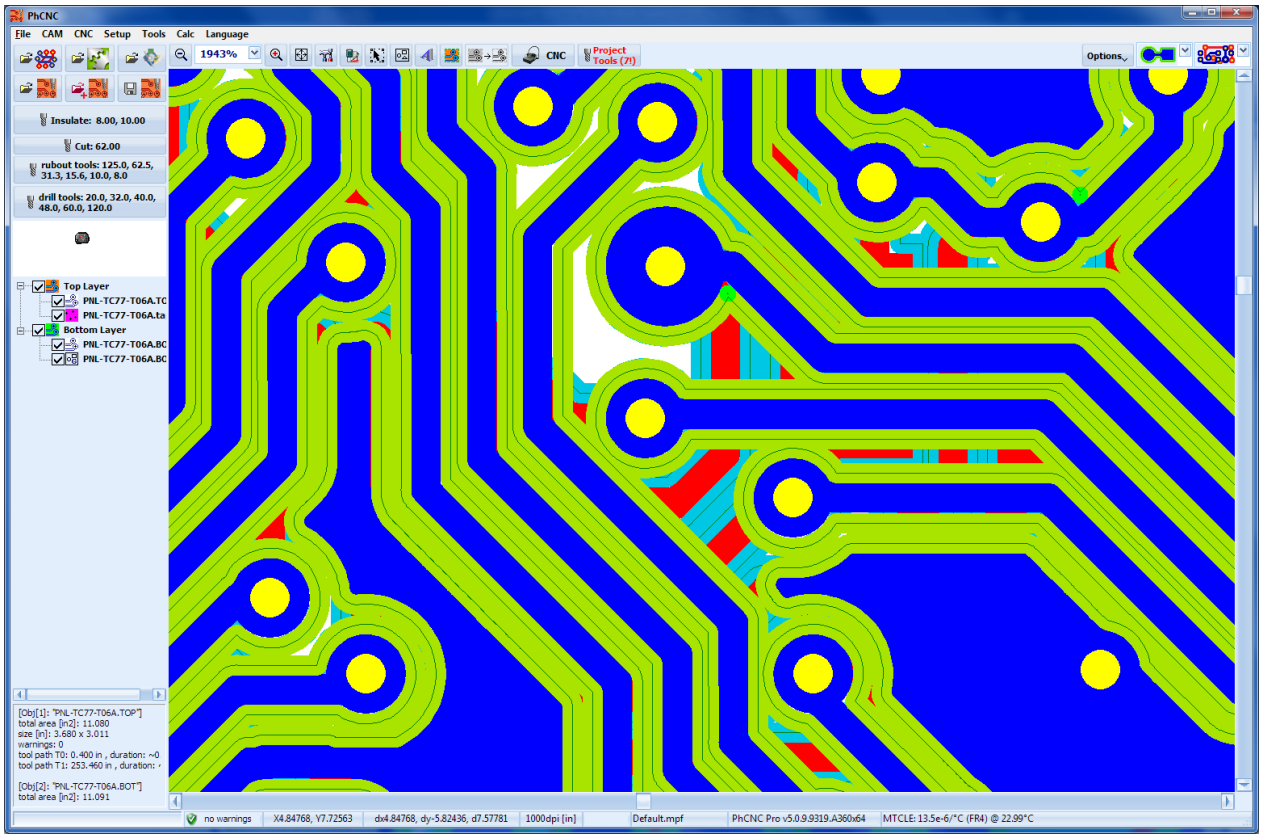
1.2 How to produce PCB prototypes using PhCNC?

- Load your CAM data (Gerber, Drill, Drill definition, and Mechanical) that comes from your CAD system
- Enter cam mode arrange your board (one or more) in the work-space, determine the copper rubout regions, clone (copy), rotate, check cutout (mechanical layer).
- Enter CNC mode, run the control (G-Code) programs created by PhCNC. Depending on your design, you will have to run some or all of the following: drill, insulate, copper rubout, cut internal shapes, and cutout.



2. CAM mode

When you start PhCNC it loads in CAM mode. In this mode you will be able to load and edit your CAM data in order to be ready for prototyping in CNC mode.



When in CAM mode you will be able to 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

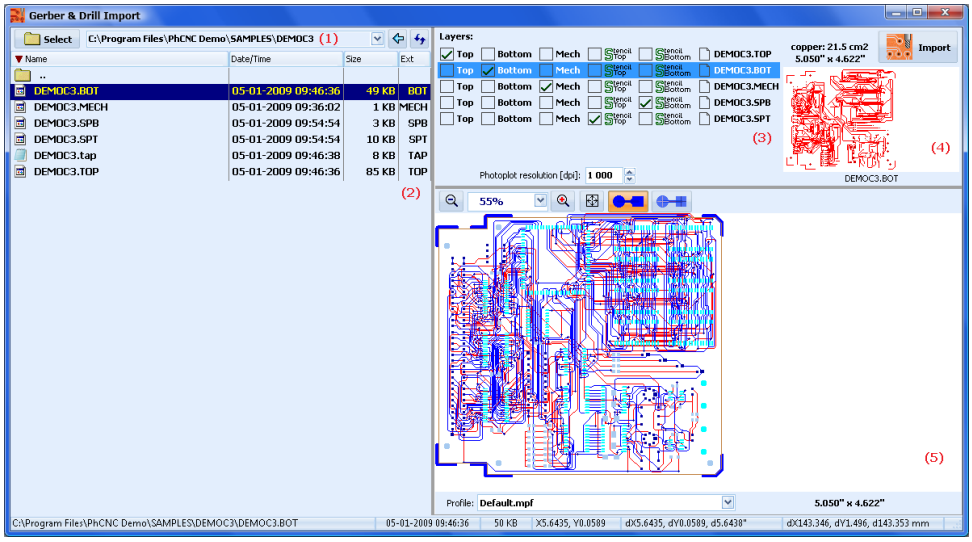
2.1 Gerber/Drill Import

The newest revisions of PhCNC (1.9.2 and up) utilize a visual approach to the process of Gerber/Drill files import. It is based on GerberView technology of PhCNC.
The new button , replaces

used on older revisions.

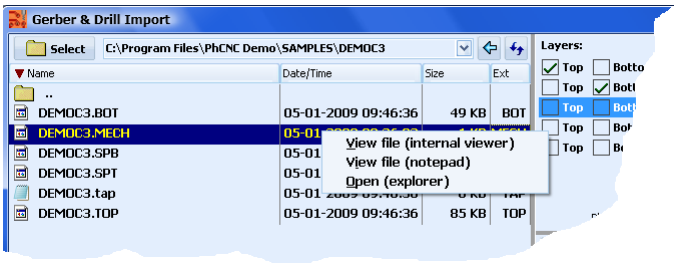
2.1.1 Window for Gerber/Drill files import

Shown below is the basic view of the new 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 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 now very simple. Folders can be selected with this button Select 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.

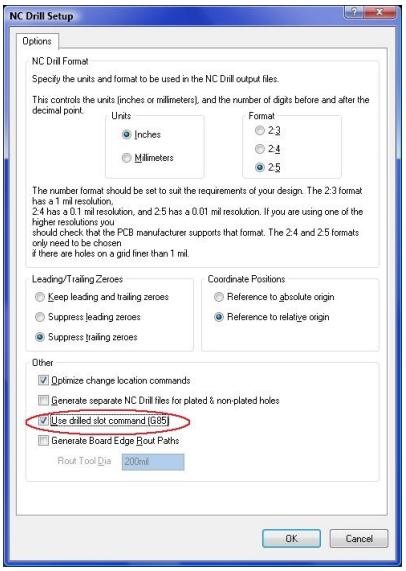
2.1.2 Gerber/Drill files import

To select a file simply click on it in the list **(2)**. PhCNC 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.
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.
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 next file selection.

Note:
You can change the status/layer of selected files at any time.
Beginning with v3.24.2 it is possible to import more than one drill/drill-slot files (up to 8).

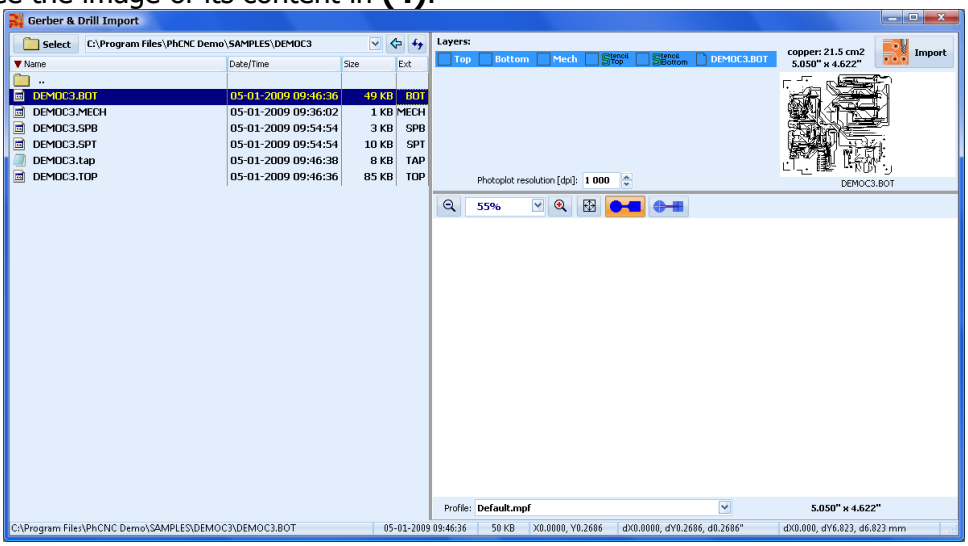
Note:
Only drill-slot files containing G85 command are good for this import

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

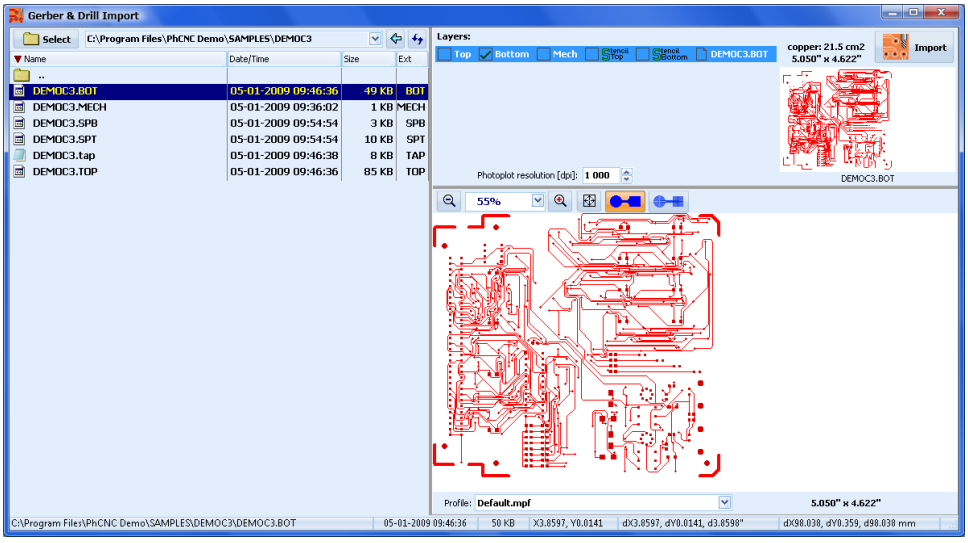


Important:
A router bit or drill bit designed for drill-slot operations will serve you better than traditional drill bits.

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)**.

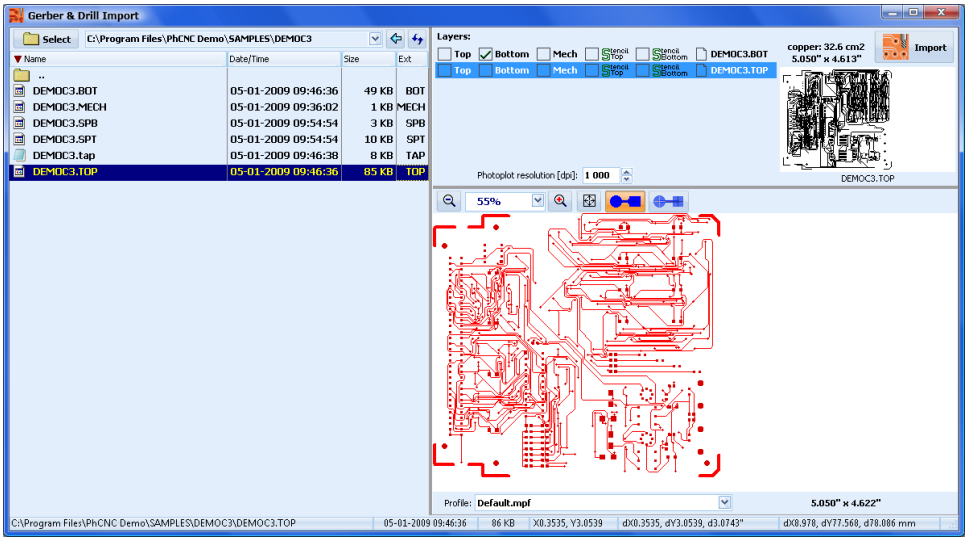


We will select "Bottom" as the status of this file.

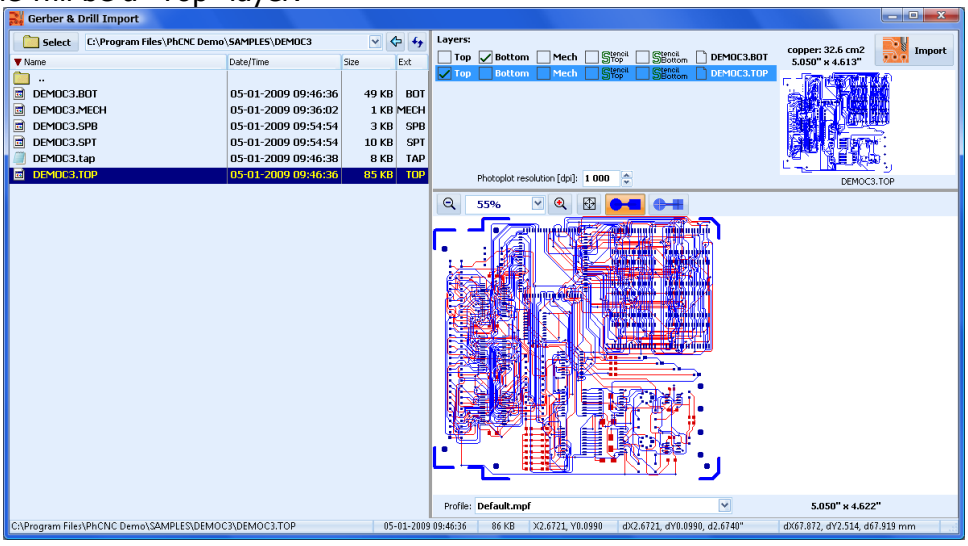


It will appear in the "Gerber View" window **(5)** with the color for bottom layer.

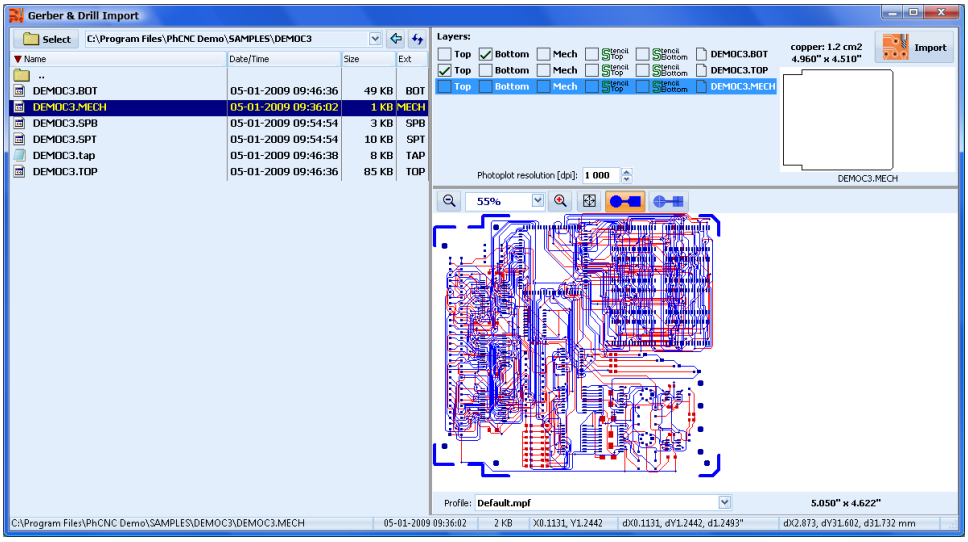
Next file selection:



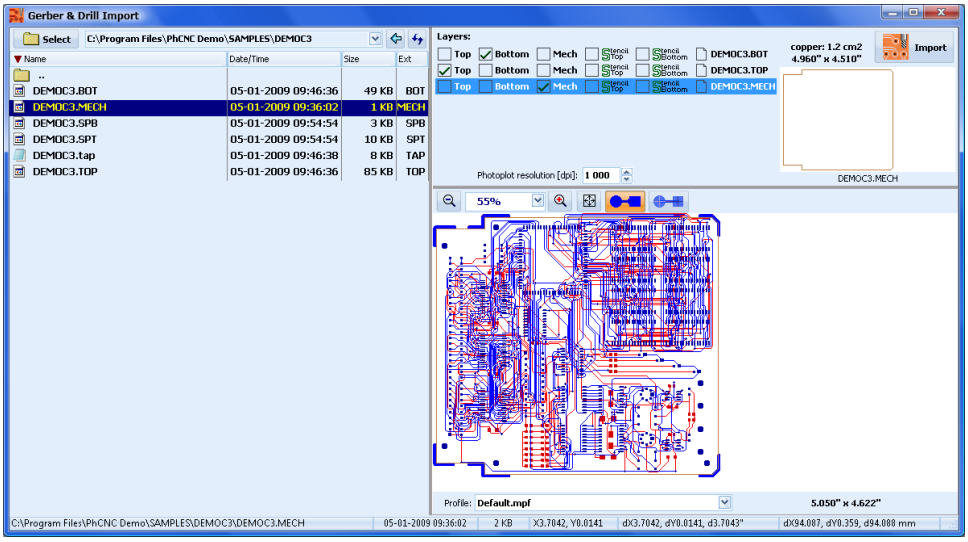
The selected file will be a "Top" layer:



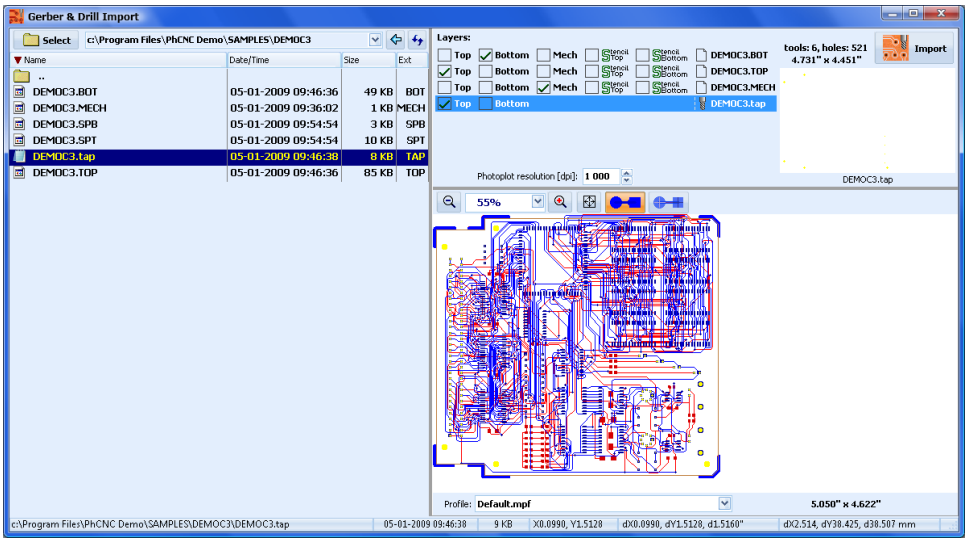
Third file selection:



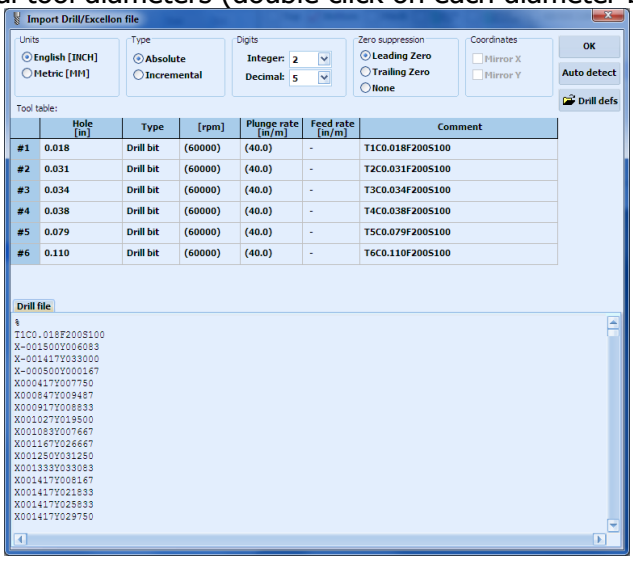
This file is a "Mech" (mechanical) layer file. It contains the internal cuttings and final cutout of the board.



The next selected file is recognized by PhCNC 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 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 Drill defs 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 a 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.

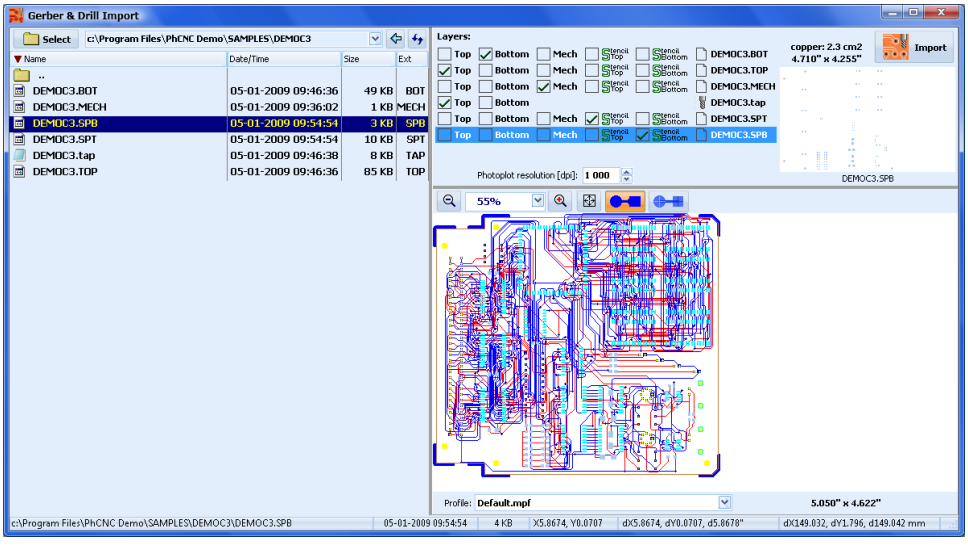
In all cases that the drill file(s) need manipulated we recommend the following sequence:

- 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.

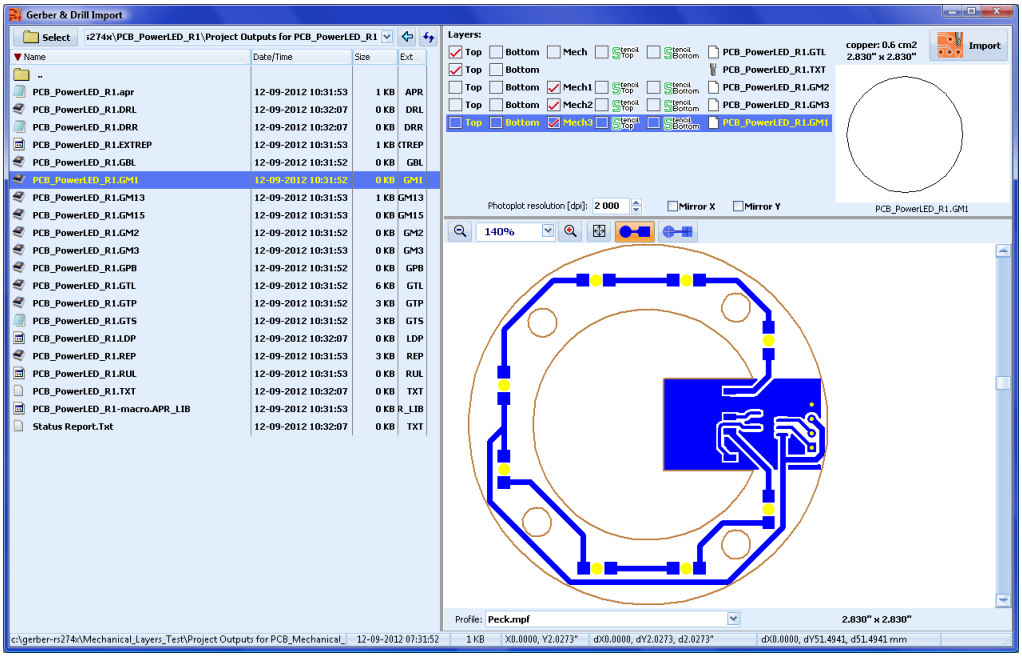
If for some reason the file format is not correct (like inch/metric, fractional digits, leading/trailing zeros suppression etc.) it will be very obvious. To correct the problem use double click at file name and change the necessary settings.

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



Important:
Stencil import is available only in PhCNC Pro versions (For more details see 2.1.4).

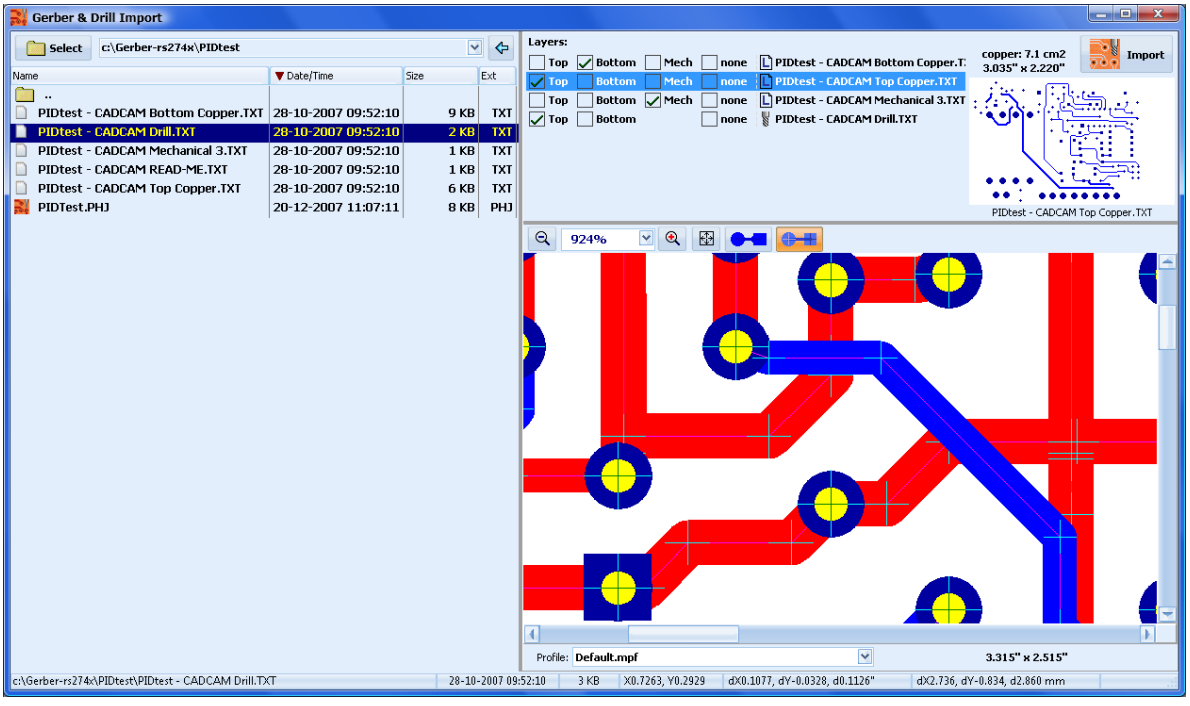
Note:
PhCNC revisions v3.19.3 and later, allows the operator to import more than one mechanical layer (up to 8 layers are allowed). This is shown on the screen below. Each layer is imported from a separate Gerber file.



After the import of more than one mechanical layer, PhCNC 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.

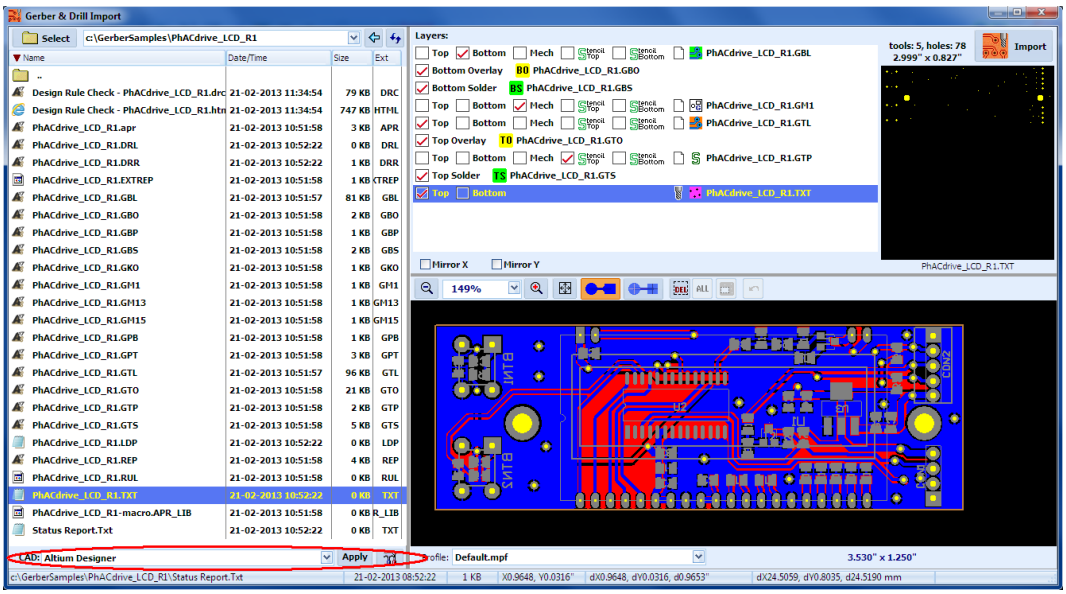
In the video clip below, the "Gerber & Drill files import" is demonstrated:
http://youtu.be/JIx5mMC7Cqg

2.1.4 Profiles for automated Gerber & Excellon import

PhCNC v5.33+ has ability to be created profiles for automated import to the generated from the PCB CAD systems Gerber & Excellon files for the layers.

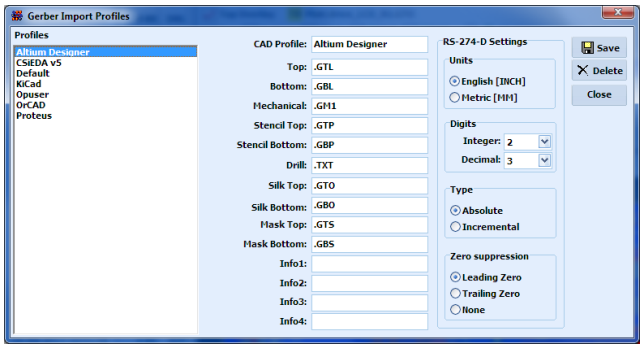
The predefined profiles are shown in the list below. When the user select the desired PCB CAD profile and press the button "Apply", PhCNC will select automatically the files corresponding to the PCB layers (according to the selected CAM profile).

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



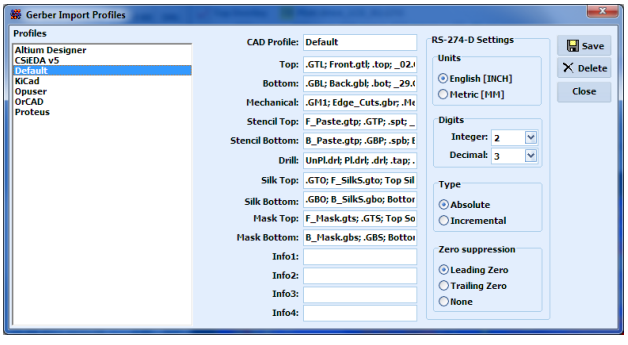
The predefined profile "Default" is universal profile, and will work for most of the PCB CAD outputs (Gerber & Excellon).

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 then one extention for the file name for every layer. Typical example is the profile "Default":

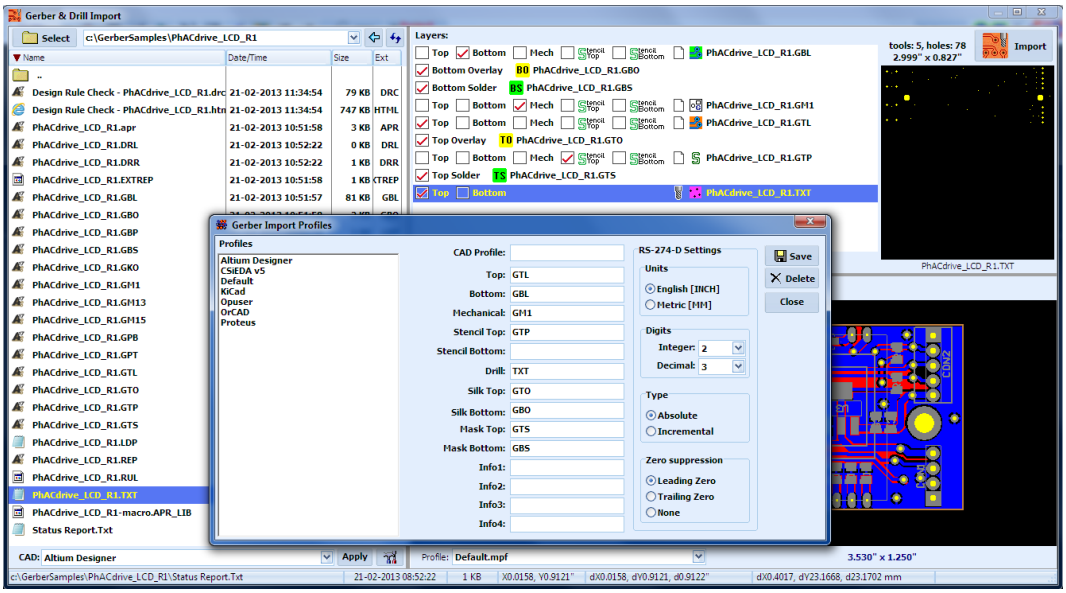


The different file extensions must be separated with ";".

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:



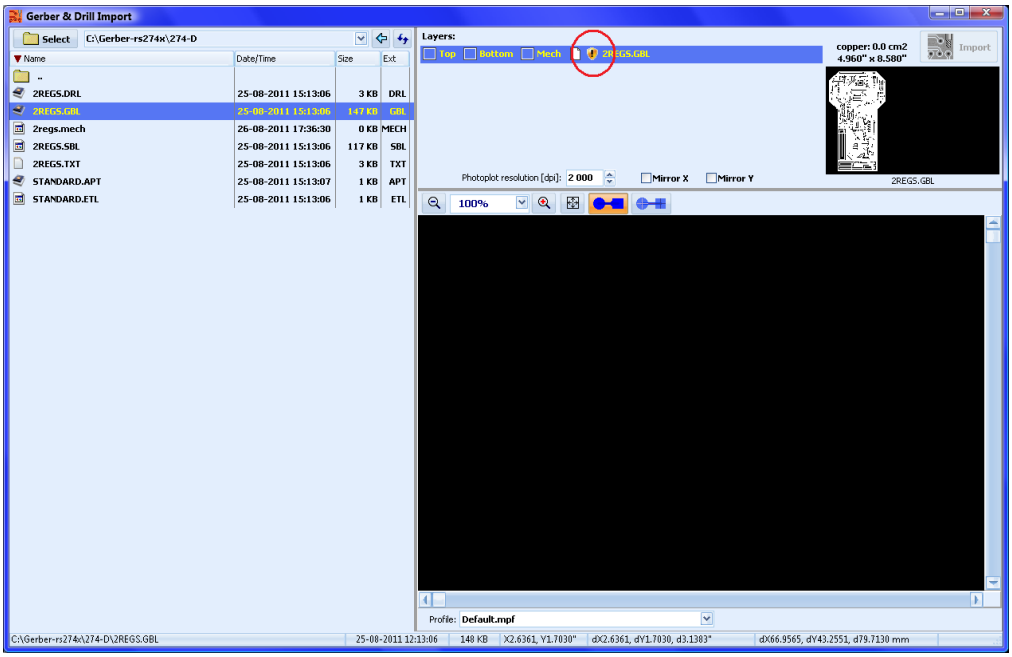
- Select the Gerber & Excellon files for the layers (ready for import into PhCNC)
- Press the button
Now PhCNC will fill the unique file names extentions (as is shown above) using the names of already selected files.

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

Video:
https://youtu.be/KQ6m_ytng90

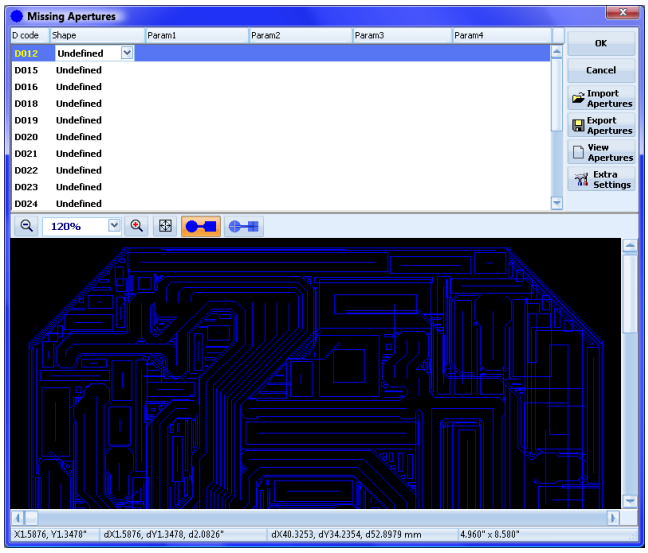
2.1.5 Gerber RS 274-D files import

Gerber files in RS 274-D format are considered obsolete. All PCB CAD systems now generate Gerber files in RS 274-X (called Extended Gerber). Despite this we offer the possibility to import RS 247-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):

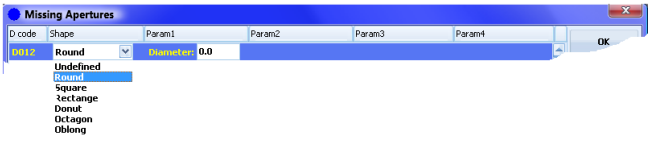


During the file selection an additional window will appear for apertures to be entered. Due to the fact that there is no standard format for aperture descriptions in RS 274-D they must be entered manually

2.1.5.1 Manual aperture entry

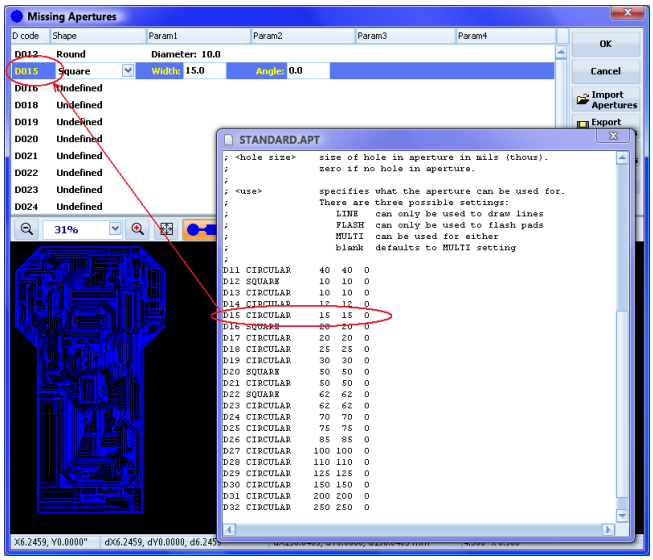


To enter/change parameters you have to click on it. This will open fields for its aperture type and parameters.



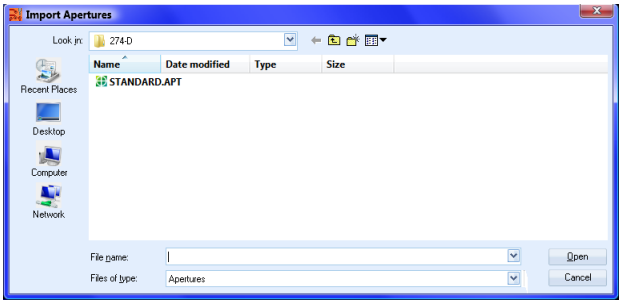
After the apertures and their parameters are entered it is recommended to save them in a *.APT file for future use (another layer or another project) using the button "Export Apertures".

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.

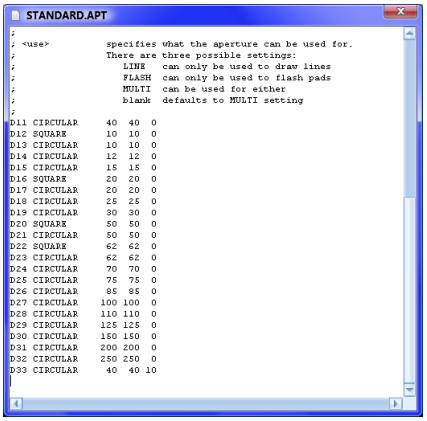


2.1.5.2 Aperture definitions import in PhCNC (file *.APT)

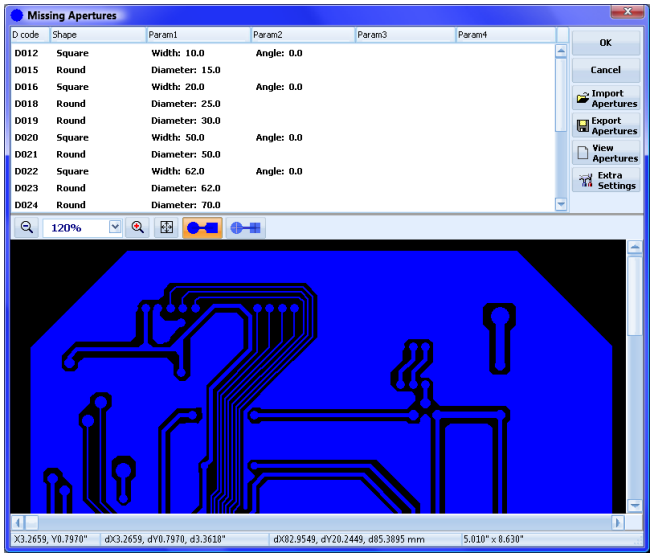
You can import apertures definitions in PhCNC when you use a RS 274-D Gerber format.
Once the aperture file is generated (or supplied) you can enter it using "Import Apertures"



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

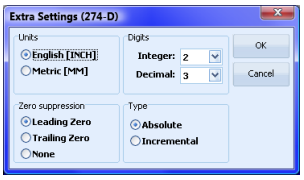


After the import of the apertures it is highly recommended to review the graphics of the layer for missing or improperly entered apertures. After any changes in the apertures, PhCNC will redraw/display the changes in the layer.

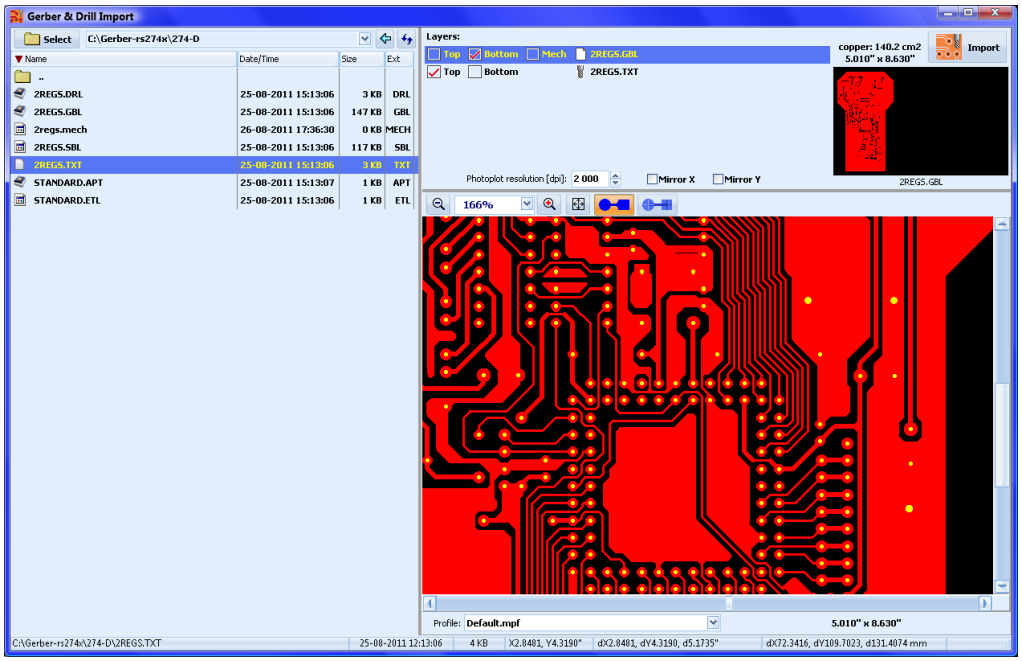


2.1.5.3 Extra Settings

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



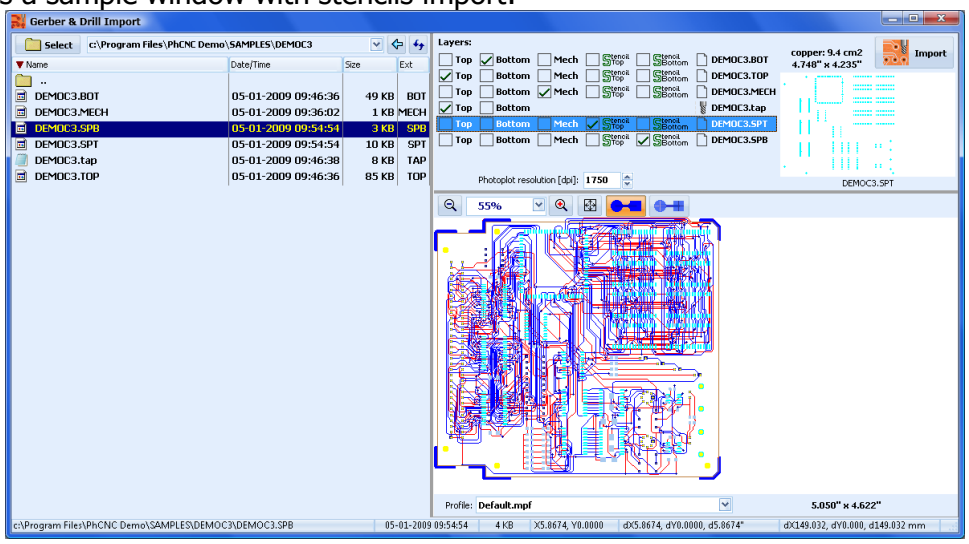
Entering wrong parameters in this window will definitely create a wrong interpretation of the Gerber file and will be visible in the graphic window. It is important to carefully check the image and its size and make sure that it is what you intended to have.
Shown below is the file after correct import:



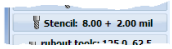
Important:
If the CAD system that you are using can export Gerber files in both RS 274-D and RS 274-X, you definitely have to use RS 274-X (Extended Gerber). This will save you time, efforts and reduce chances of errors.

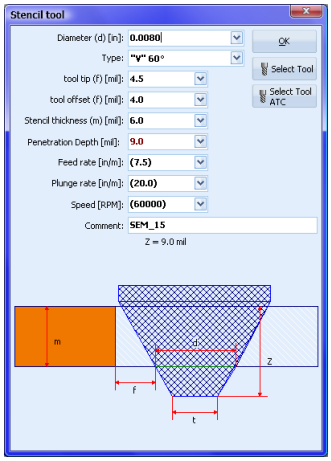
2.1.6 Gerber files for Stencils Import

PhCNC (versions v1.10.1 and up) are able to 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 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:

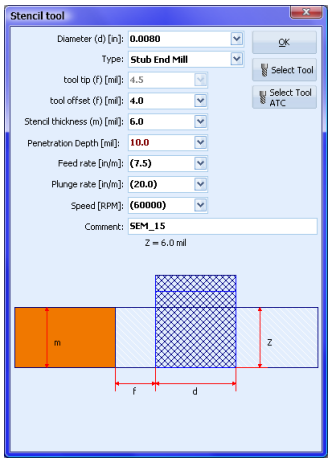


When stencil files are imported in the project, additional buttons will appear on the left side of CAM window shown below.





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.

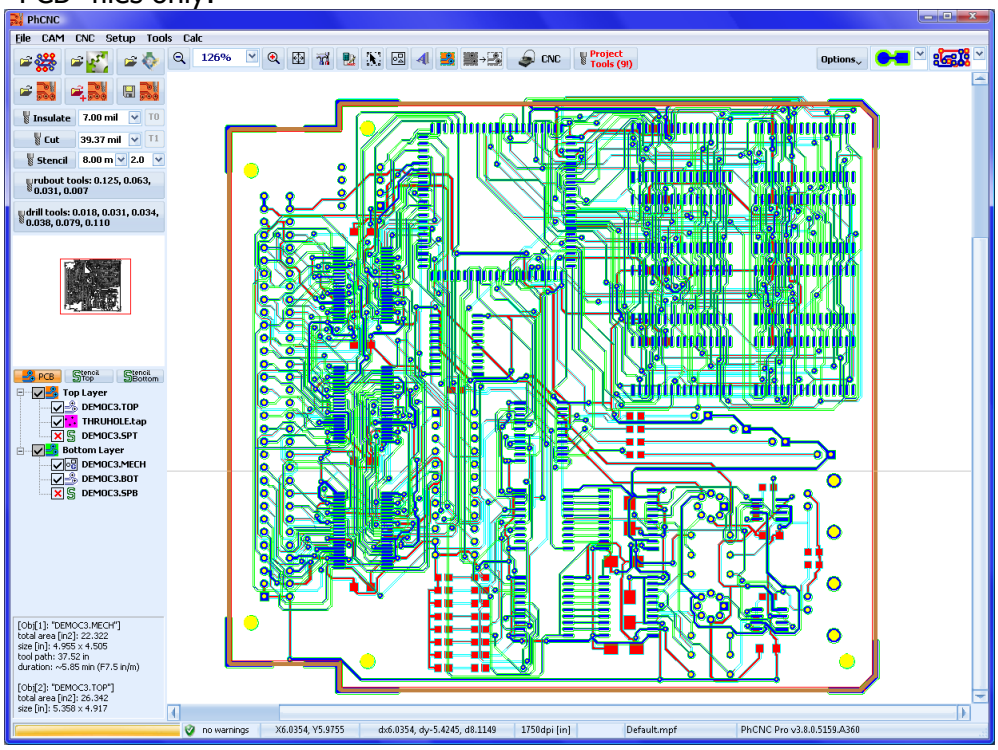
Tool penetration depth is calculated automatically by PhCNC and shown in the tool change window.
When using a "Stub End Mill" the depth is only recommended (should be more than material thickness).

2.1.6.1 Stencil layer(s) visualization

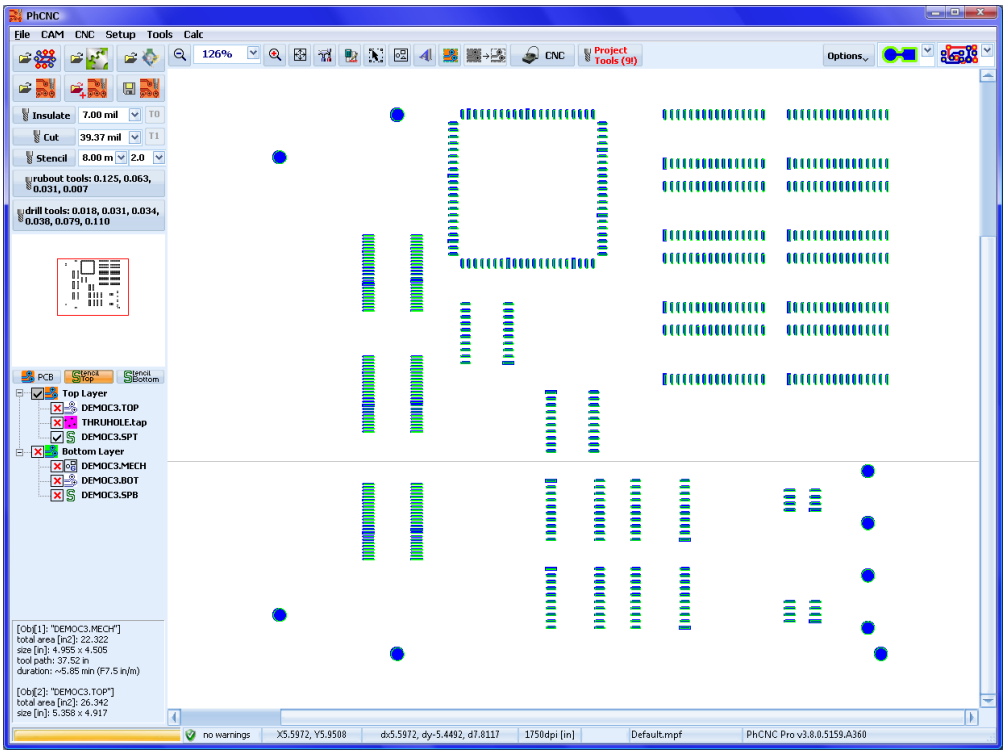
If a project containing stencil layer(s) has been imported, additional buttons that control the visualization of all layers will appear.

The "PCB" button, shows all files except the stencils
The "Stencil Top" & "Stencil Bottom" shows Top & Bottom stencil images

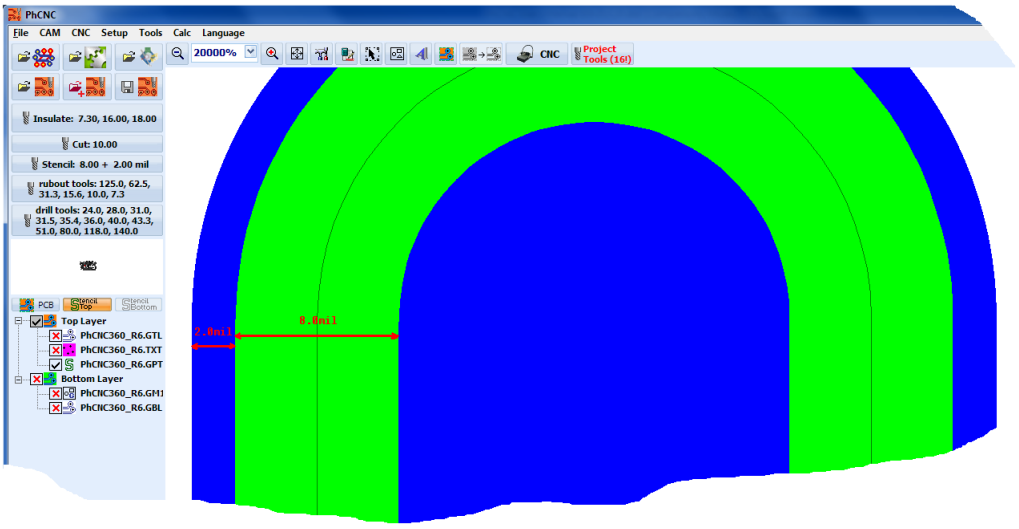
Shown below "PCB" files only:



Stencil Top:

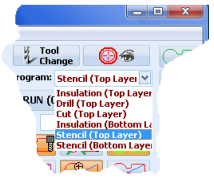


Below is an example of tool path dimensions and "safety zone" compensation



2.1.6.2 Stencils machining in CNC mode

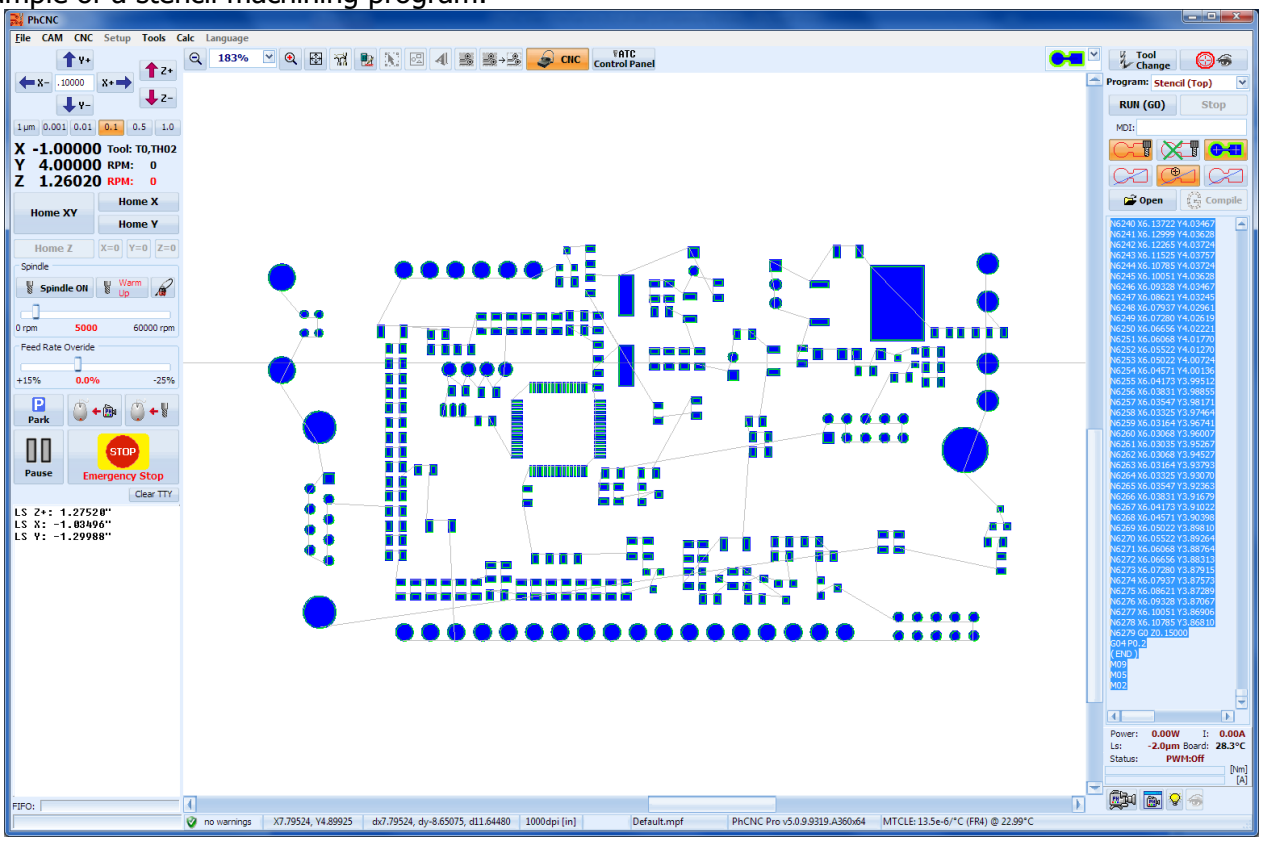
Entering the CNC mode the operator will see the programs for stencil machining:



Important:

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

Example of a stencil machining program:



Note:

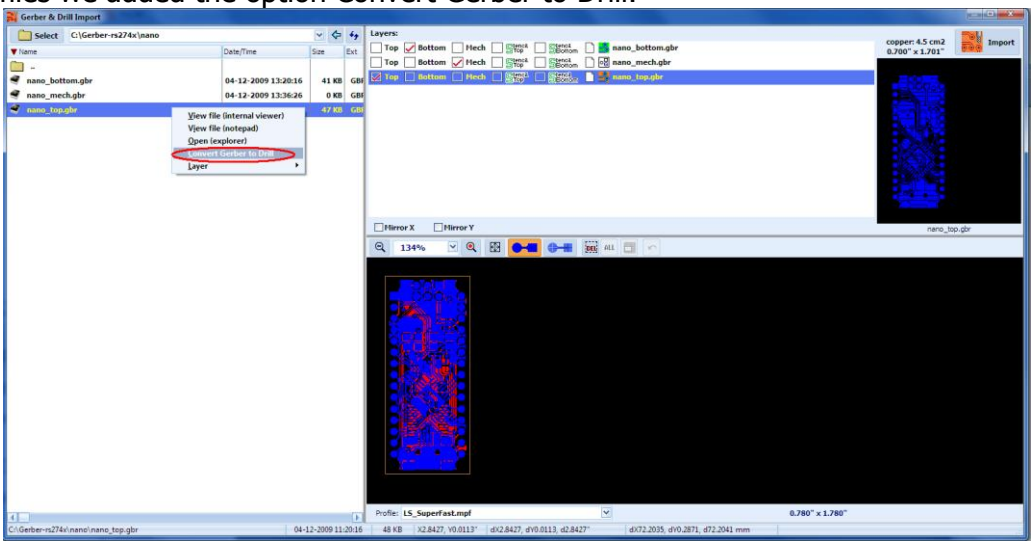
When you machine a PCB, your "bottom" files are mirrored along the X axis. Stencil machining does not mirror the "Stencil Bottom" file. Here the "Bottom" means for the bottom side of the board and does not need to be mirrored. "Top" and "Bottom" stencils are separate work pieces.

Important:

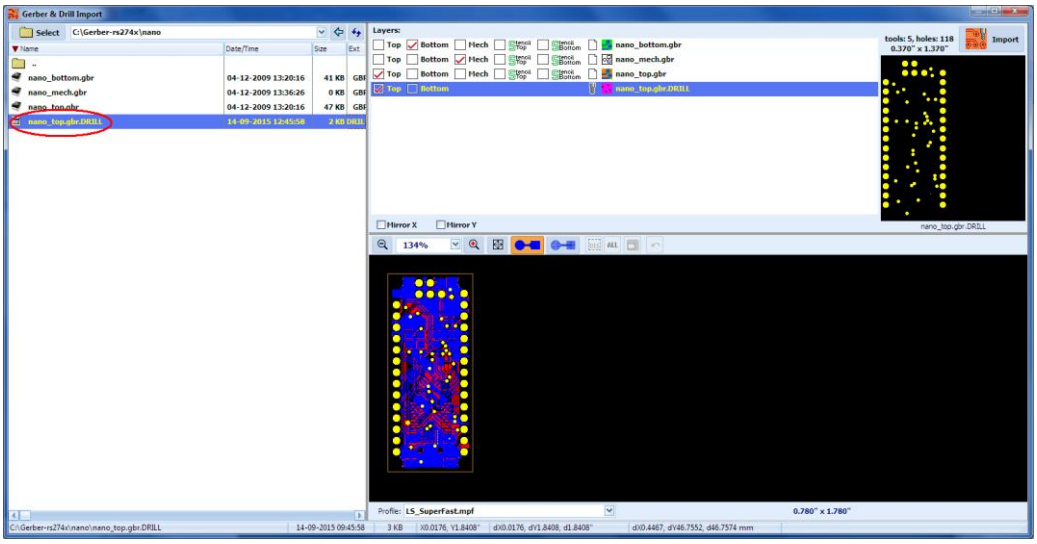
Due to significantly different properties of the materials used for stencils, there are separate settings for the machining in the "Setup" menu:

2.1.7 Convert Gerber to Drill

PhCNC 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 extention .DRILL. The new file then can be selected as a Drill file for the project.



IMPORTANT

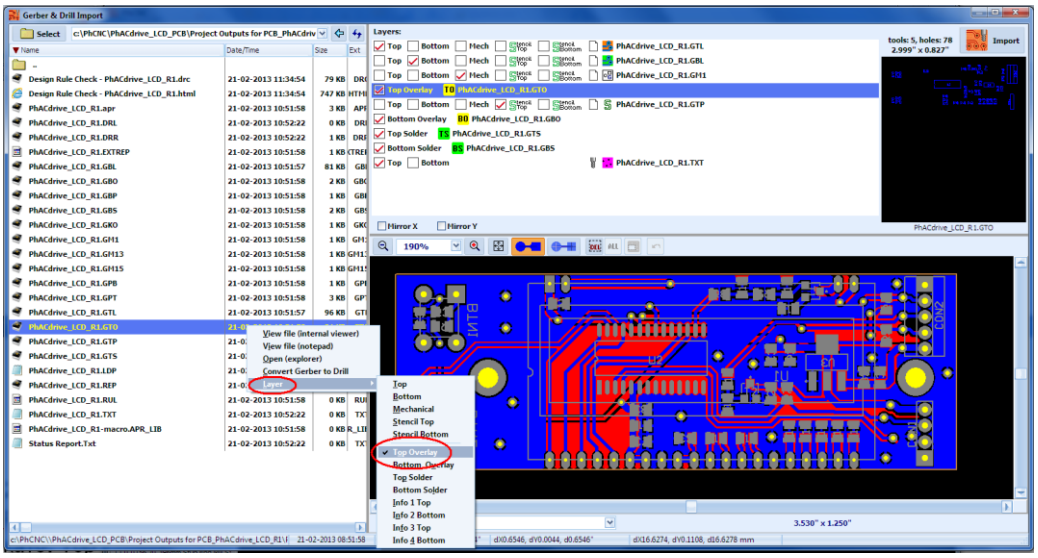
In order "Convert Gerber to Drill" to function properly the folder should allow to write files.

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 in order to be able to see all details.

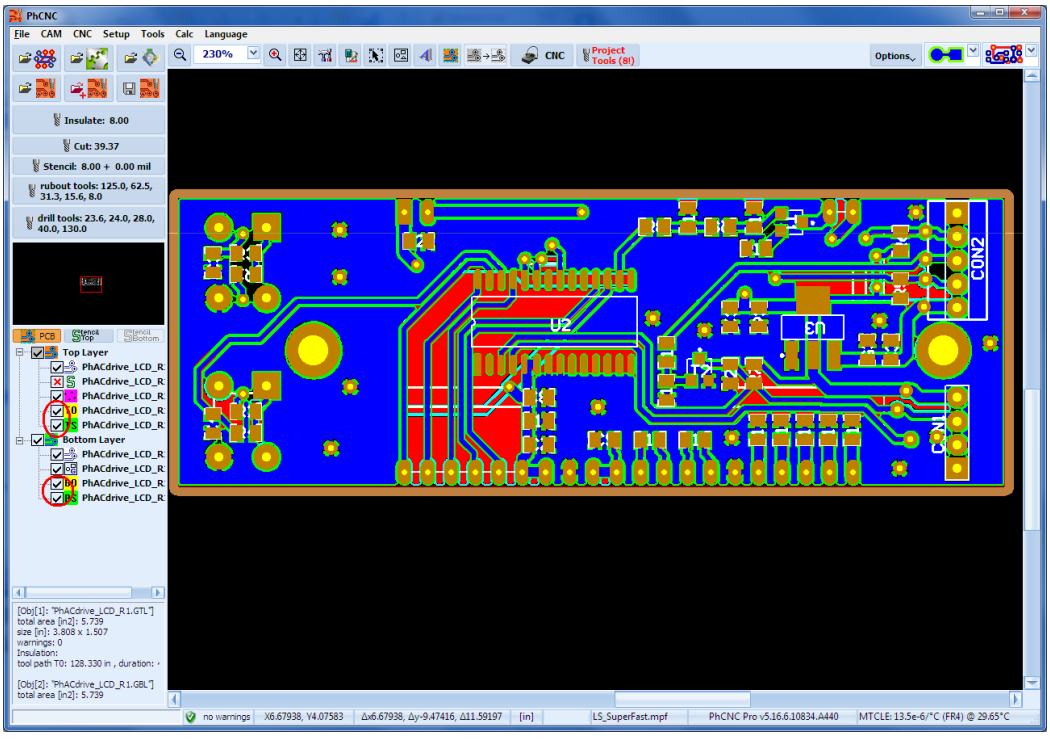
Convert Gerber to Drill - video:
https://youtu.be/U6KZnUt10Yo

2.1.8 Info layers import

PhCNC 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.

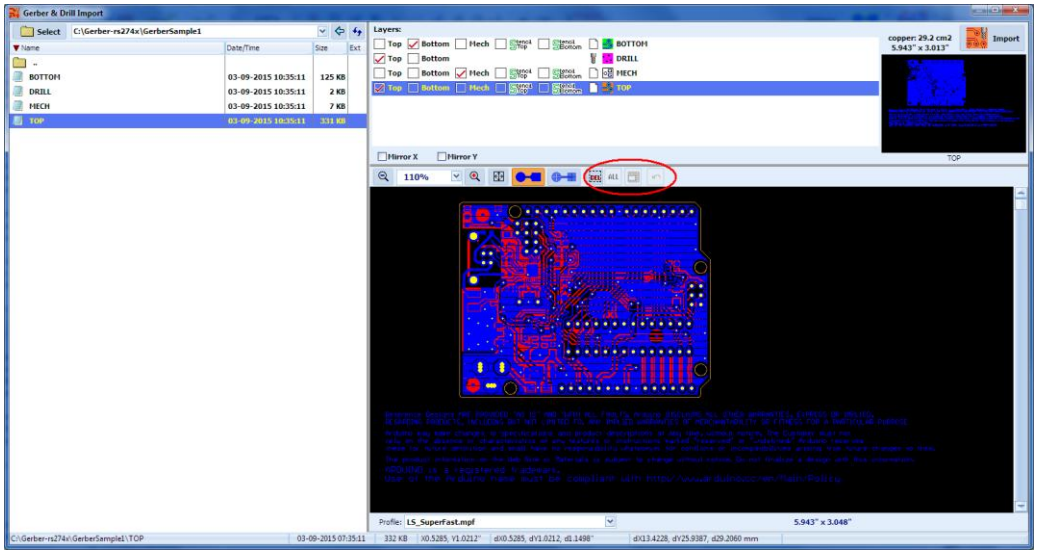


Layers for machining: "Top", "Bottom", "Drill", "Mechanical", "Stencil Top" (Top Paste), "Stencil Bottom" (Bottom Paste), also can be selected from popup menu.
After an import in PhCNC, info layers can be shown/hidden the same way as the layers for machining.

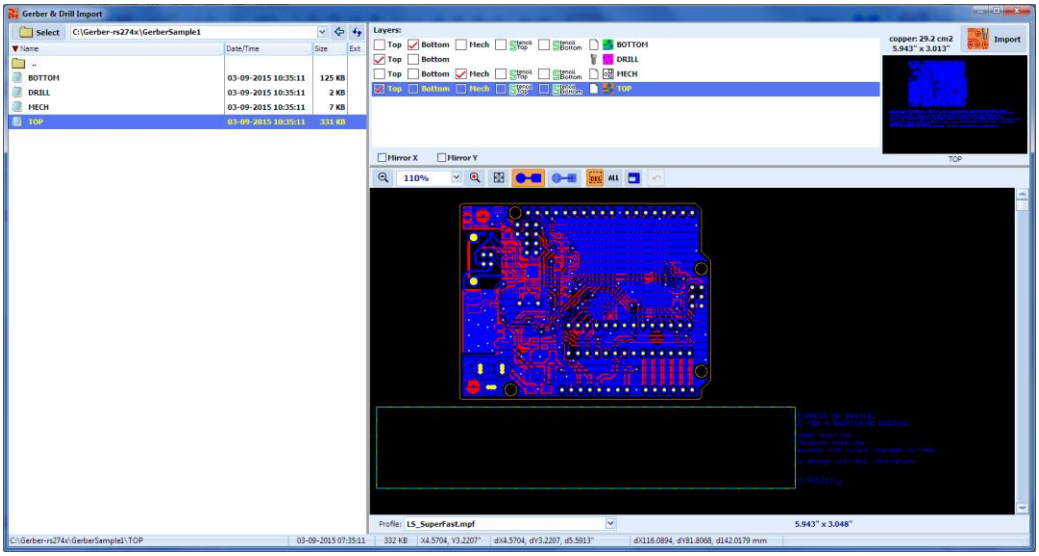


2.1.9 Gerber editor

PhCNC 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 DEL. 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. 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/JY9lpaCIgEg

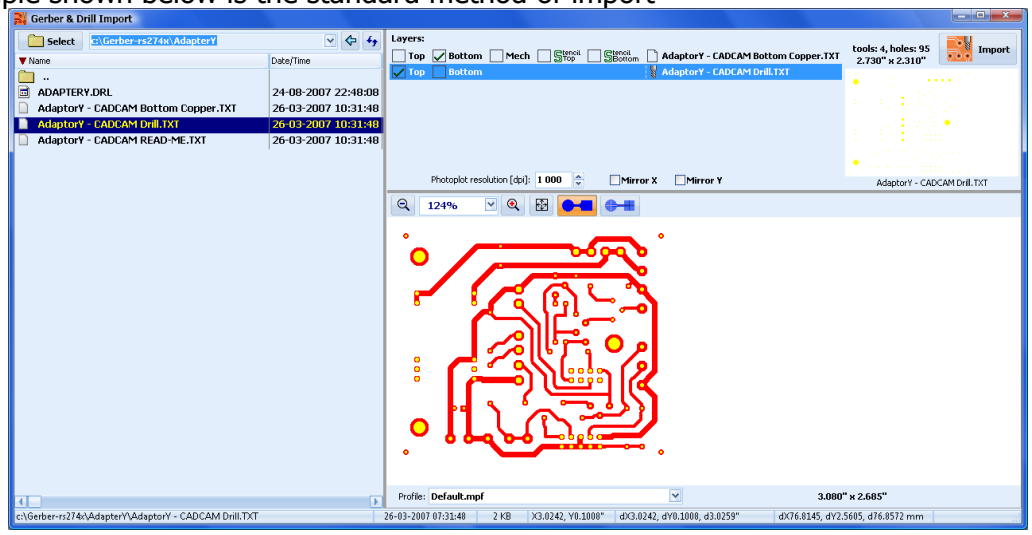
Gerber import example (editor) #2
https://youtu.be/4MSkDD8JGjw

Gerber import example (editor) #3
https://youtu.be/v7VtASRzN0I

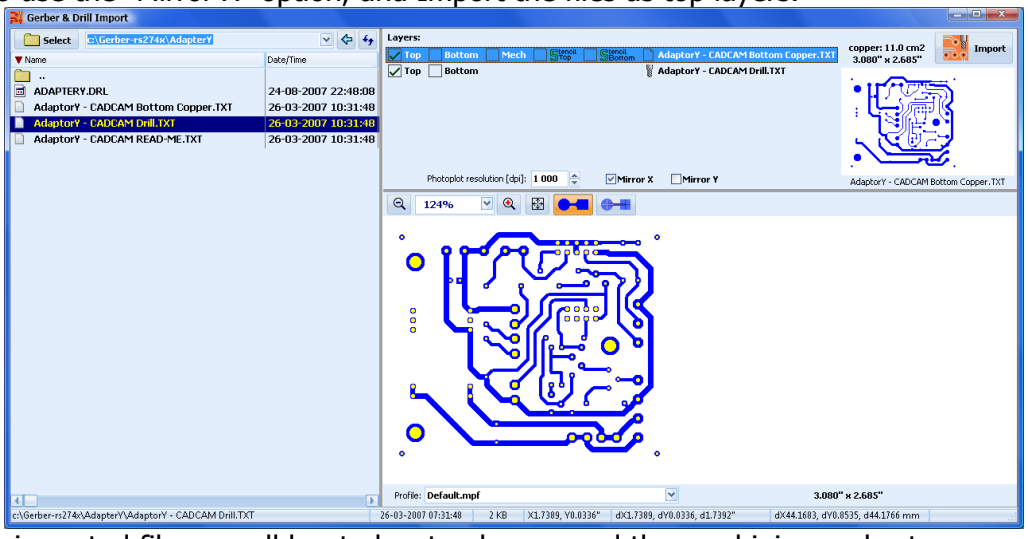
2.1.10 Single layer PCB import

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.



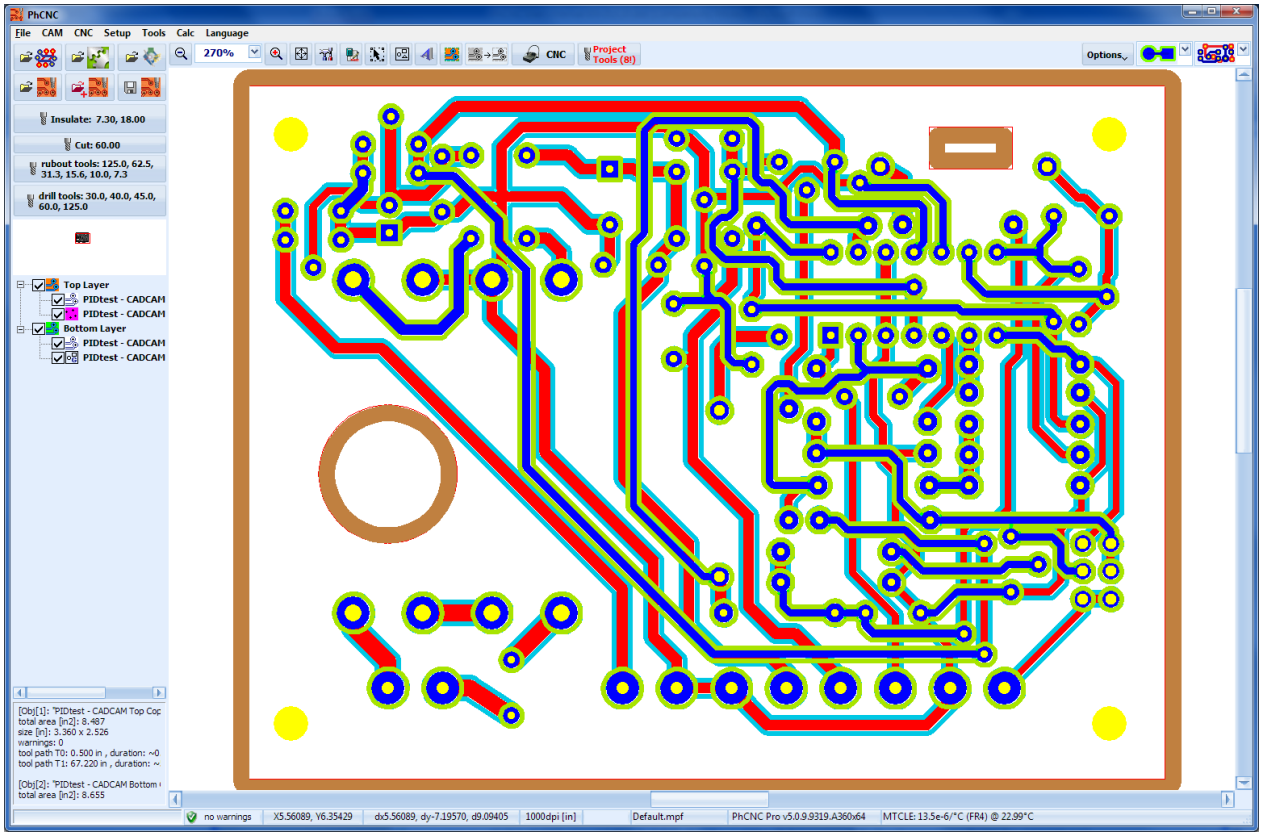
This way the imported files are all located as top layers, and the machining and setup are easier.

Loaded PCB projects will be placed at the center of the work-space.
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.



F_A1.PHJ

2.2 PhCNC Project Files

PhCNC 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.

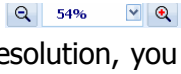
PhCNC supports two modes of loading .PHJ project files:

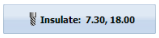
Button when pressed closes the current project (if it is not saved yet, you will receive a message asking if you want to save it) and opens the .PHJ file you select with all the tools for insulation, cut-out and rubout defined (including DPI resolution) in the .PHJ project that you open.

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.

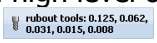
For your convenience, the "File" menu lists recently opened .PHJ projects (up to 8 files). Upon exiting PhCNC, you will be asked to save the current project (if it has been modified) and even if you answer "No" it will be saved to a special file called "Recent Project" available in the file menu automatically 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

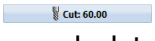
The Buttons & drop down area  is for Zoom In/Out or to set the zoom percentage. If your monitor is set for the correct resolution, you will have the correct size image at 100% zoom. It is very convenient to use a mouse with a wheel for zoom In/Out. The zoom is always centered to the mouse pointer.

The button 

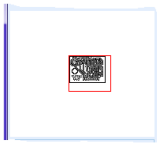
is a **Very important setting!** It is for the insulation tools, used in the calculation of the insulation tool paths. If you change insulation tools, PhCNC automatically recalculates and redraws all insulation tool paths.

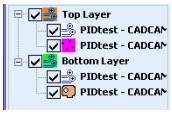
Important:
PhCNC insulation tool path calculations, **do not allow intrusion into the traces and pads**, if your tool diameter is bigger than the space between traces and/or pads they will be left non-insulated.
The standard version of PhCNC does not detect and display this problem (See 2.13 Design rules 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 

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

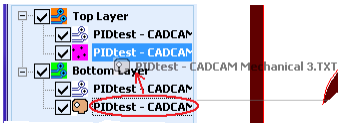
Window 
is the panoramic window it shows all objects placed within the machine workspace. The Panoramic Window is a convenient way to view your entire project. You can left click inside this window, to view the desired portion of the workspace. This is convenient if you do not have a mouse wheel.

Window 

is the loaded files window it shows all the loaded files and the layer they are on. Clicking on the check-mark ☑ 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.

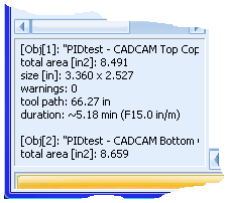
Important:
Disabled layers are not manipulated by the CAM edit functions. For example, if you like to select a rubout area in one of the layers, you can disable (hide) the other ones. If you need to move a file from one layer to another click on file name, hold and drag.



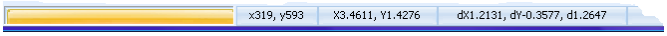
In the example above, the Mechanical layer is being moved from the Bottom to the Top Layer.

File type symbols:

- - Gerber file
- - Drill file
- - Mechanical layer ("PhCNC 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.



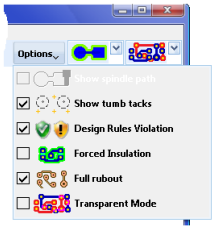
This is the status bar it contains the following fields in the order listed below:

- 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.

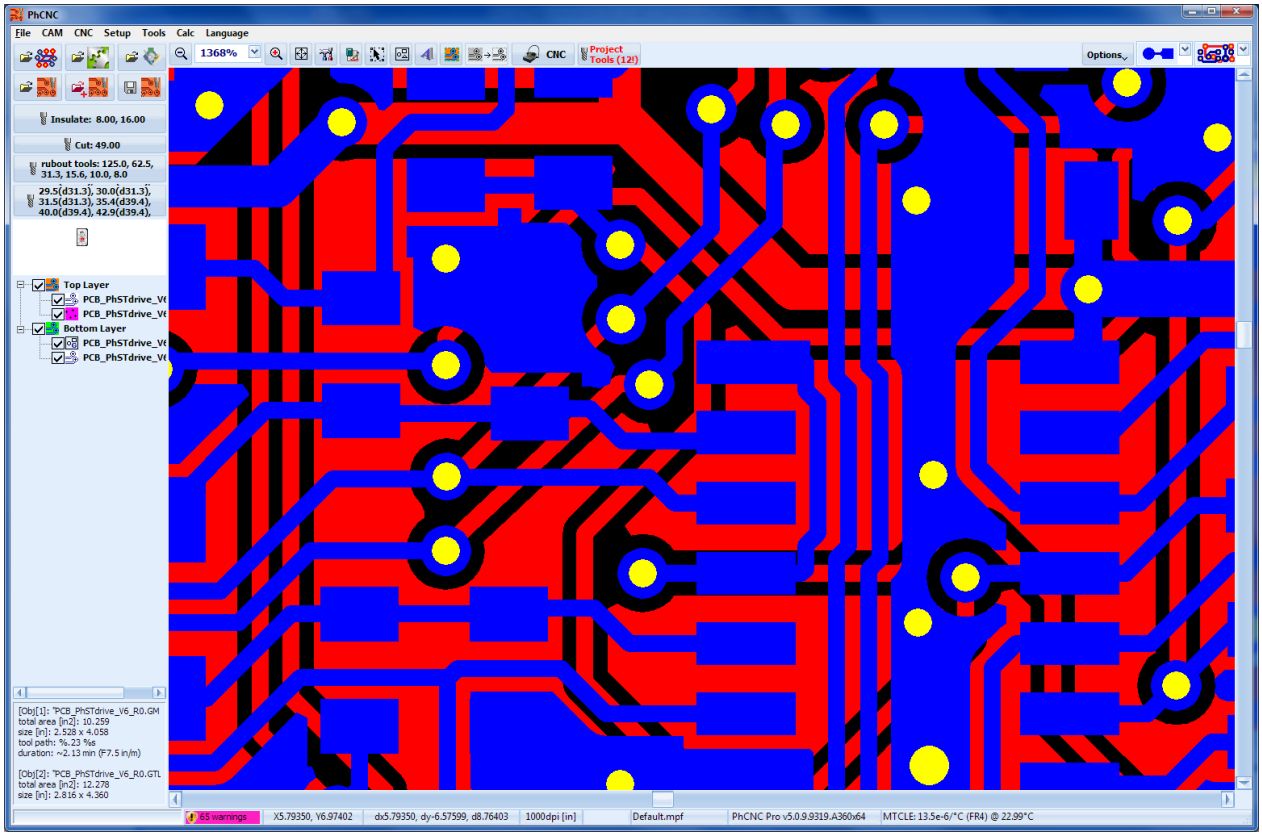
Drop down is for controlling the way objects are displayed on the screen and will give you the following choices:

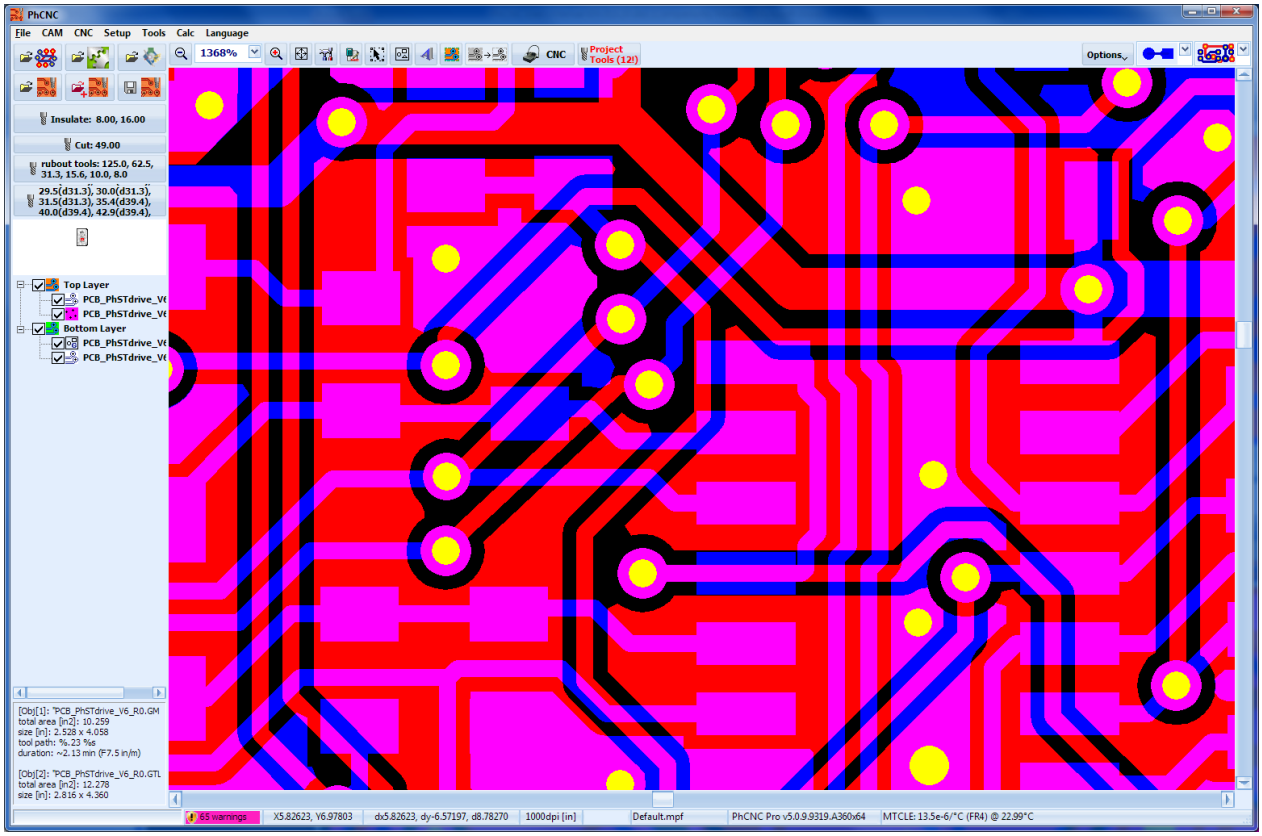
Selecting image will display the tool path center line only
Selecting image will display the tool path & real thickness
Selecting image will display the tool path & real thickness & center line
Selecting image will display the tool path & real thickness & Gerber
Selecting image will display the tool 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:



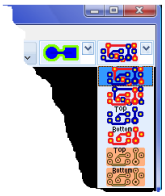
- Show spindle path this PhCNC Pro option will show/hide spindle positioning lines generated in CNC mode. Active when you return from CNC mode. Useful for positioning objects on the desktop.
- "Show thumb tacks" - this **PhCNC Pro** option will show / hide thumbtacks and is discussed in detail in section 2.14
- "Design rules violation" - this **PhCNC Pro** option will activate/deactivate the DRV and is discussed in detail in section 2.13
- "Forced Insulation" - this **PhCNC Pro (1.16.1+)** 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 Pro** option will activate/deactivate the full rubout and is discussed in detail section 2.15
- "Transparent Mode" - this PhCNC Pro 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. Bellow 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.

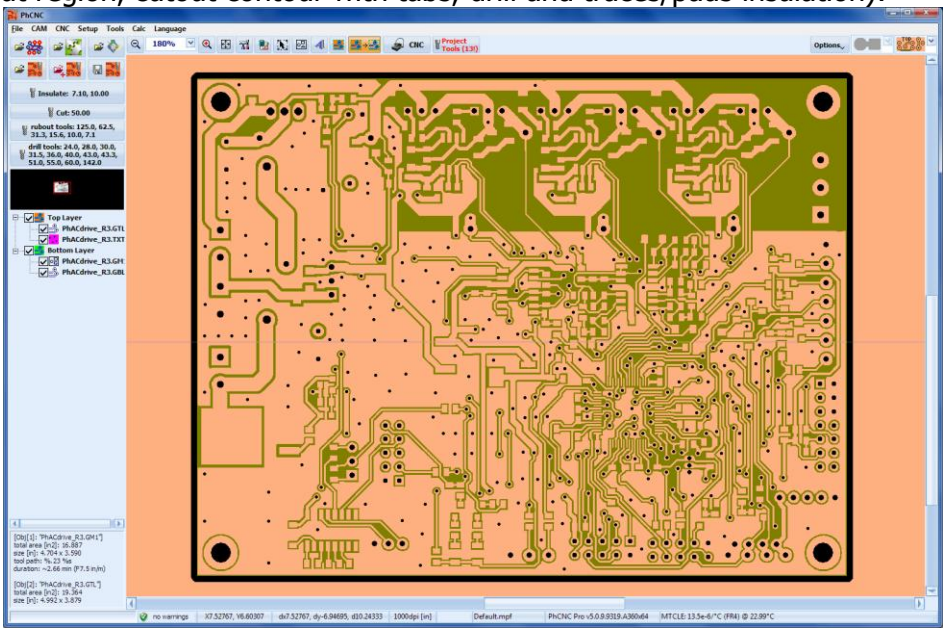
Buttons & are for a special visualization mode called "copper view". In this mode the screen shows what should be the final result after executing all machining programs. It uses close to real colors to show copper and typical base material. In CNC mode of PhCNC these buttons are inactive!
Buttons: ,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

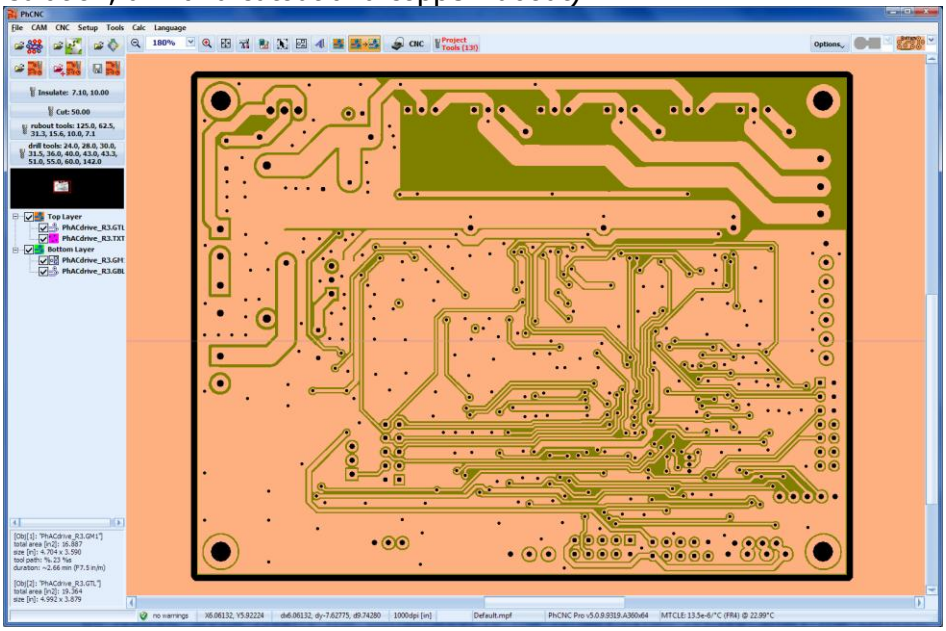
by default is active.
In CNC mode of PhCNC all these buttons are inactive!

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):



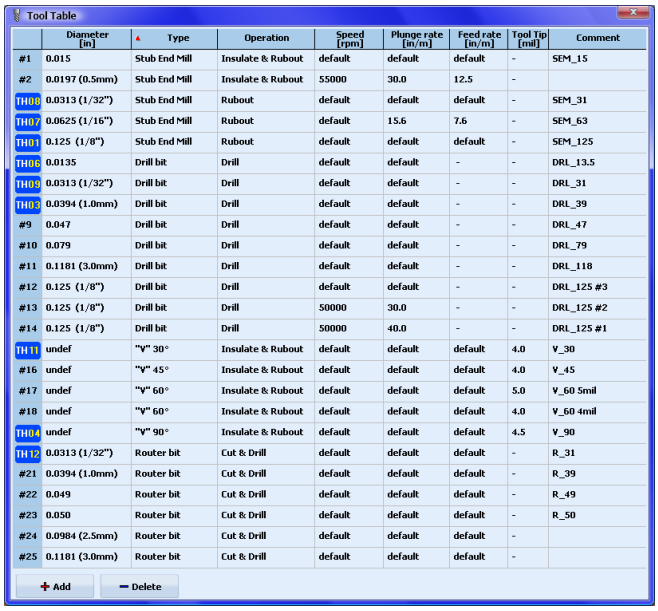
All buttons above control the visualization of the project. They do not affect the settings or the machining process in CNC mode.
The following Quick access tool bar buttons are accessible on the tool bar:

Button is for enabling the panoramic view window
Button is for the setup of PhCNC
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 CNC is to switch between CAM/CNC modes
Button Project Tools is to enter the project tools list

2.4 Tools

In order to make the operation of PhCNC 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]

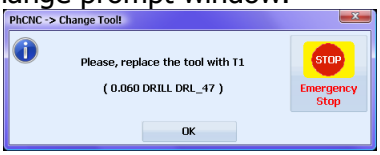
To edit the main tool table click on "Tools" in the upper tool bar and select tool table from the drop down. The tool table will then open as a window as shown below:



Buttons + Add & − Delete 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.

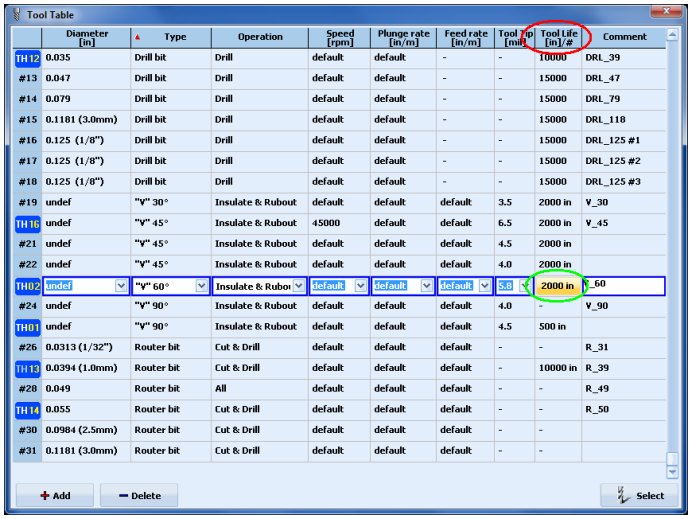


- **Tool Life [in]/#** is a field that shows maximum tool usage acquired by the manufacturer specs or by own experience

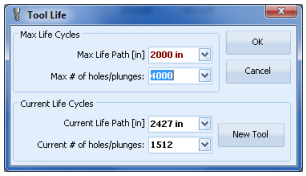
In this column of the tool table (**Tool Life [in]/#)** you can enter a value that if exceeded will prompt you to change tool. It can contain the information below:

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

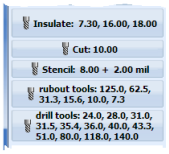
It can be set for every tool (if needed) and will prompt the operator that the limit set is exceeded. PhCNC records both (total number of material entries and total length of cutting) for routers and end mils. It records only the number of drill cycles for the drill tools.
Example:



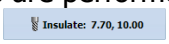
Shown above is the column in the tool table that handle the tool life (in red circle).
Clicking on the column for any tool (in green circle) will allow you to enter needed parameters in the window below. Holes/plunges represents number of entries in the material. Current tool usage is listed in the lower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

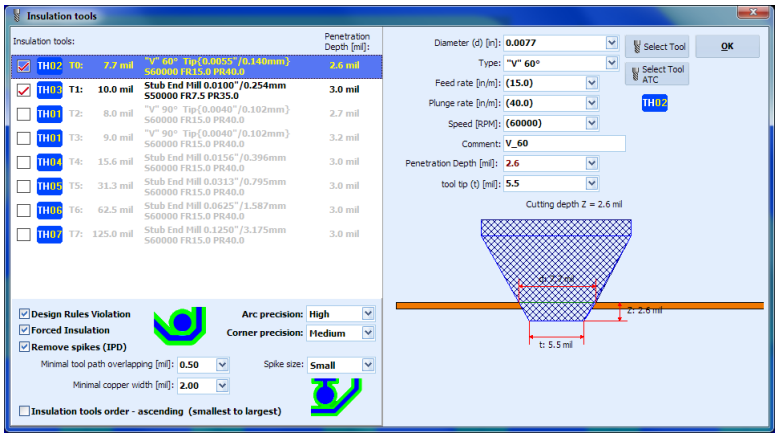


Important:
You may sort out the table by clicking on the column title.
("▼" shows which column is selected for sorting order).
Second click on the column title changes the order of sorting (ascending to descending and back).
In the left portion of the screen (PhCNC in CAM mode) you will find buttons for tools and tool tables for each operation.



2.4.1 "Insulate" Tools

PhCNC (v5 and up) 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 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 system recognizes this and will use the same tool (in ATC models).
The number of tools that will be used depend only of 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 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 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".

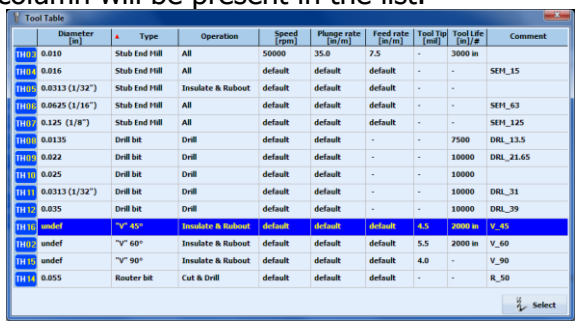
- **Remove Spikes (IPD)**

"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.

Buttons Select Tool & Select Tool ATC allow the operator to select a tool from main tools table or from tools already in the tool holders (ATC models only). For ATC models it is desirable to select tools from the existing ones (in tool holders). If this is impossible for any reason, PhCNC will stop the operation and require manual tool change with the appropriate tool.

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.

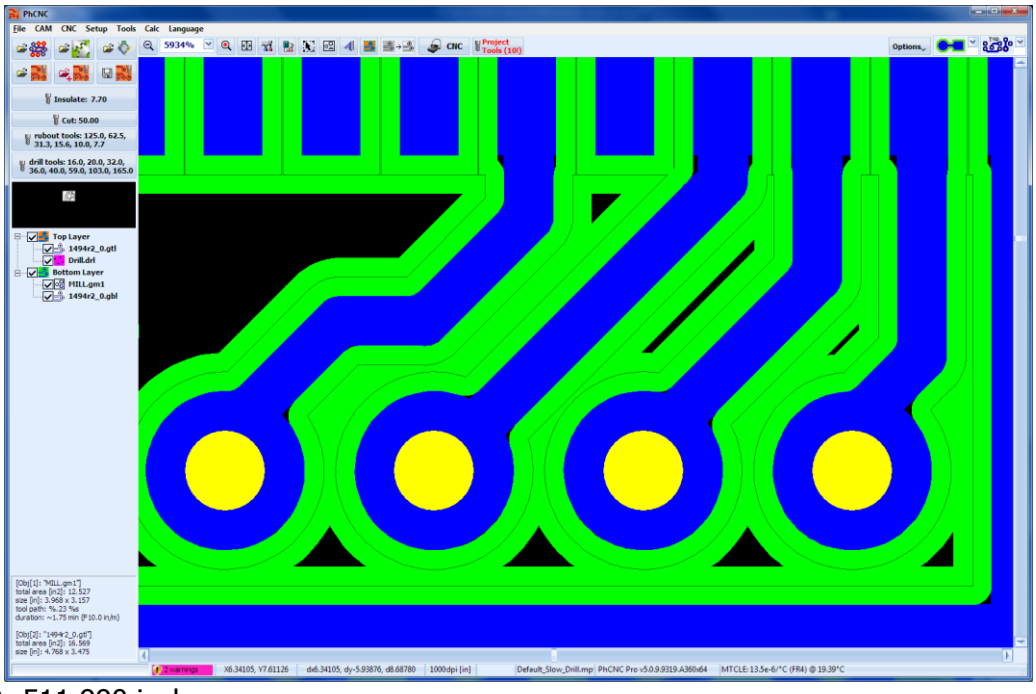


The information contains actual parameters (If "default" is selected you will see the value from the global table). These are the parameters that will be used for the program generation for this process.

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

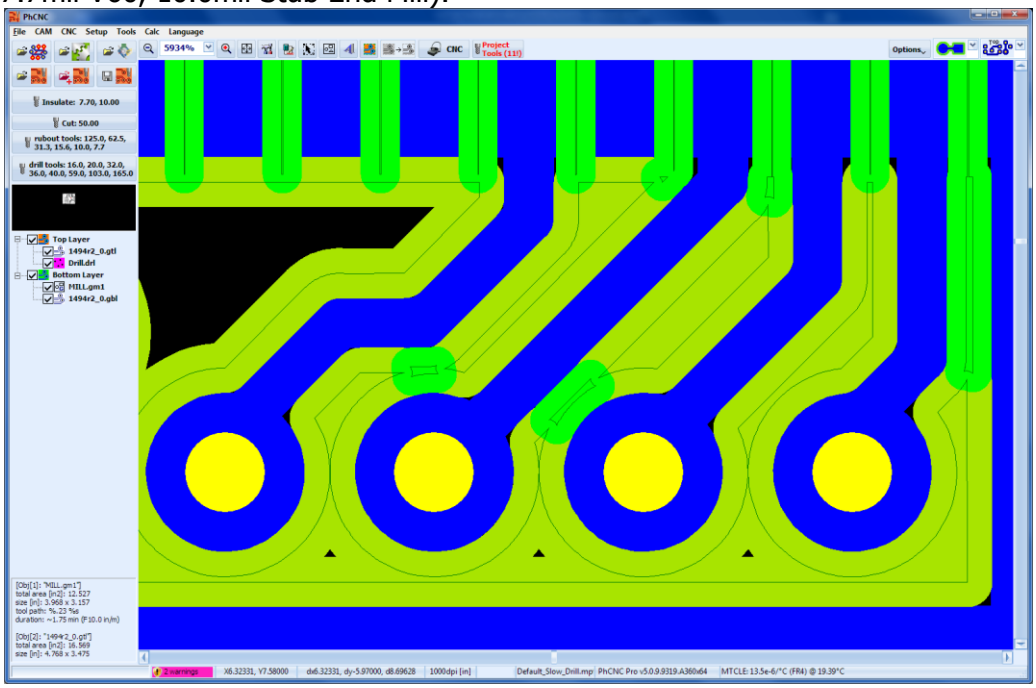
As we mentioned above a PCB design can be insulated with one two or more tools. Shown below is a single design machined with 1,2 and 3 tools

Single tool (7.7 mil V60):



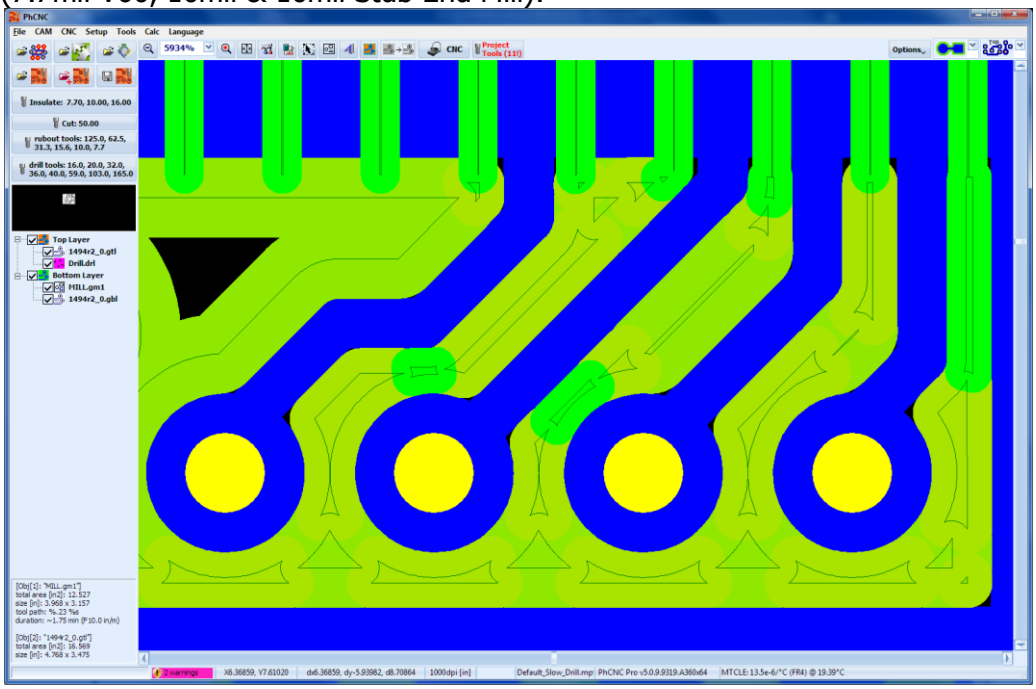
tool path T0: 511.990 inch

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):

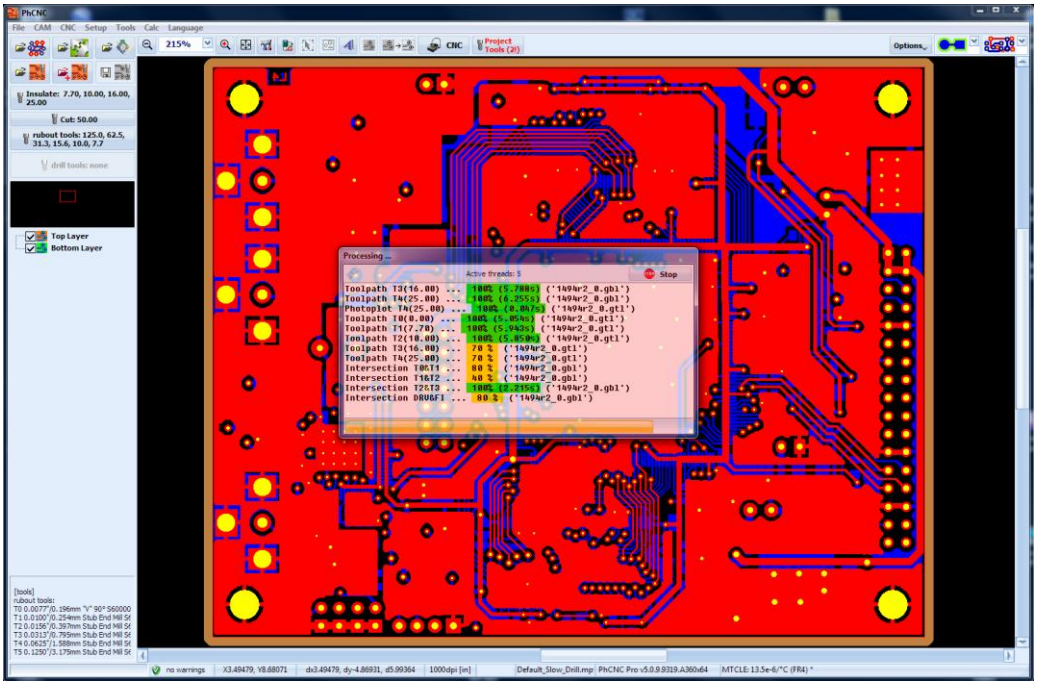


tool path T0: 114.540 inch
tool path T1: 177.600 inch
tool path T2: 389.940 inch

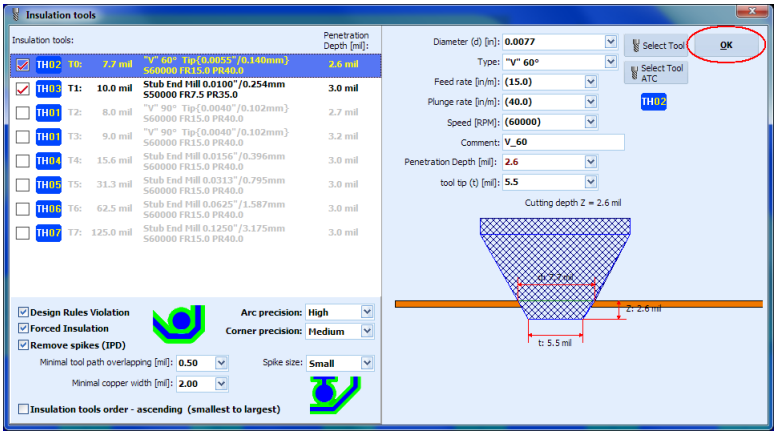
Important:

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.



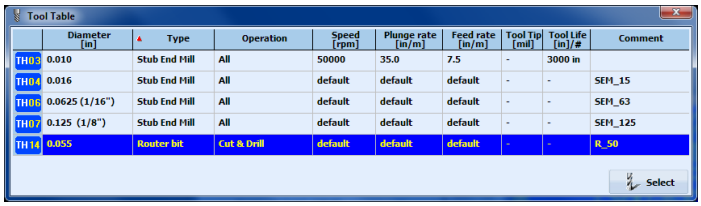
In case of interrupted tool paths calculations they can be resumed by pressing "OK" button in "Insulation tools" window.



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.

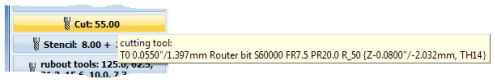
2.4.2 "Cut" Tool

To select the cutout tool left click on the Cut: 60.00 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 Select button.



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.

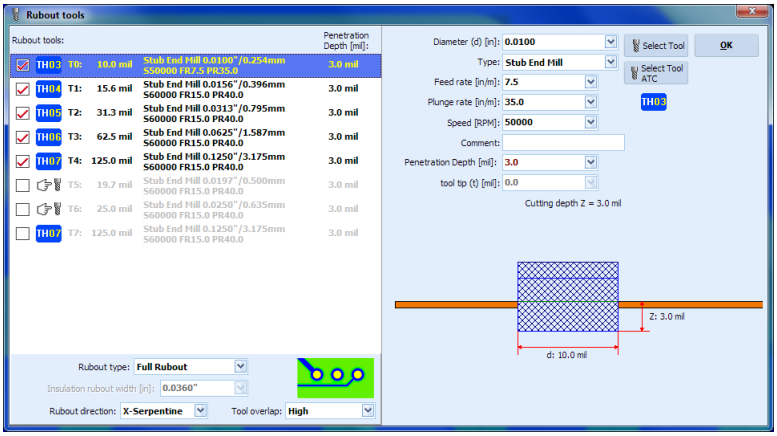
If you place the mouse pointer on the tool button, you will see the information for the selected tool in a popup window.



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 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 of 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 rubout tools: 125.0, 62.5, 31.3, 15.6, 10.0, 7.7 button.



Important: In the table above you have to enter the desired copper rubout tools. PhCNC 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 rubout tools: 125.0, 62.5, 31.3, 15.6, 10.0, 7.7 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 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 posible).

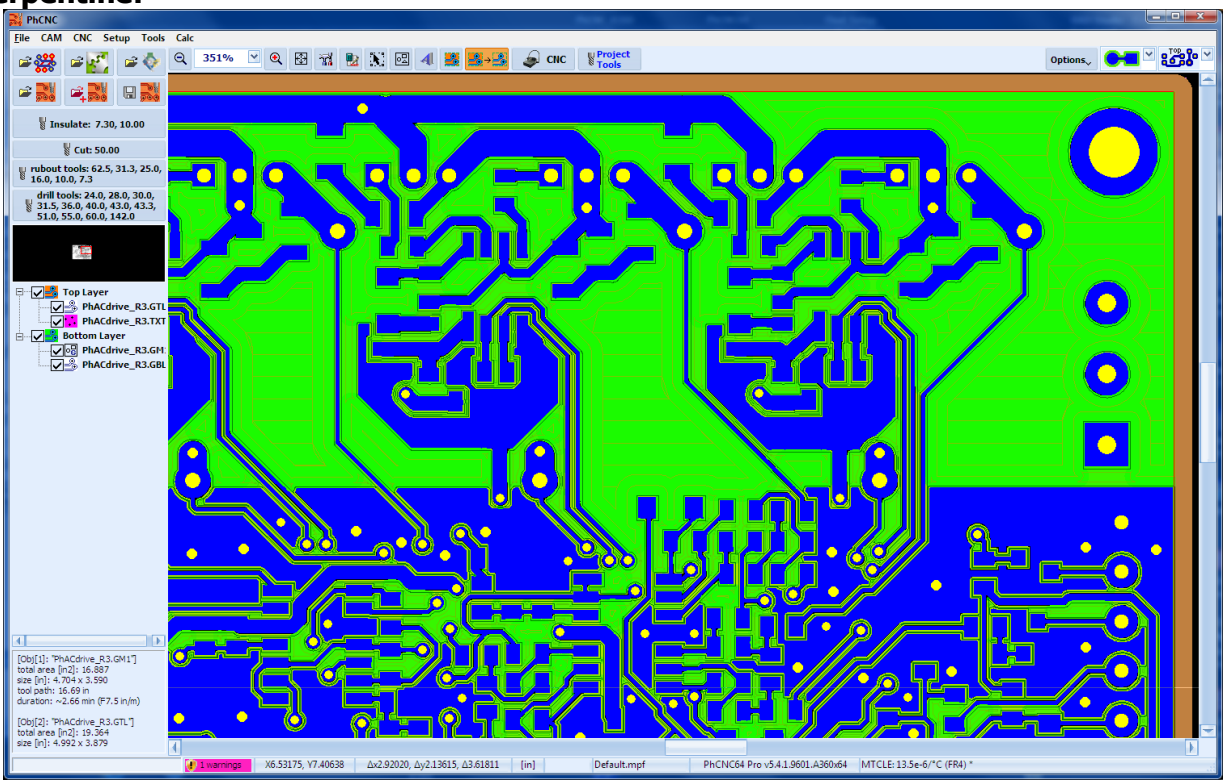
Rubout process starts with the smallest tool in ascending diameter order. This process is usually next after Insulation and this order saves one tool change.

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.

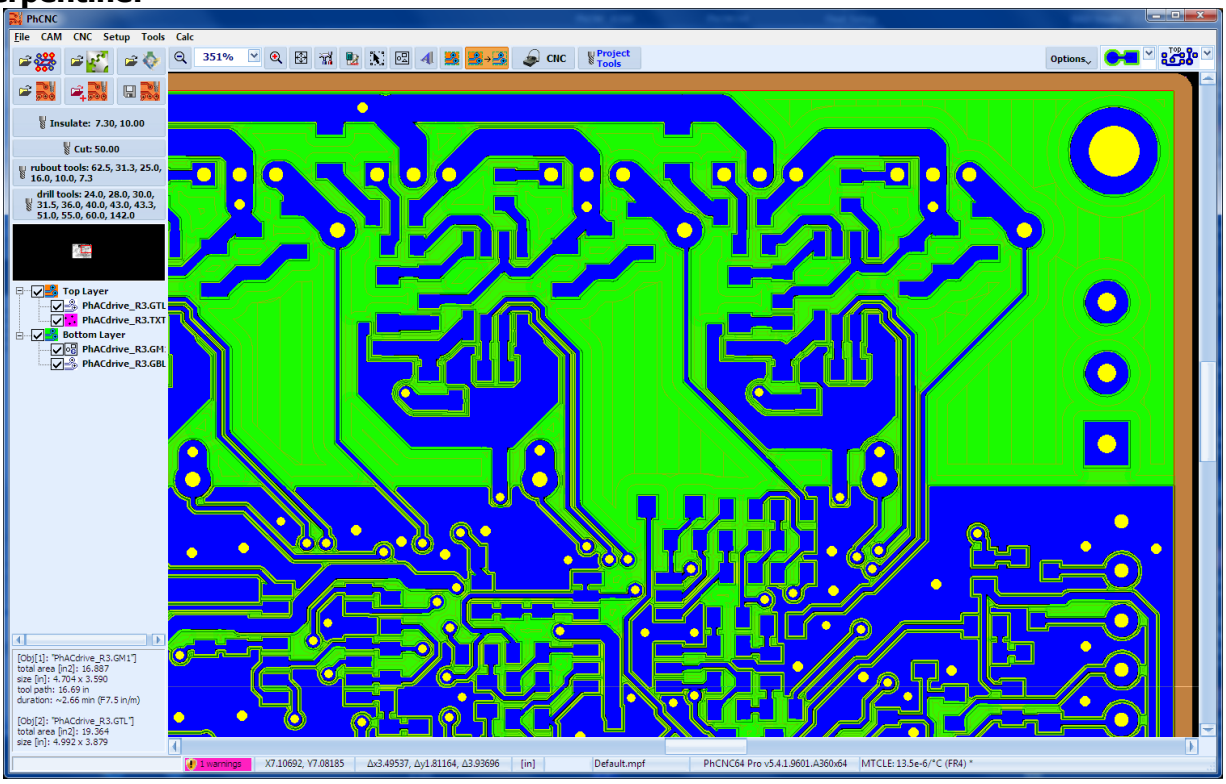
Important:
When Full rubout is selected it is not needed to select the option Remove spikes. Full rubout covers Remove spikes process. If you change your mind after performing Remove spikes this will not create any problem. Simply the time used for the Remove spikes will be a wasted time.

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

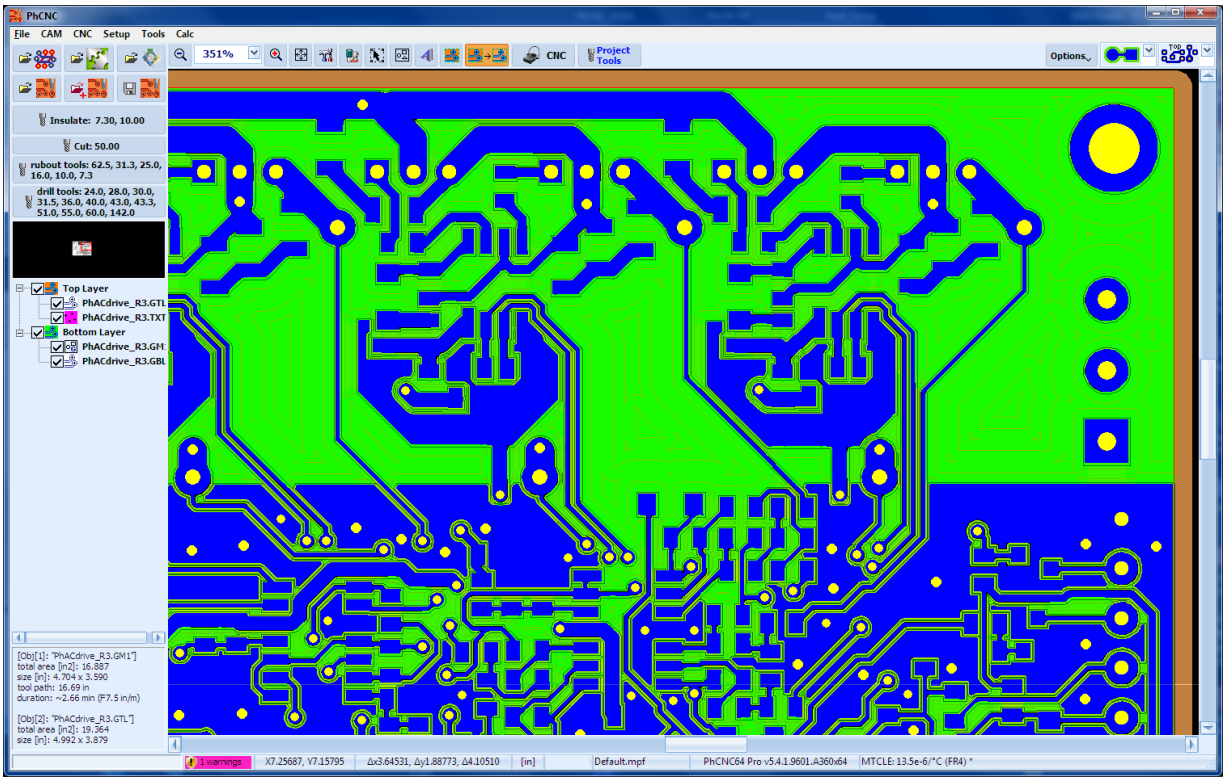
X-Serpentine:



Y-Serpentine:



Conical:

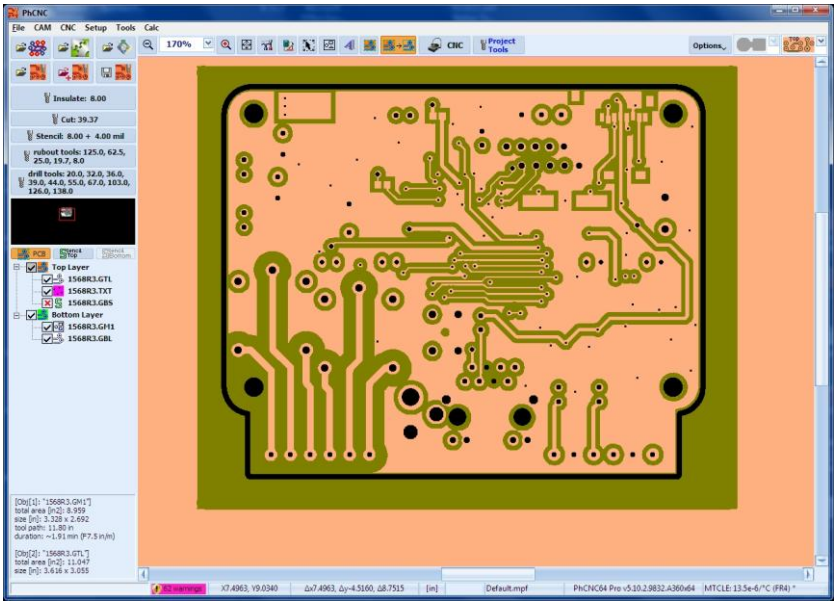


Notes:

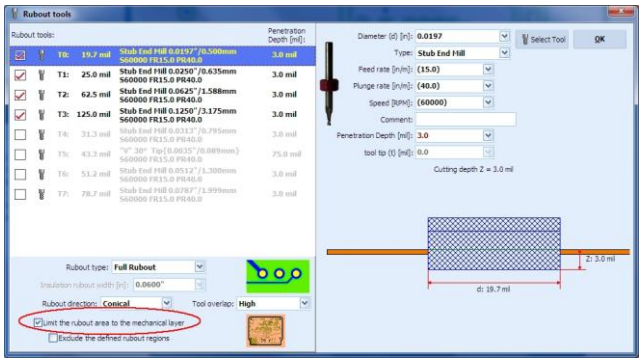
"X-Serpentine" and "Y-Serpentine" are recommended for usage in PCB projects that contain significant surface of copper to be removed.
"Conical" is more efficient for designs that have "poured copper" or when "Insulation rubout" has been used.

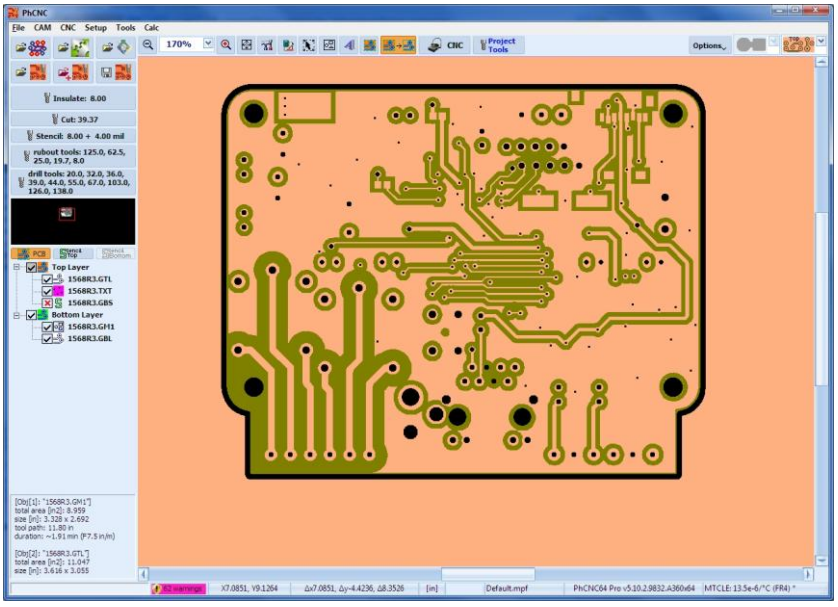
The options below **"Limit the rubout area to the mechanical layer"** Allows to use the area defined by the "mechanical layer" as a "copper rubout" area. Very convenient option for boards with more complicated mechanical shapes and internal cuts.

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



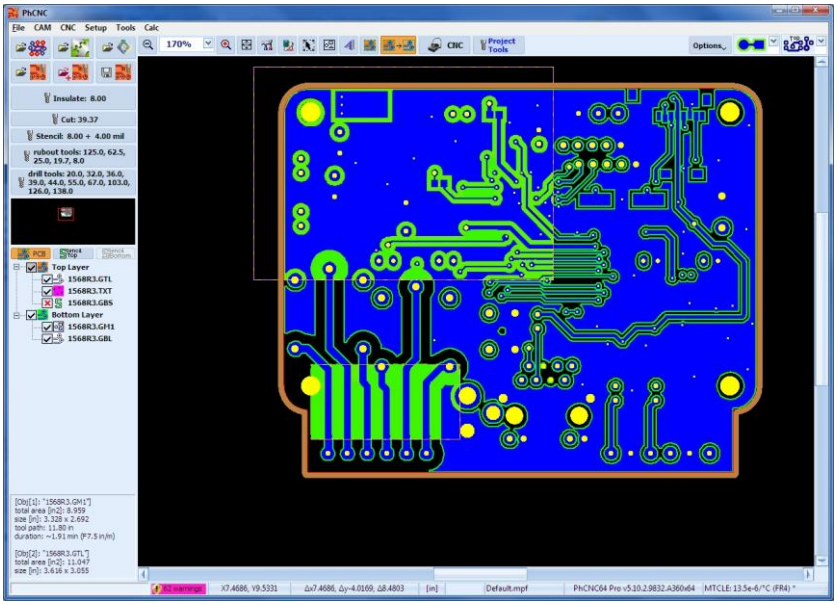
When "Limit the rubout area to the mechanical layer" is checked, the mechamical layer is used like rubout region:



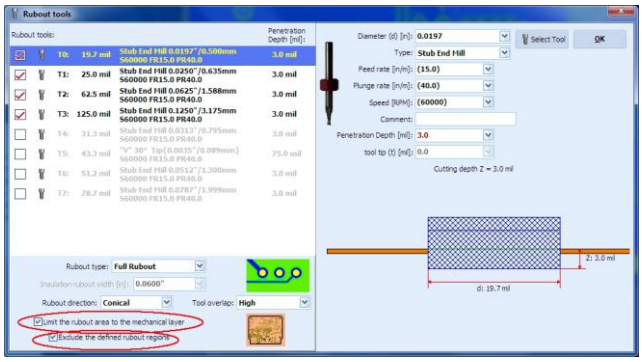


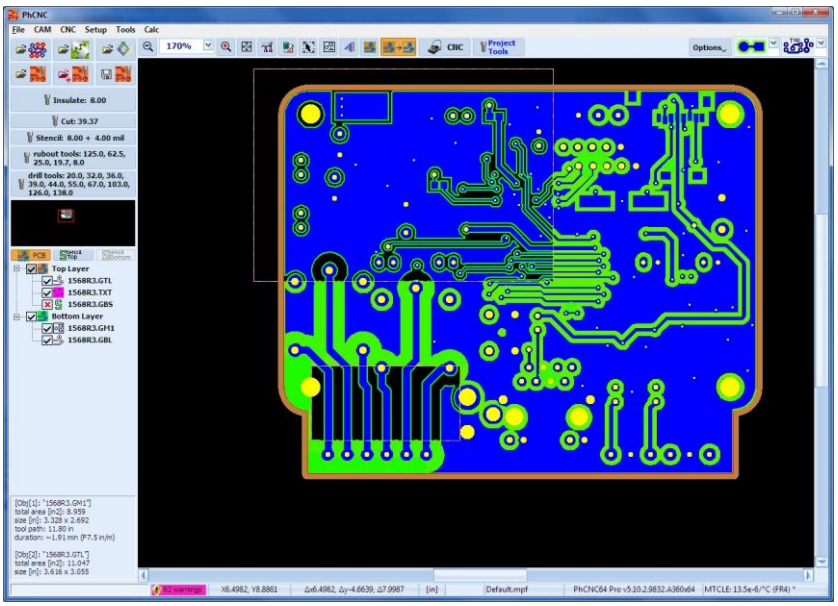
The option "Exclude the defined rubout regions" has meaning, only when "Limit the rubout area to the mechanical layer" is checked.

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:

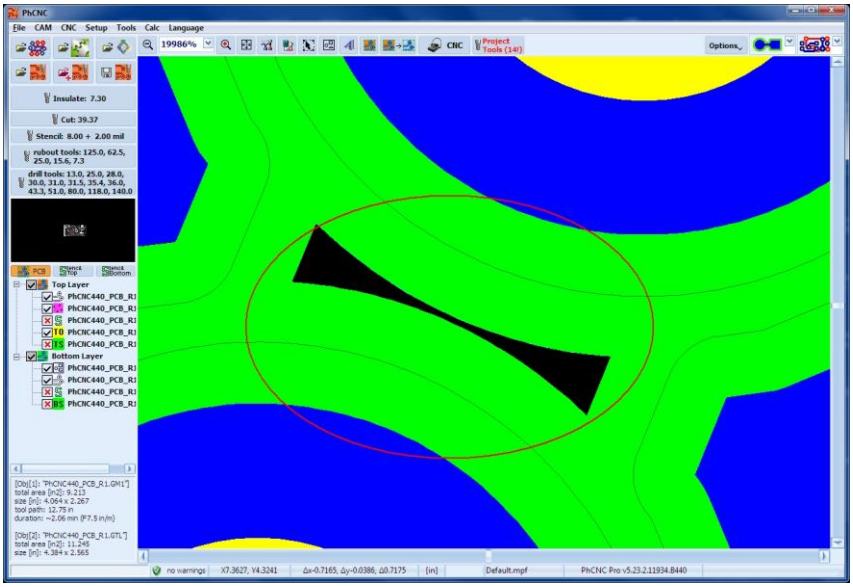




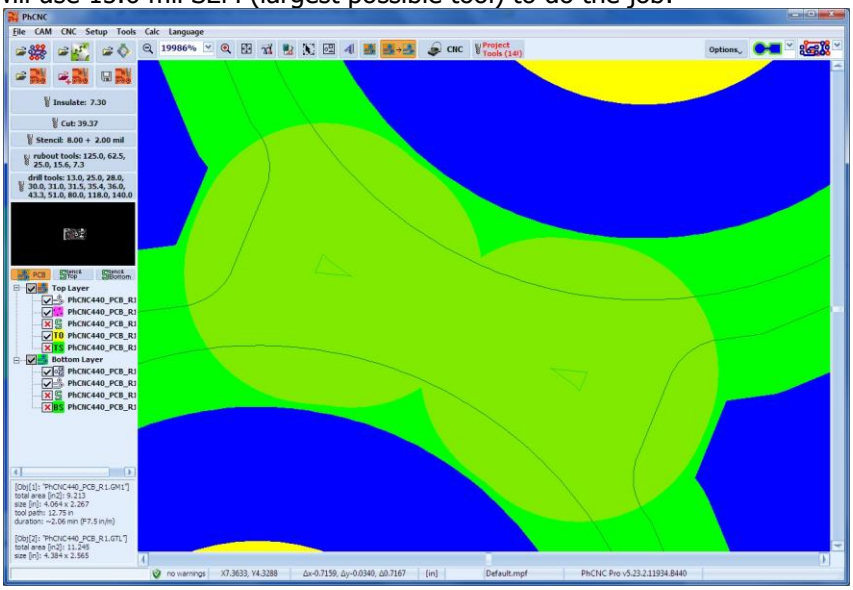
"Fish-tail SEM optimizations" menu option

PhCNC 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:

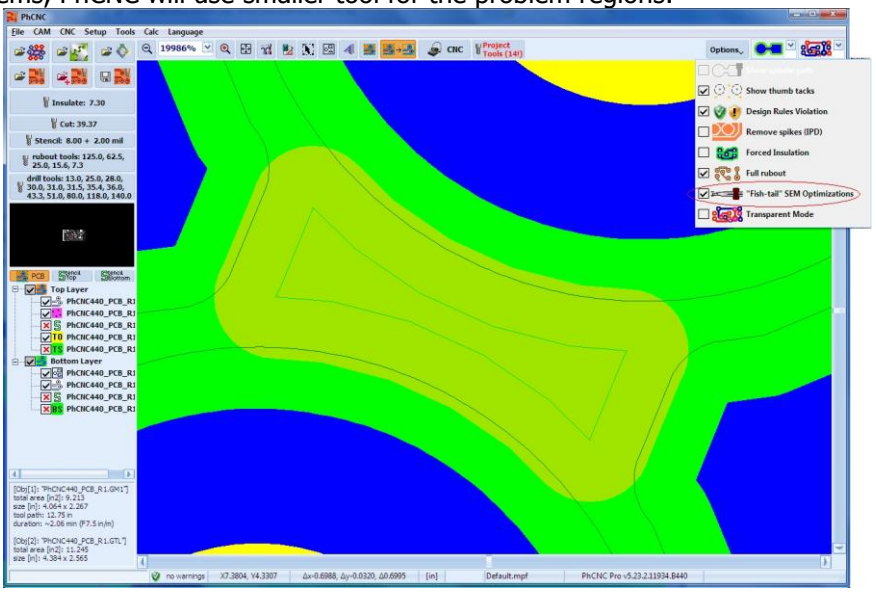


In this case, PhCNC will use 15.6 mil SEM (largest possible tool) to do the job:



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).

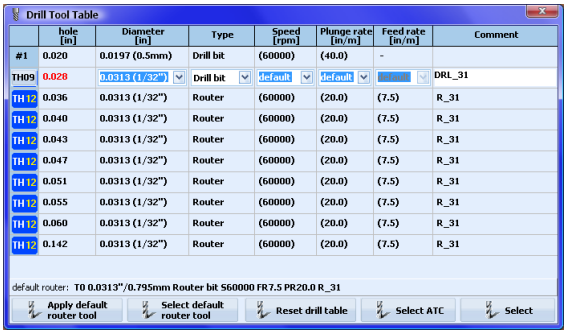
To avoid such problems, PhCNC will use smaller tool for the problem regions:



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 drill tools: 0.020, 0.032(D0.047), 0.040, 0.048, 0.060(R0.050), 0.120(R0.050).



The column "hole [in]" can NOT be edited; it contains the hole sizes from your CAD system (usually Excellon2 format file).
You have to 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 then the hole. The 2nd tool in the tool 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.

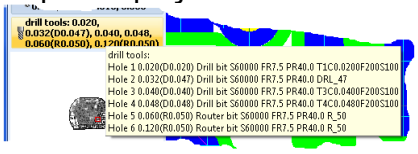
The new drill table contains a few new buttons related to usage of router tool for drilling of larger holes. If your project has a small number of larger holes it is more efficient to drill them using router tool, thus saving tool expenses and tool changes during drill operations.

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

Button Apply default router tool 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.

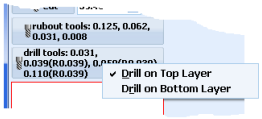
Button Reset drill table resets all tool selections.

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.



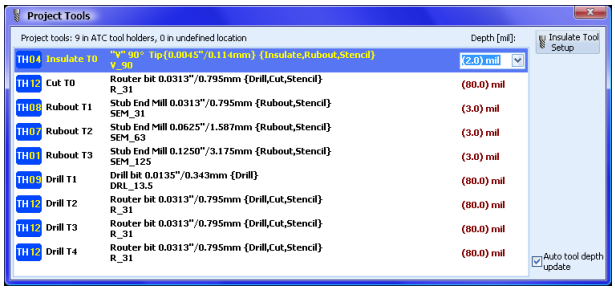
2.4.5 "Stencil" Tool (Pro version only)

"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

In order to make the reading and understanding of this manual better, we will call the positions that tools are placed in the ATC (automatic tool change) models "tool holder(s)" or simply "holder(s)". Also we will call the entire array of tool holders in a machine the "machine magazine" or simply "magazine" (The machine will have 12 or 16 tool holders depending on the model).
Button Project Tools opens a window that contains a list of tools, that consists 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:

- Tool holder number (ATC models only)
- 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

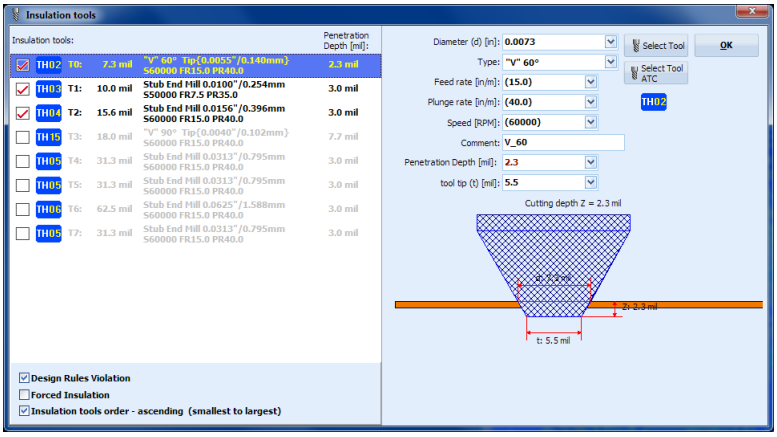


If "Auto tool depth update" check box is checked, changing the penetration depth of a tool will change the penetration of all tools in this operation. As an example if you change the penetration of one tool in the "copper rubout" operation, the change will spread to the rest of the tools involved in 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.

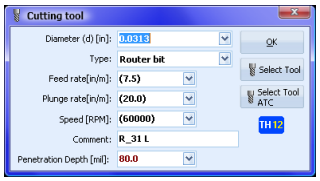
Tool penetration values are entered as positive values to cut into the material. A negative value will not cut the material and will display a warning message before you start this is handy for test runs. A value of Zero "0" will recall the default value for this operation or tool. For changes of tool 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. Using an ATC machine it is always recommended to select tools from the machine magazine or prearrange the magazine to fit your project. This will allow you to machine the entire board side without any manual action.

Some examples of tool selection:
"Insulation Tools"
To change/select the insulation tool click on "Insulate Tools Setup":

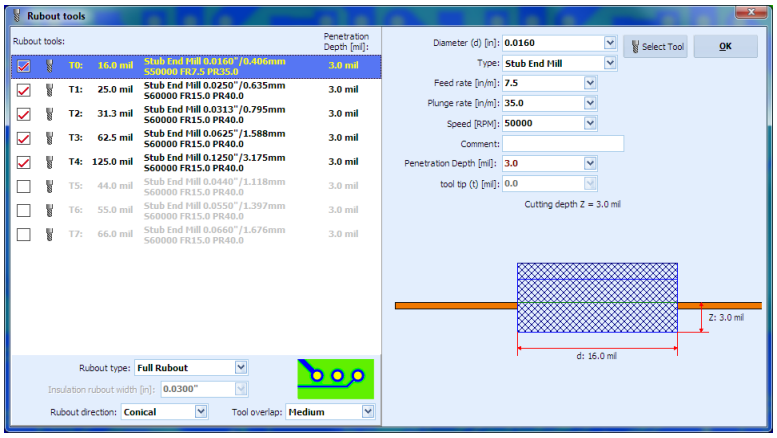


If you have (or already arrange) the necessary tool in the machine magazine simply click on "Select Tool ATC", it will show you only the tools that are currently in your machine magazine. Using similar technique you can make your selection for the tools for other operations.

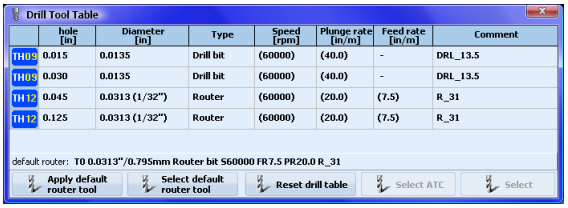
"Cut Tool"



"Rubout Tools"



"Drill Tools"



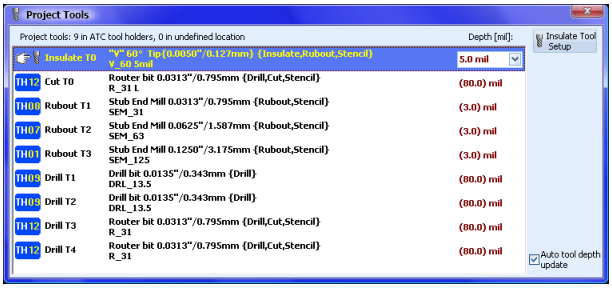
Note:

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.



During the execution of the operations the PhCNC 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

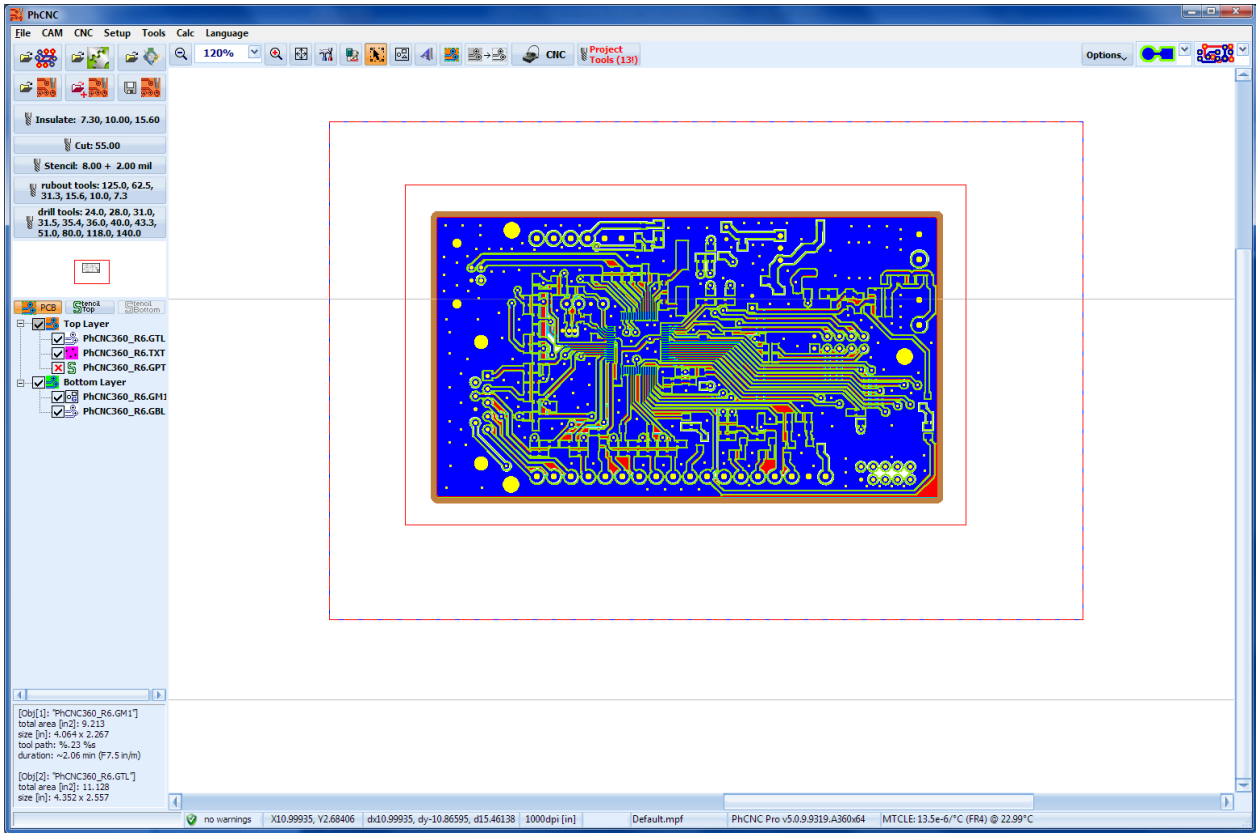
Button is used to enter object selection mode or left-click and select "Select/Clone/Move objects" from the pop-up menu.

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

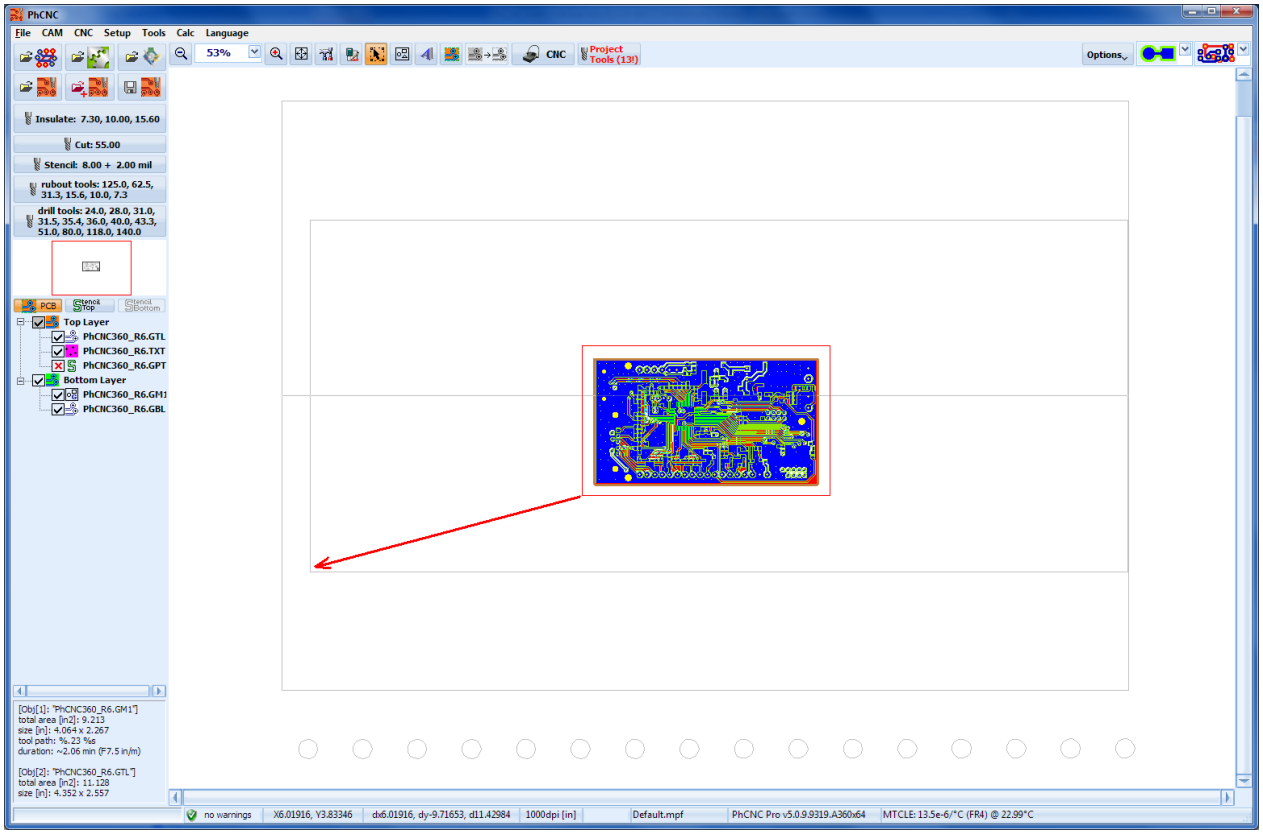
PhCNC in CAM mode has several ways to move objects:
- click and drag using mouse
- "Rotate & Move Selected Objects …" function from popup menu
- "Move To …" function from popup menu

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.

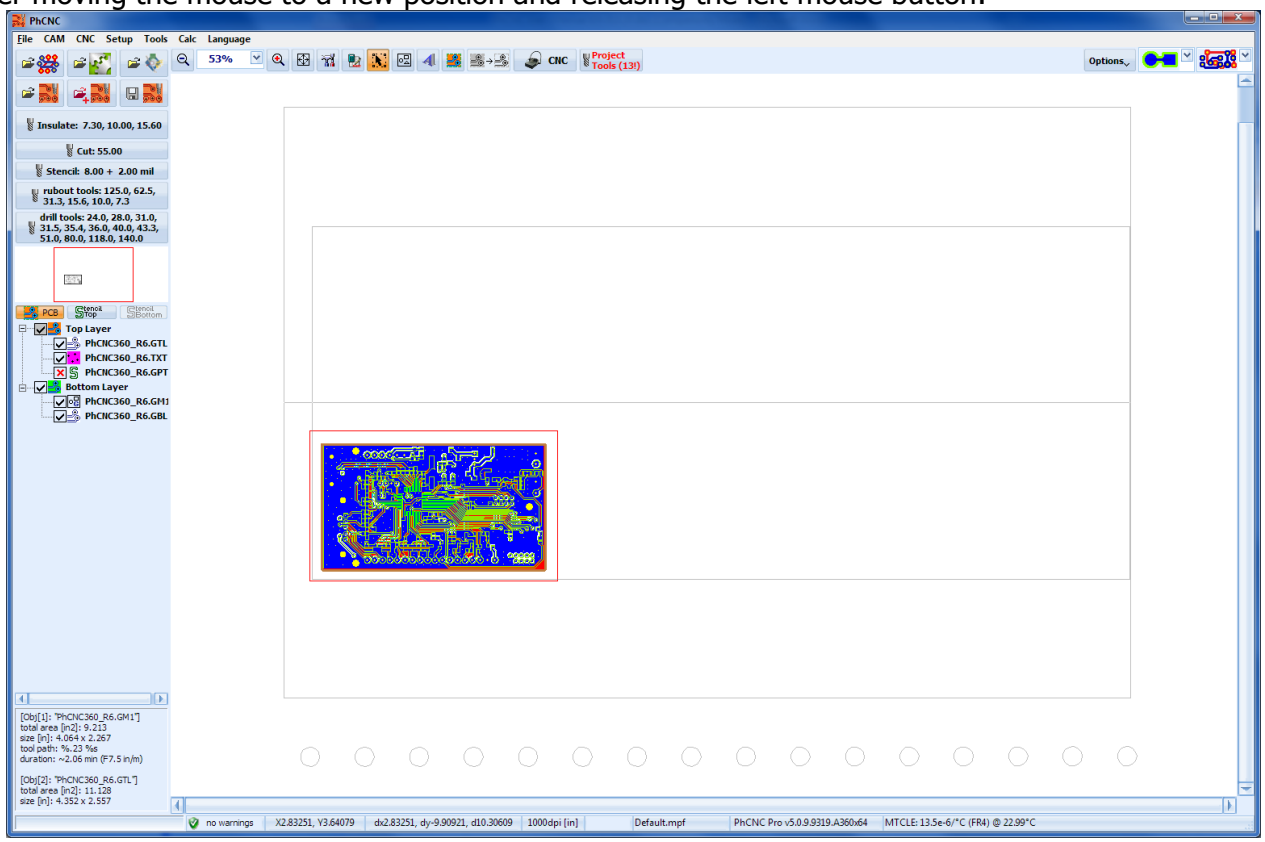
2.6.1 Moving objects using the mouse

In CAM and CNC mode of PhCNC, 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.

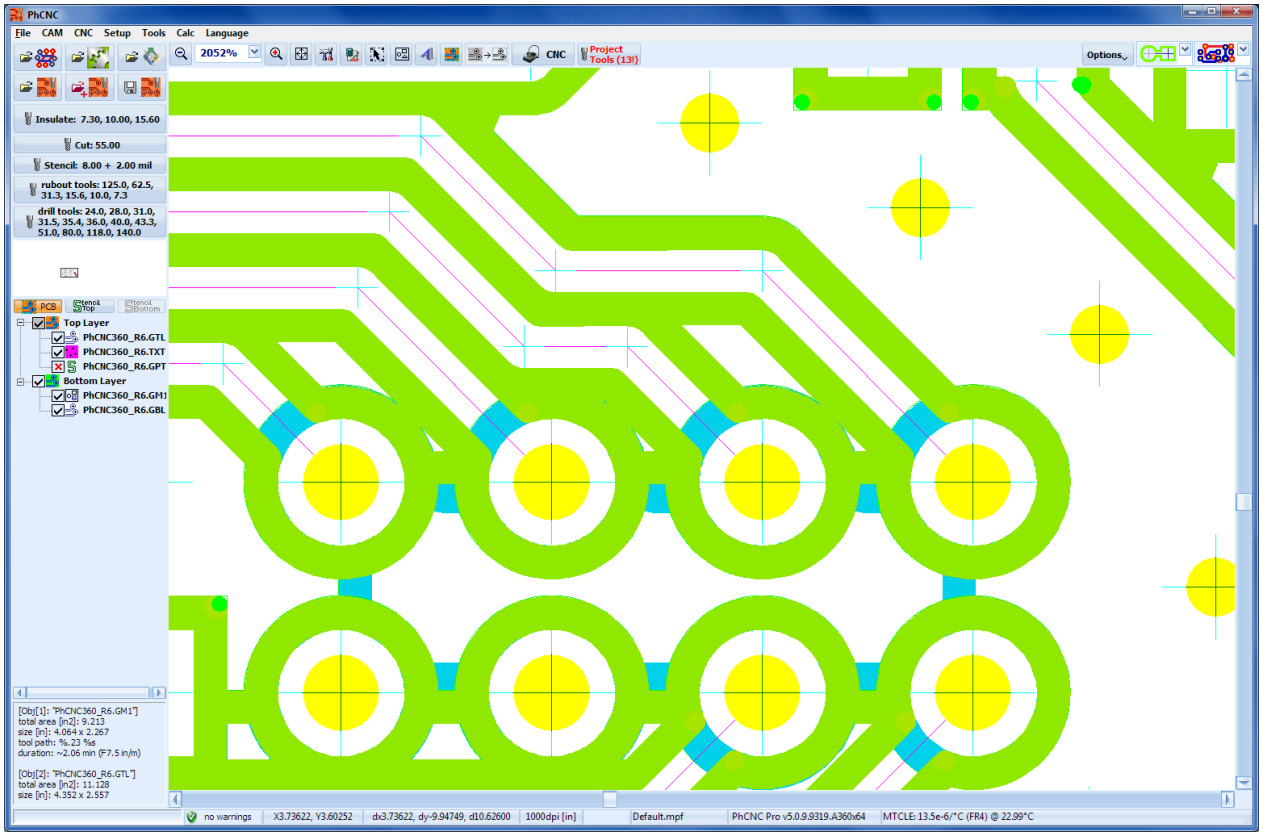


After moving the mouse to a new position and releasing the left mouse button:



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).

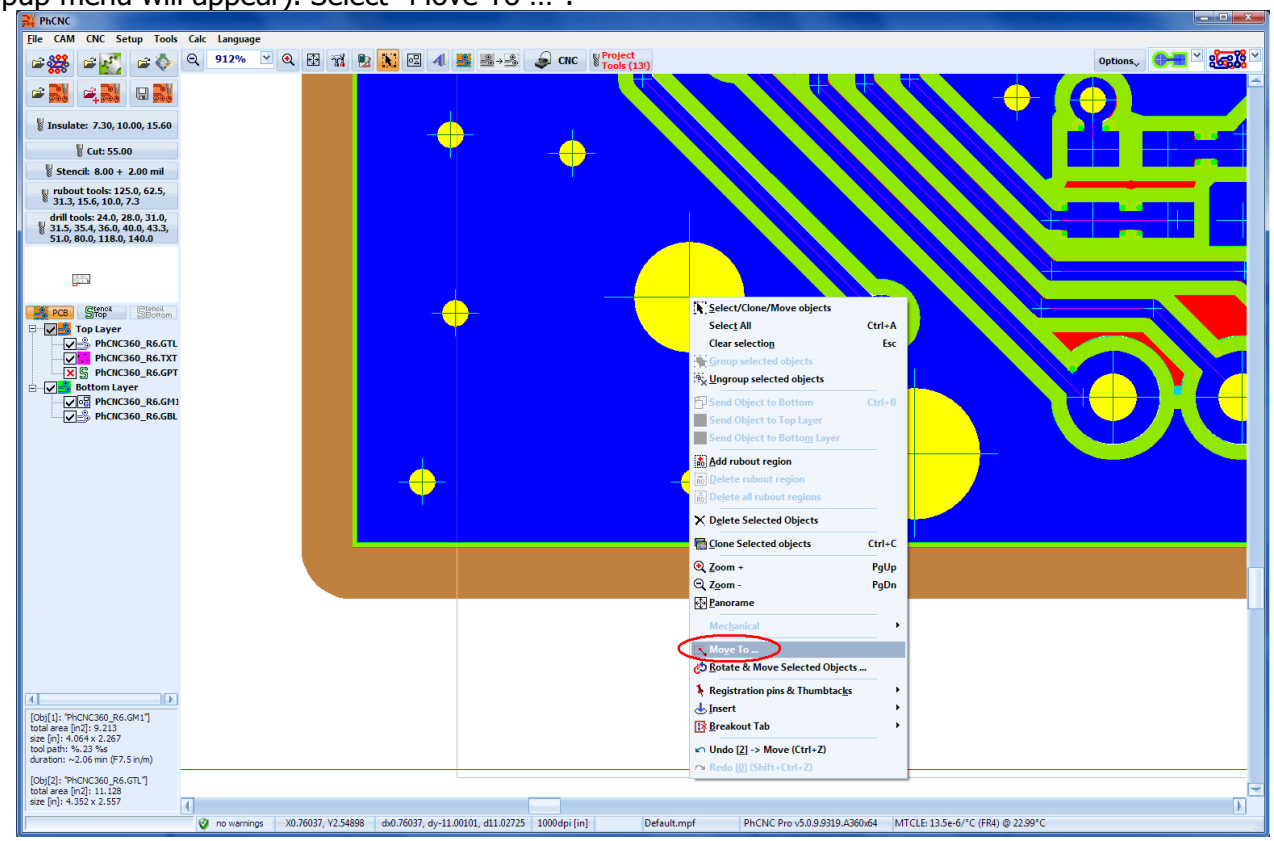


2.6.2 Move using "Rotate & Move Selected Objects ..."

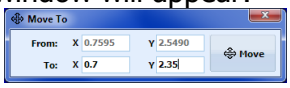
This is the best mode when you need to move an object(s) to a predetermined distance. Described in paragraph 2.8.

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 ...".

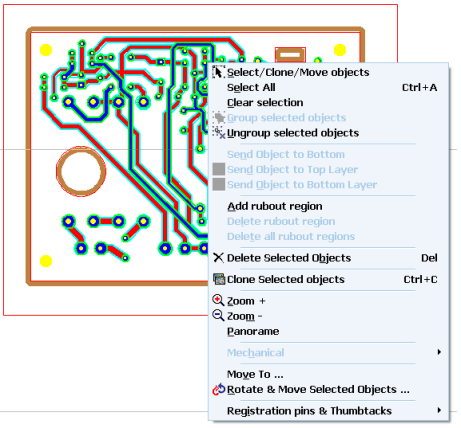


Selecting "Move To …" the "Move To…" window will appear:



"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

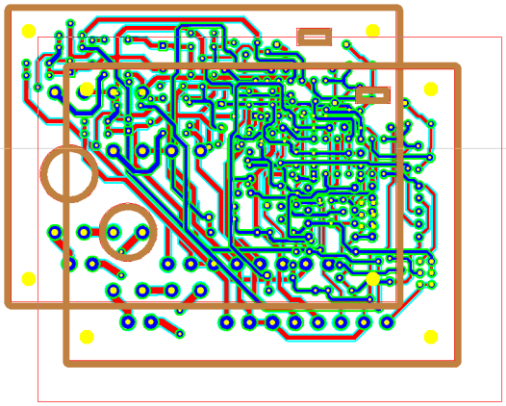


- Select an object or group of objects
- Select "Clone Selected objects" from popup menu

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)!

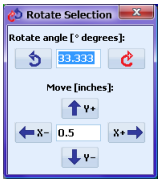


Cloned object

2.8 Rotate objects

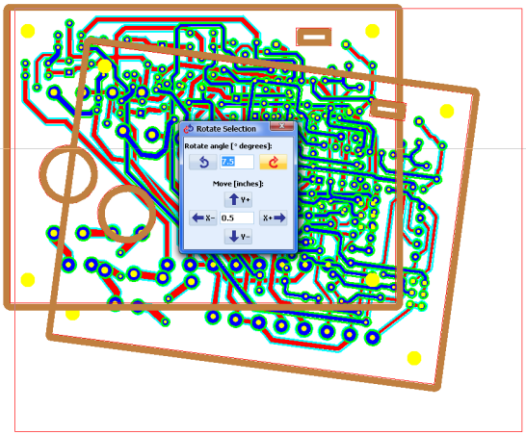
Select an object and click on "Rotate & Move Selected objects" from popup menu:

Rotate Selection Window will appear:

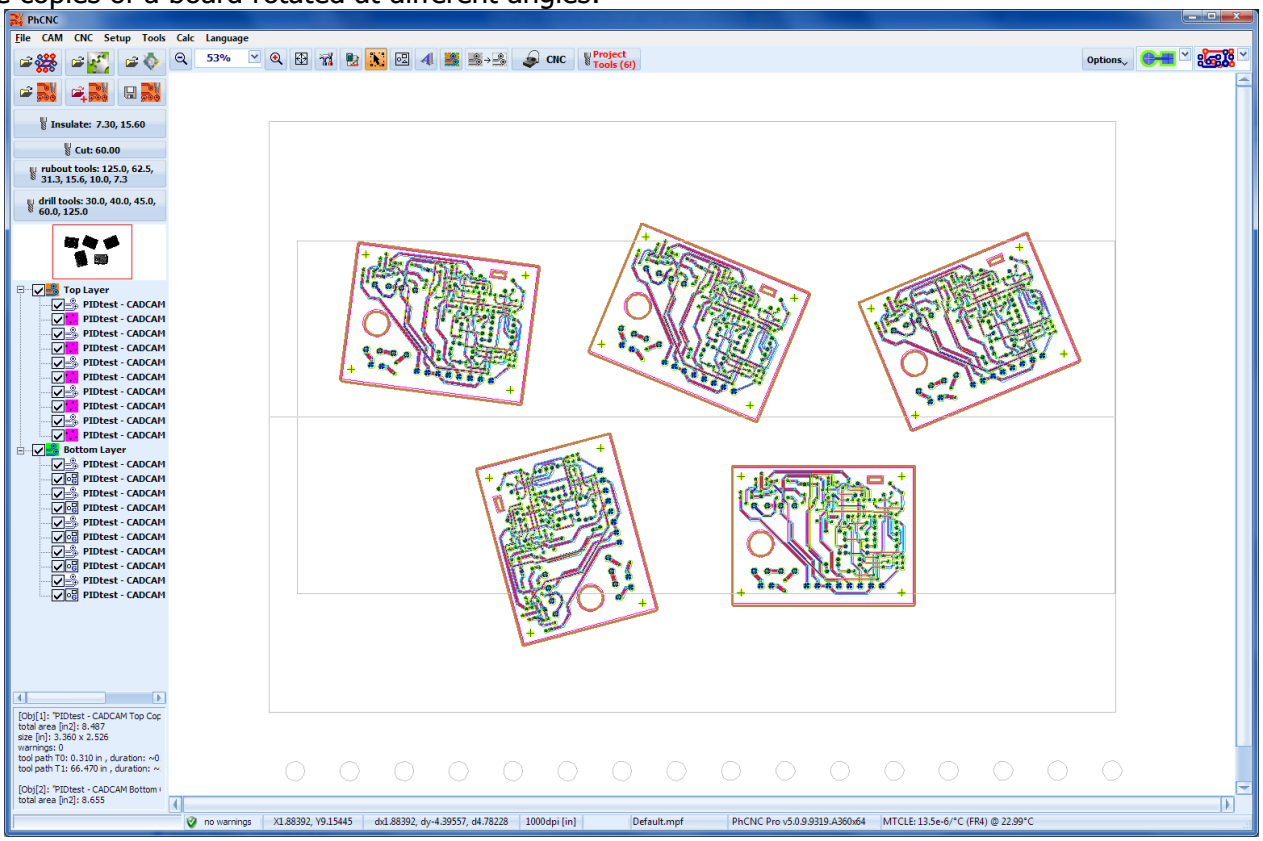


You may set any angle and distance.
Using the direction buttons you will be able to move and rotate the object.

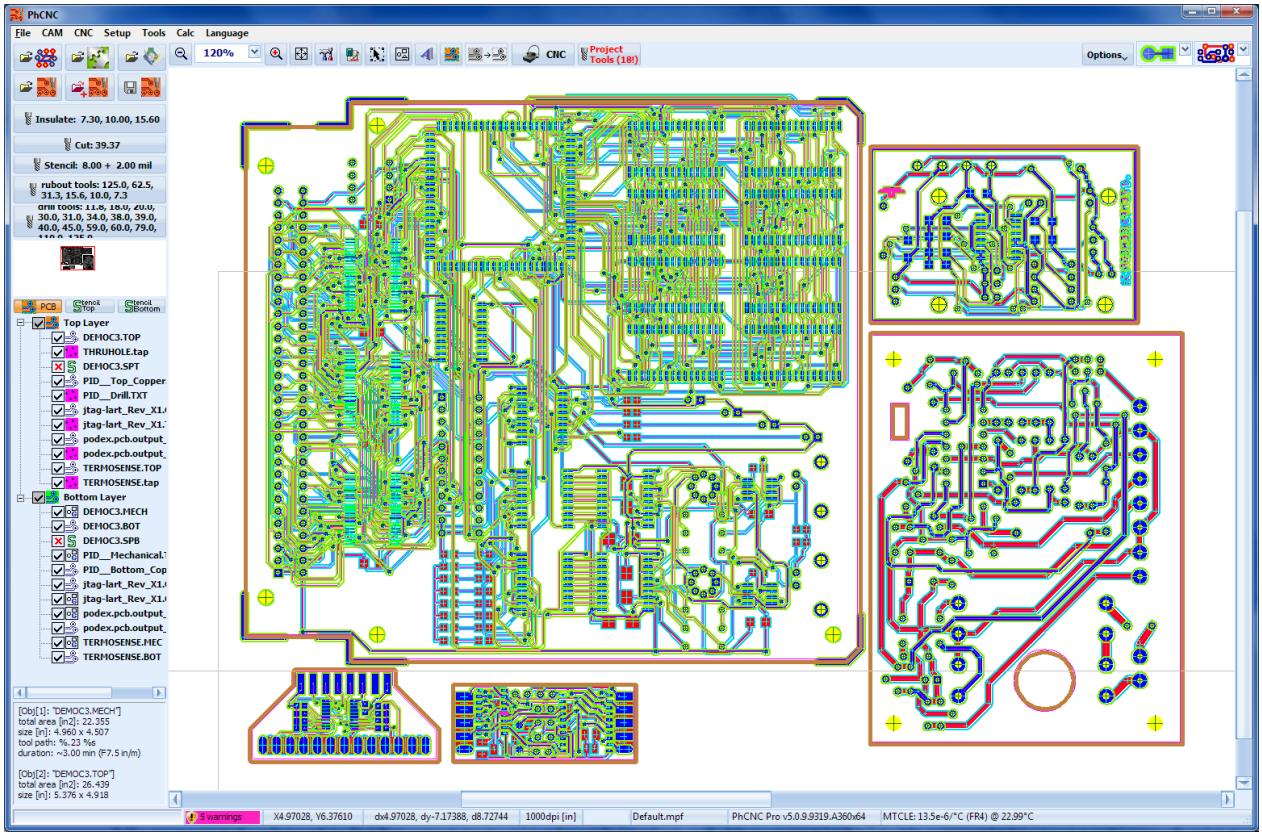
For Example the object below has been rotated at 7.5° CW



Example 2:
Five copies of a board rotated at different angles:



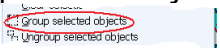
Example 3:
Five projects imported and arranged (some are rotated):



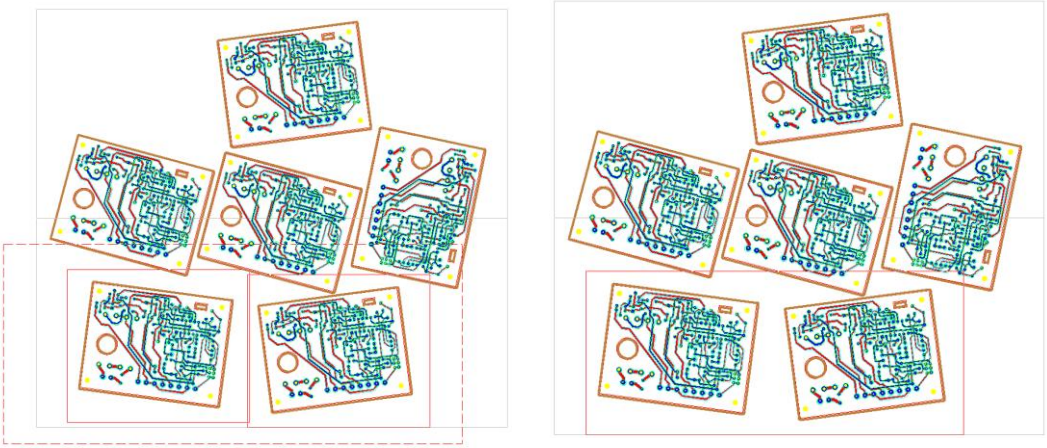
F_A5.PHJ

2.9 Group/ungroup objects

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

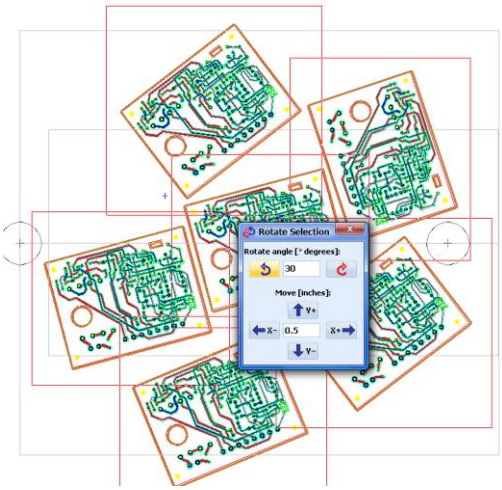


All grouped objects behave as a single object and they maintain their relative position during further manipulations.



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.



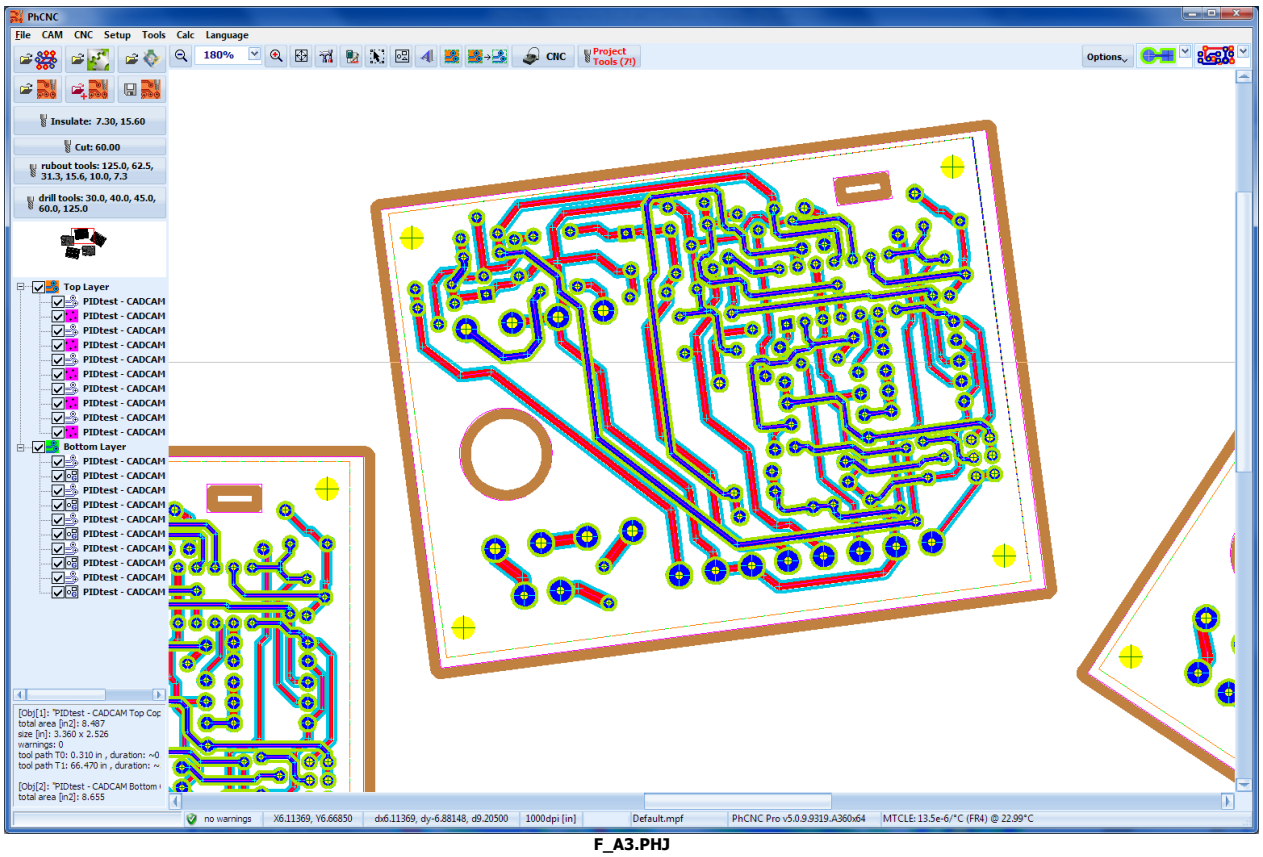
2.10 Delete objects

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.

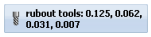
Example of rubout region creation (short dash line):

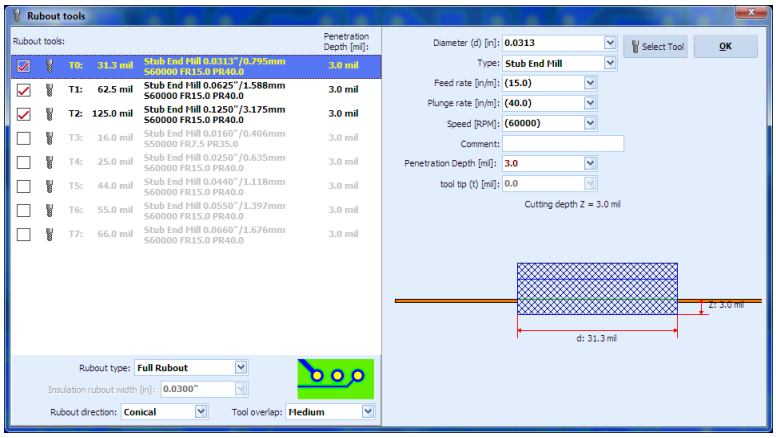


F_A3.PHJ

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.

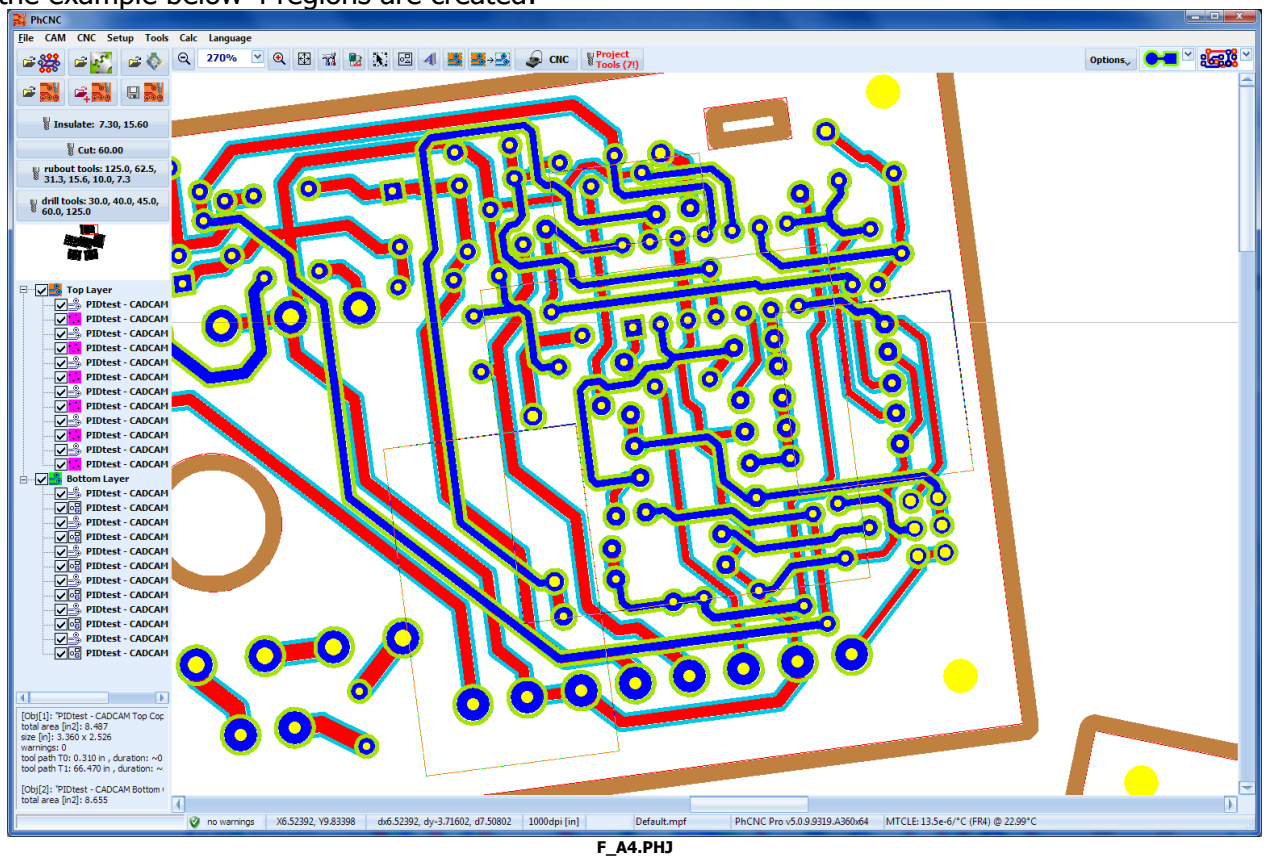


The first and the smallest tool in the rubout process will always be the insulation tool. The rest depend on your design and you can create any combination you want. Enter 0.0 for the positions that will not be used (or press "Delete"). Enter the right sizes for the tools you will use. Rubout will sort 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.

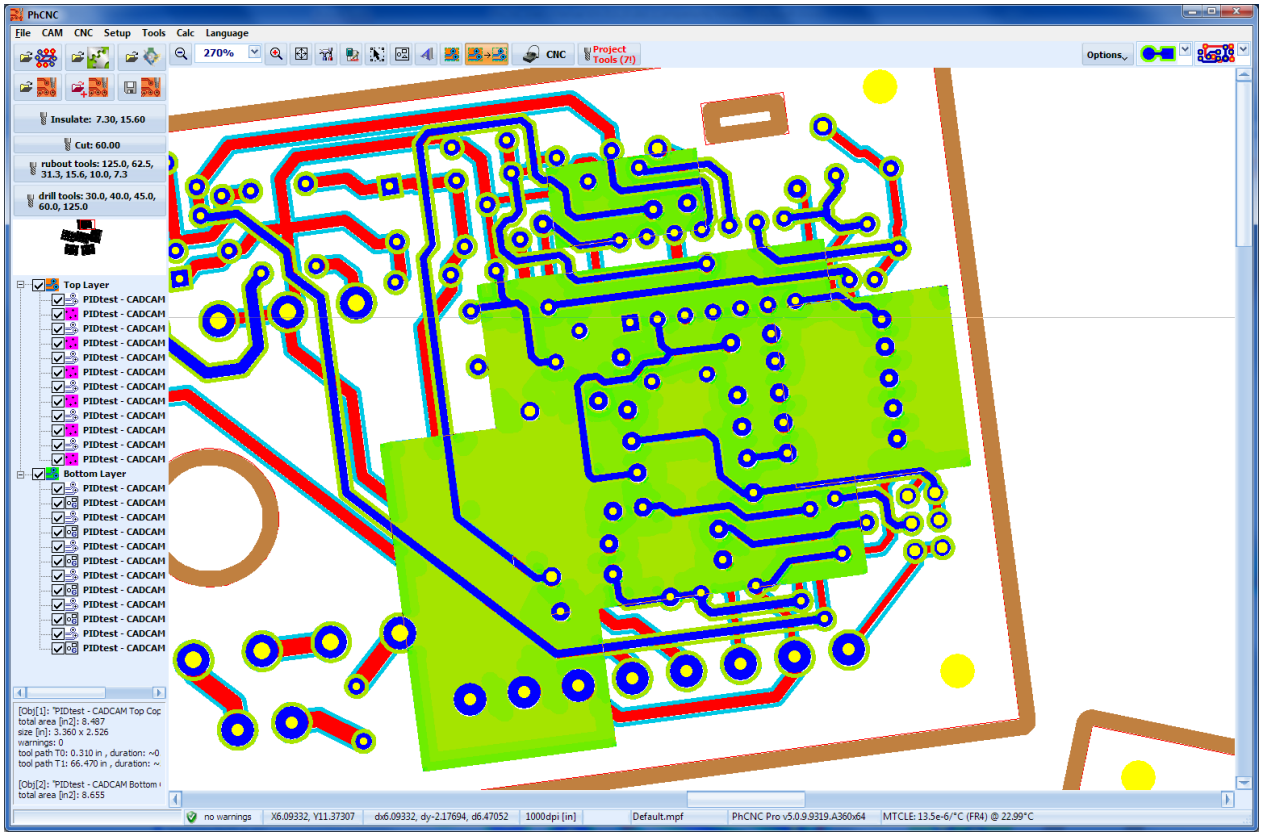
In the example below 4 regions are created:



F_A4.PHJ

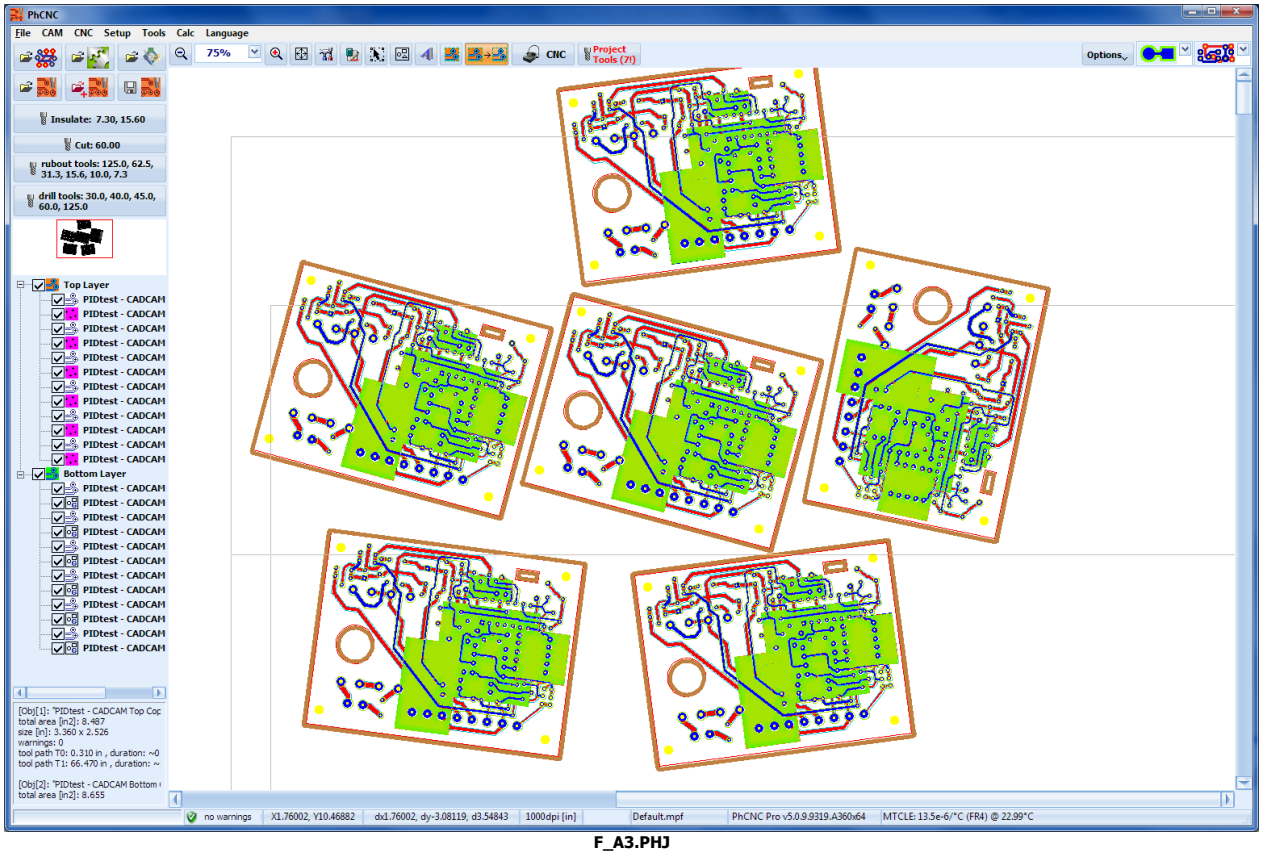
To calculate regions click on the button. First click will calculate the "Rubout regions" and all consequence clicks will toggle between hide/show them on the screen (v1.17.5+)

Here is how they look after calculation:



F_A4.PHJ

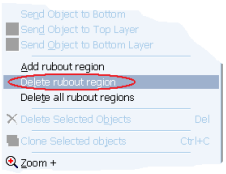
Here is the example above spread to all copies:



F_A3.PHJ

If for any reason you do not calculate the regions before entering CNC mode, PhCNC 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.



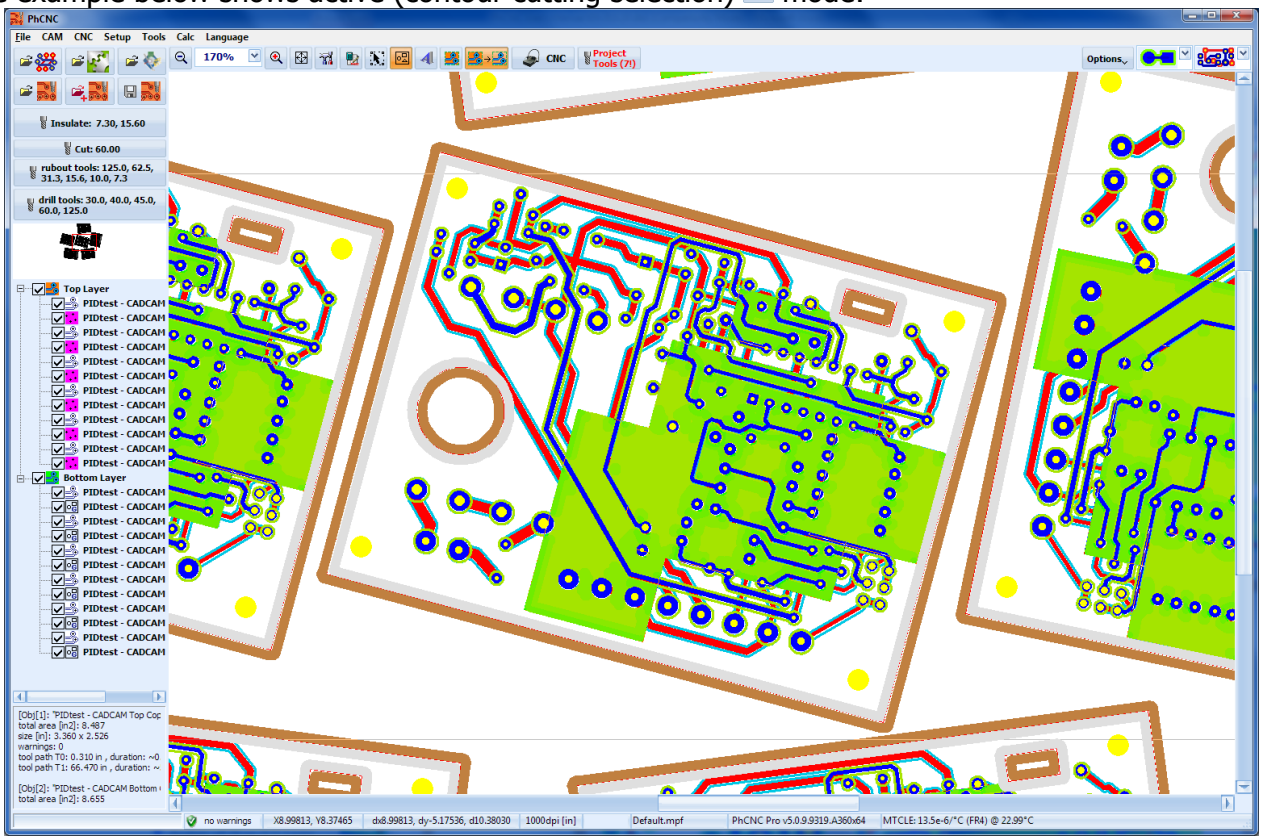
If you would like to delete all rubout regions click on "Delete All rubout regions" from the popup menu.

2.12 Editor for the mechanical layer

2.12.1 Contour cutting selections

If you have loaded a mechanical layer for contour cutting, PhCNC 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:



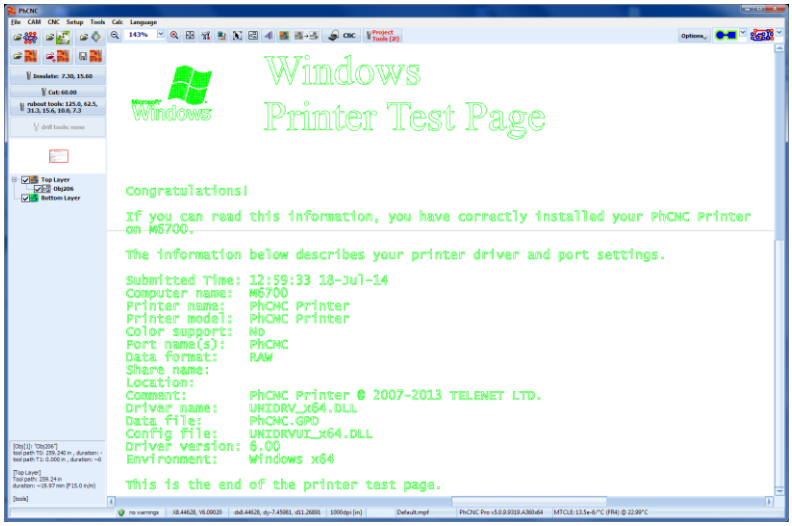
Selected contours are in light brown and the possible, but not selected, opposite compensation contours are in light gray. You may change the selection with a mouse click on the contour. When you leave contour cutting selection mode, unselected contours will become invisible and will not be executed.

2.12.2 Contours selection in the mechanical layer

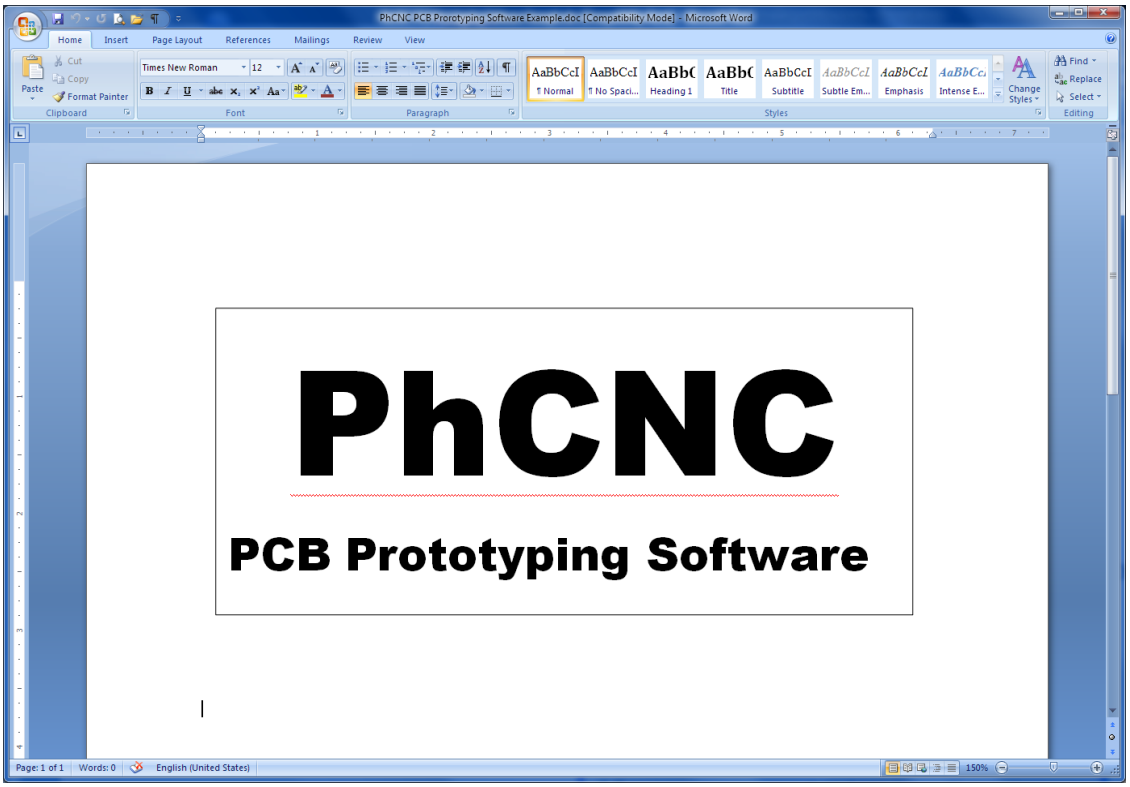
In this mode, you can perform the following operations with the contours generated by "PhCNC Printer" and (import of text and graphics).

2.12.3 Contours generation with "PhCNC Printer"

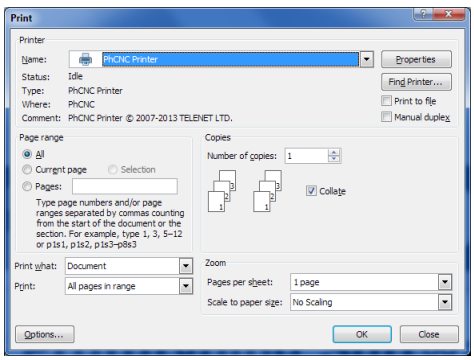
Upon starting PhCNC, the "PhCNC Printer" driver is installed, which allows you to import vector information in PhCNC from any application or CAD/CAM system.
On the figure is shown the test page of the printer.



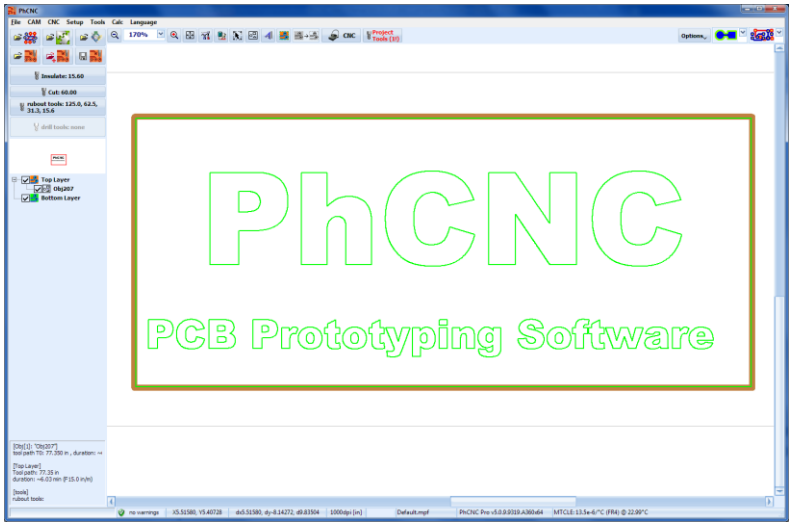
Example:
Name plate generated with MS Word:



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



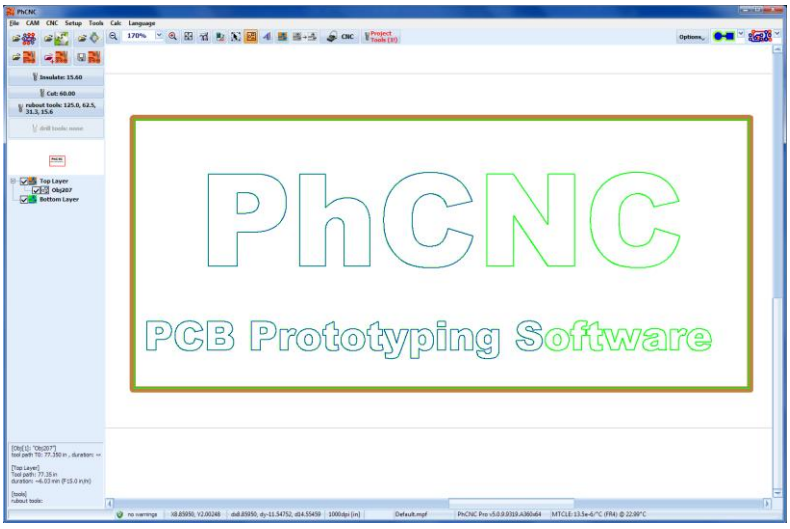
Remark:
PhCNC must be started (you can also start it after the printing)!
The vector image is imported automatically in PhCNC (as a separate object):



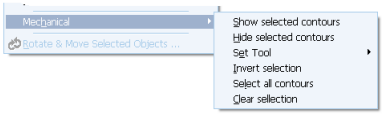
By default all contours will be processed with the tool for insulation (T0). If you want to select contours for cutout (for example the external contour), enter into the mode for editing of mechanical contours by left clicking the button.

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.



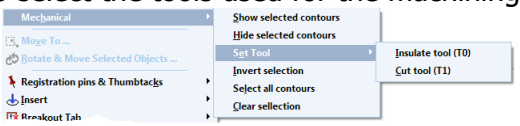
Right clicking on selected contours allows you to enter the "Mechanical" menu.



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.



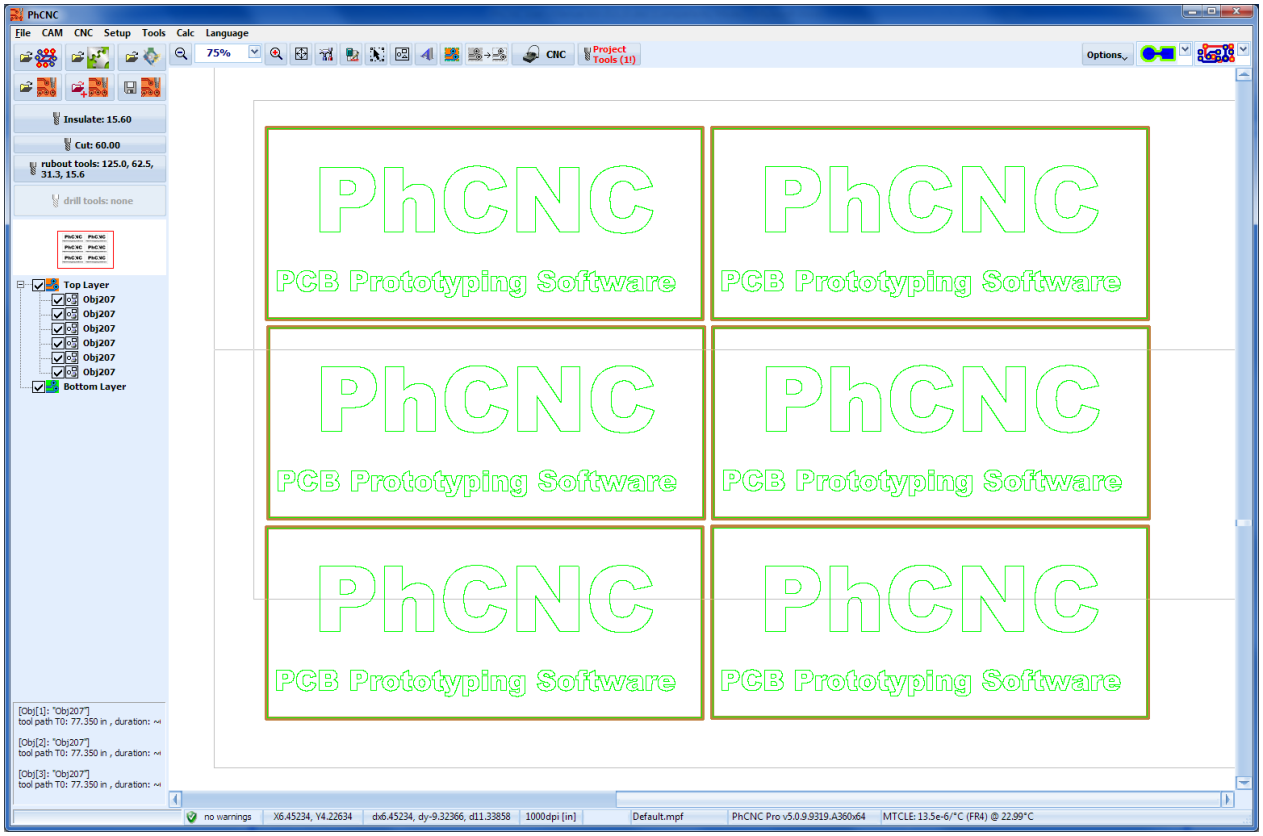
Here, T0 is the current tool for insulation and T1 is the current instrument for cut-out.

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

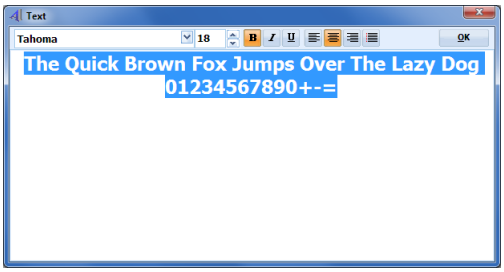
You can also use rubout. On the figure below the inscription which was selected only for insulation is now selected for rubout.

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

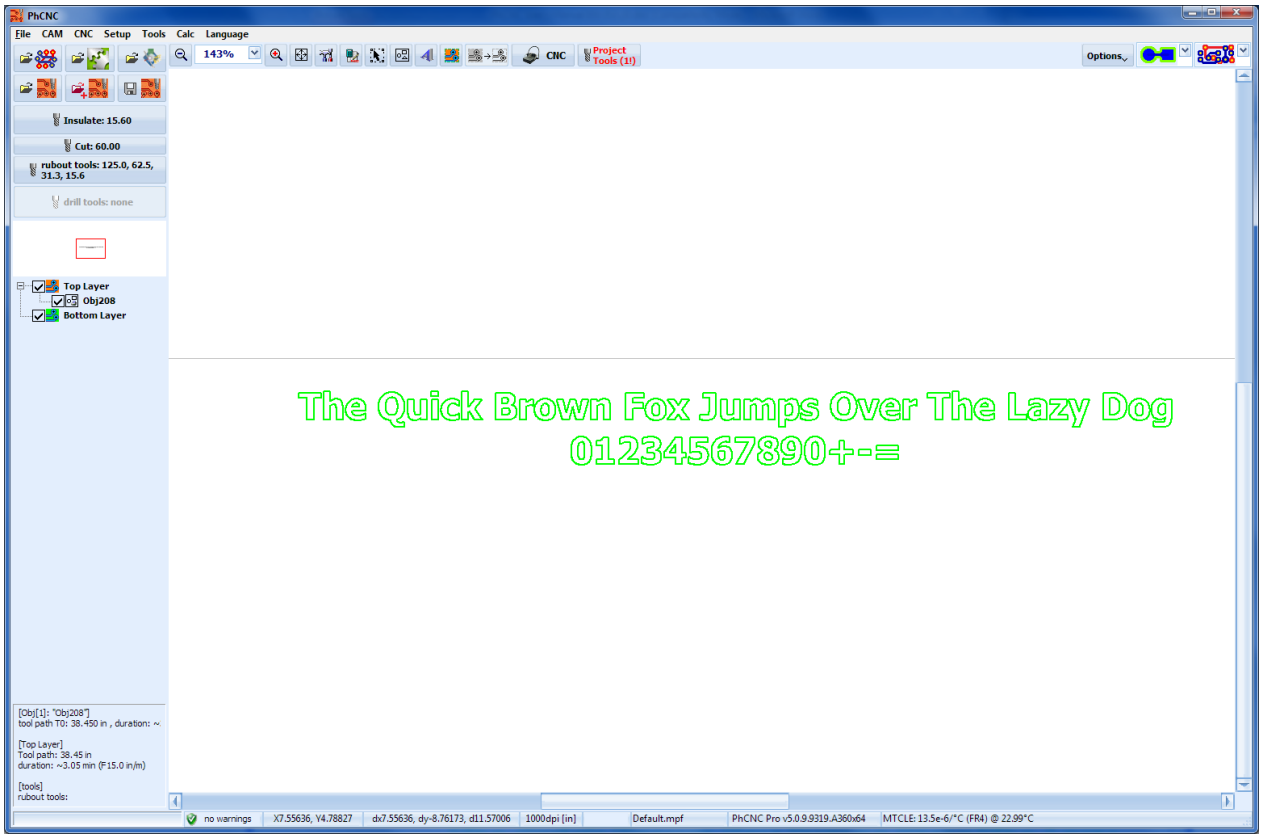
As with Gerber objects, you can rotate and clone the objects, generated with "PhCNC Printer":



2.12.4 Generation of contours with ("Insert Text")

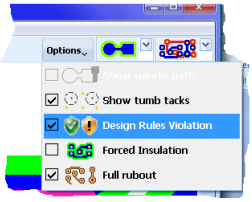


The result is shown on the figure below:

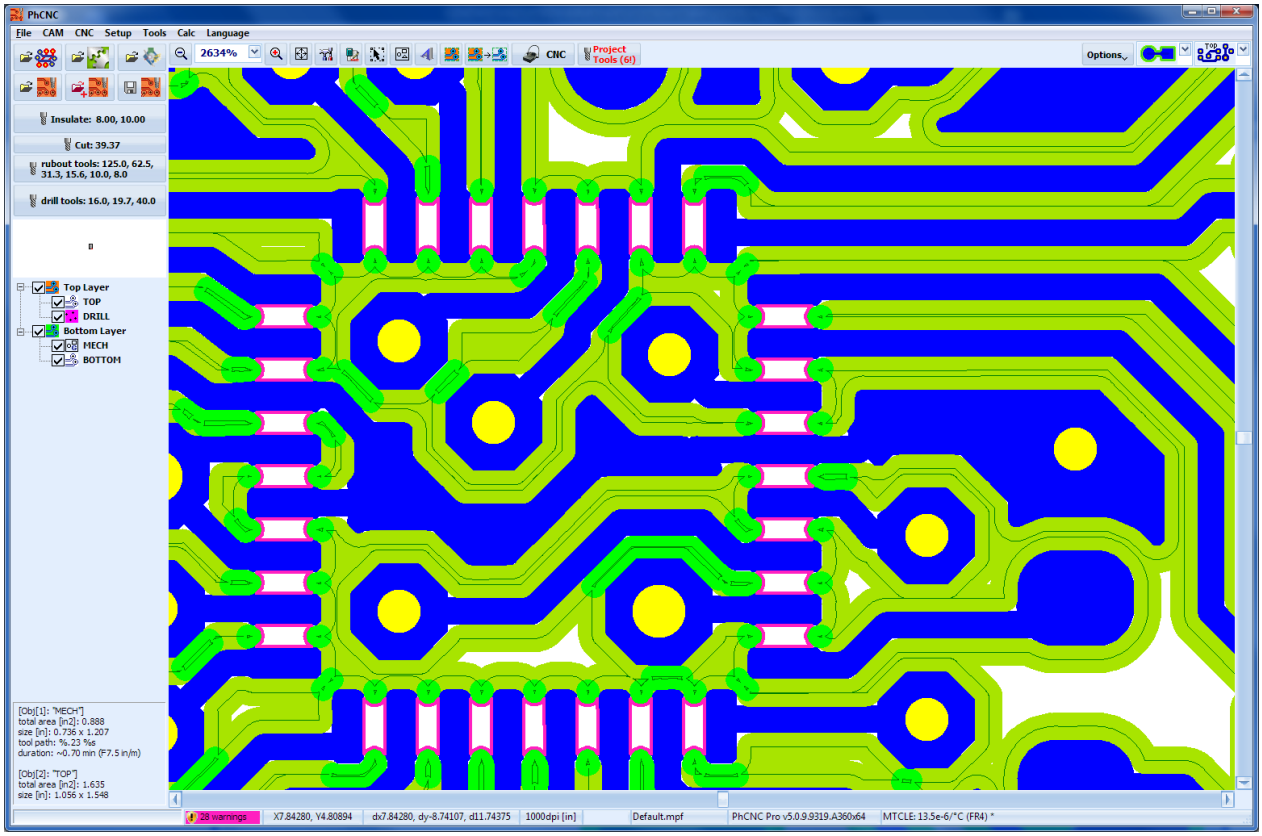


2.13 Design Rules Violation (DRV) detector (Pro version only)

PhCNC Pro 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.



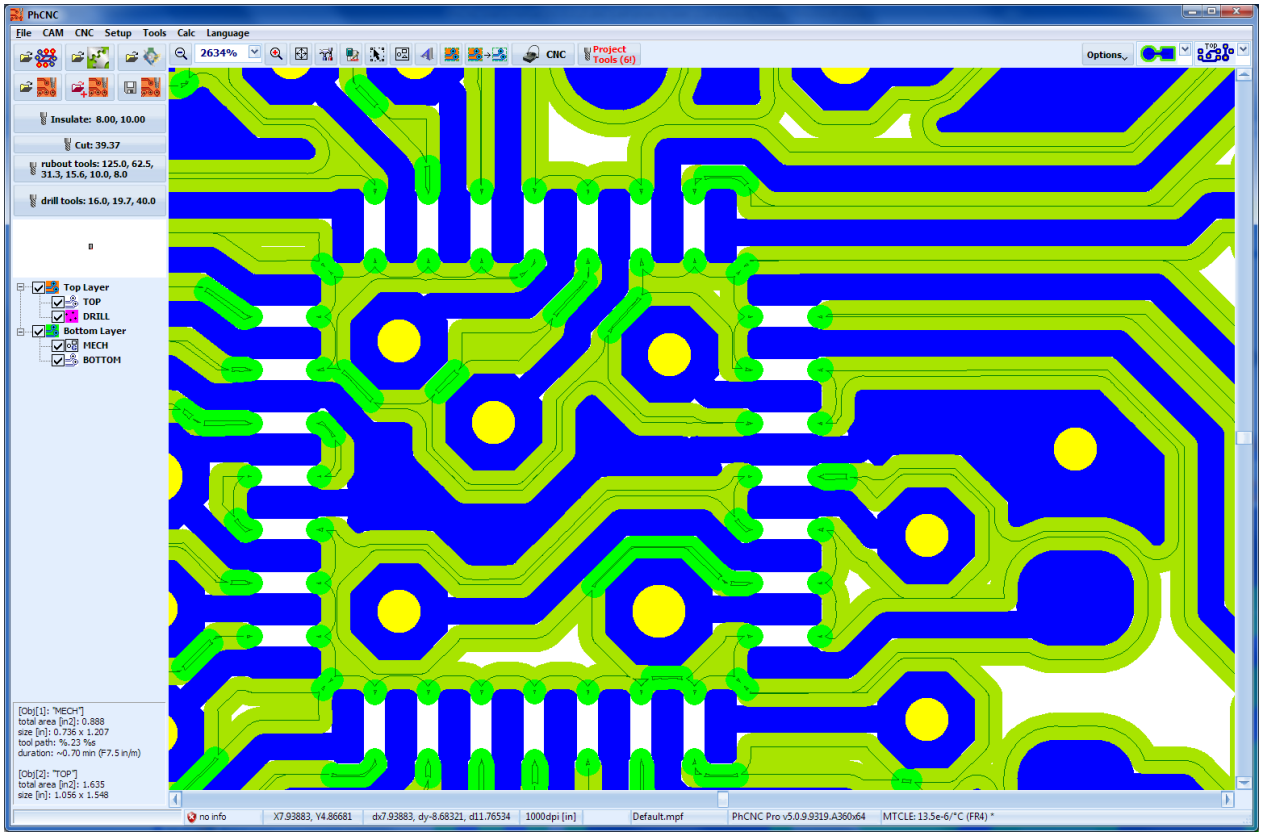
When this option is checked DRV detector is active. It checks for DRV compatibility for each generated tool path. The DRV spots are presented with easy to see default pink:



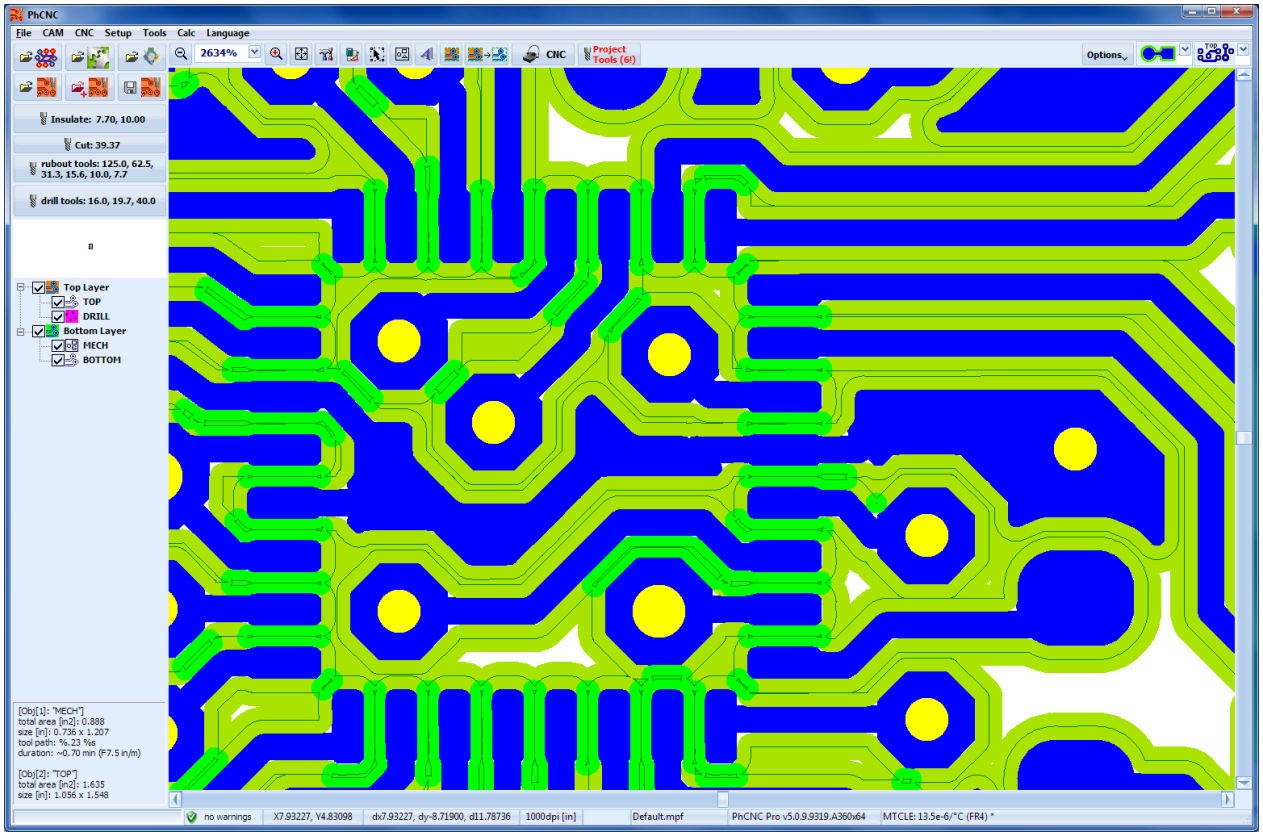
The status bar displays information (28 warnings) about DRVs and their number (for all layers on).

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

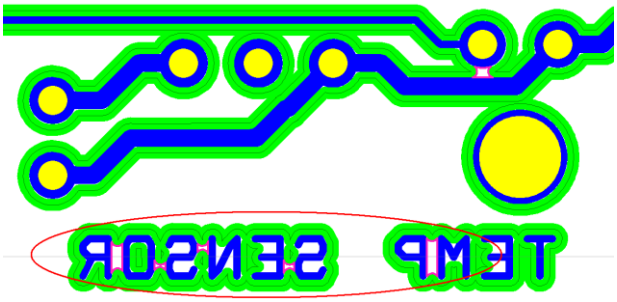
Button also shows/hides DRV spots. If it is not activated during the tool paths generation you may turn it on later and DRV detector will check for DRVs.



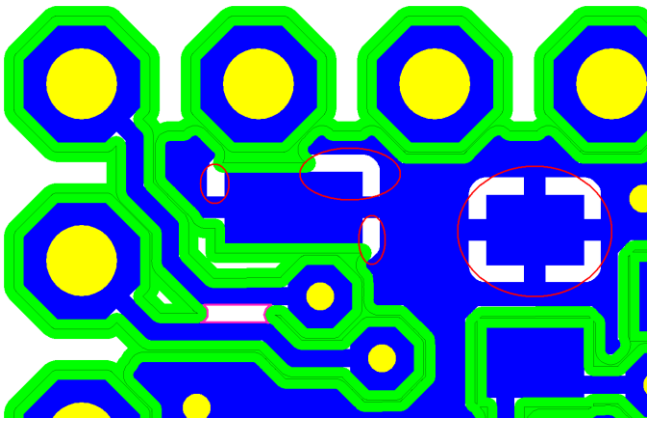
If there are no DRVs the status bar will show the no warnings indicator. If DRV detector was not activated during the tool paths generation the status bar will show the no info indicator. If DRVs exist the only practical solution is to reduce the diameter of insulation tool (See 2.23 Forced Insulation). 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 belong 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.



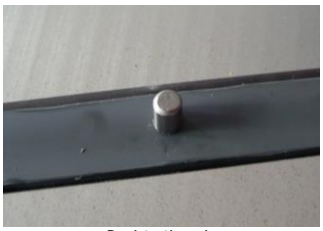
2.14 Thumbtacks & Registration pins (Pro version only)

PhCNC Pro allows you to see and add thumbtacks & registration pins visible in CAM & CNC modes of PhCNC.

Button shows/hides the active thumbtacks & registration pins.

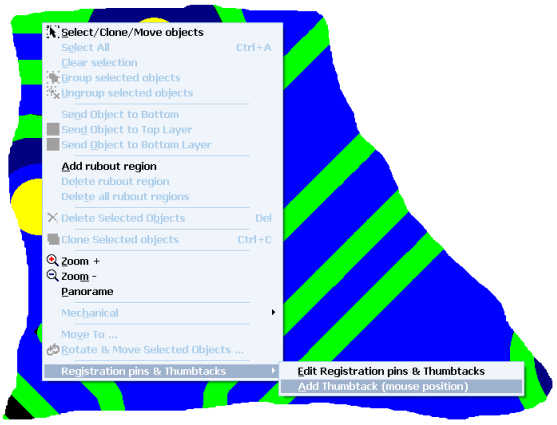
2.14.1 Registration pins

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

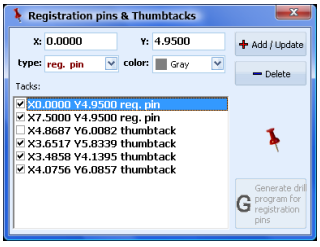


Registration pin

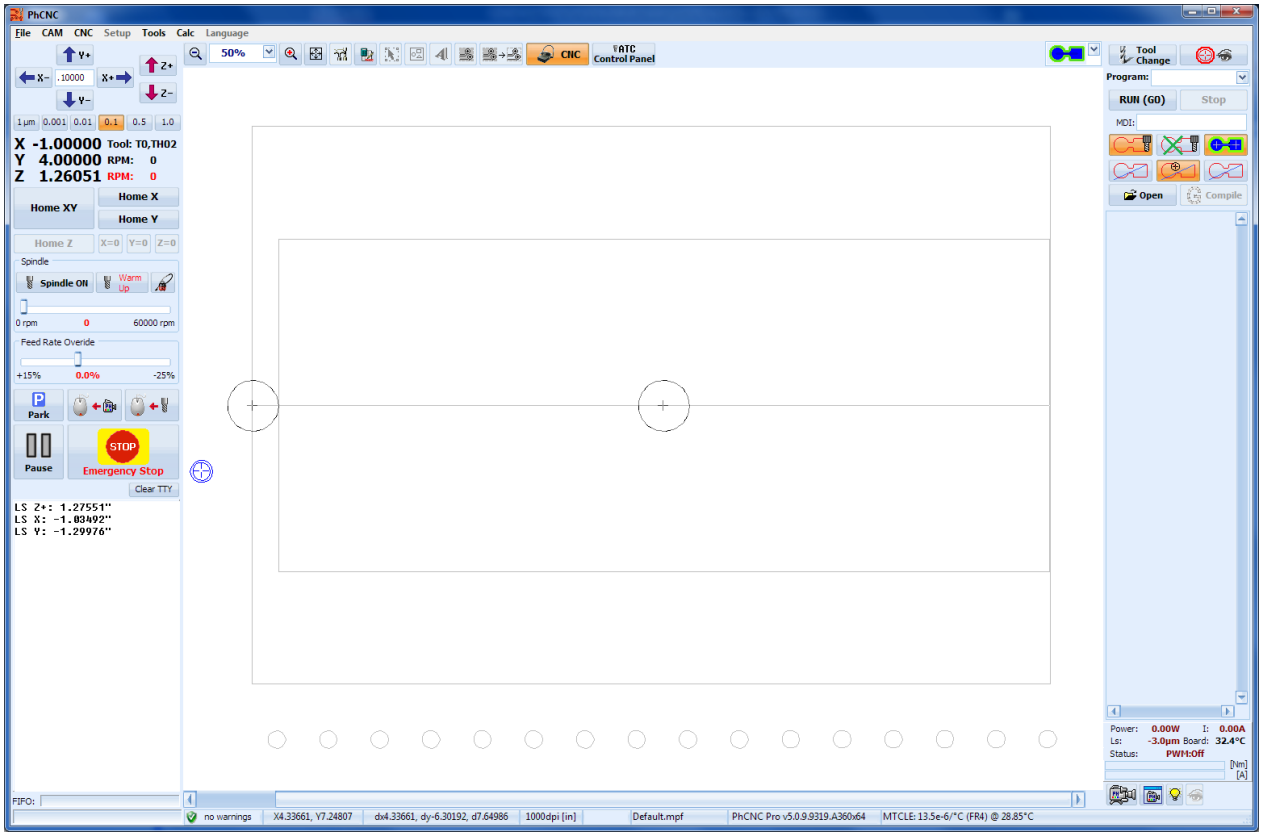
During the setup process you should not place any objects closer than 0.5 inch to registration pins. Using this option you will be able to see the real relative position between any pin and your design.
Around each pin you will see 1.0 inch diameter circle (dashed line) that represents the area that should be clear of project objects.
The usage of the pins depend of your configuration. If your system is equipped with video camera (all ATC models come with it), it is far more accurate to use it in an optical fiducial registration. Also we do not offer registration pins on all systems equipped with vacuum holding table. This option comes with camera too, if it is manual tool change model.
For manual tool change systems that do not have camera option, the operator need to install the registration pins. It has to be a center line in the middle of Y coordinates and strictly parallel to X axis. This is the line that create a mirror image and tool paths for the machining of the bottom layer.
Adding/editing of registration pins is in the popup menu:



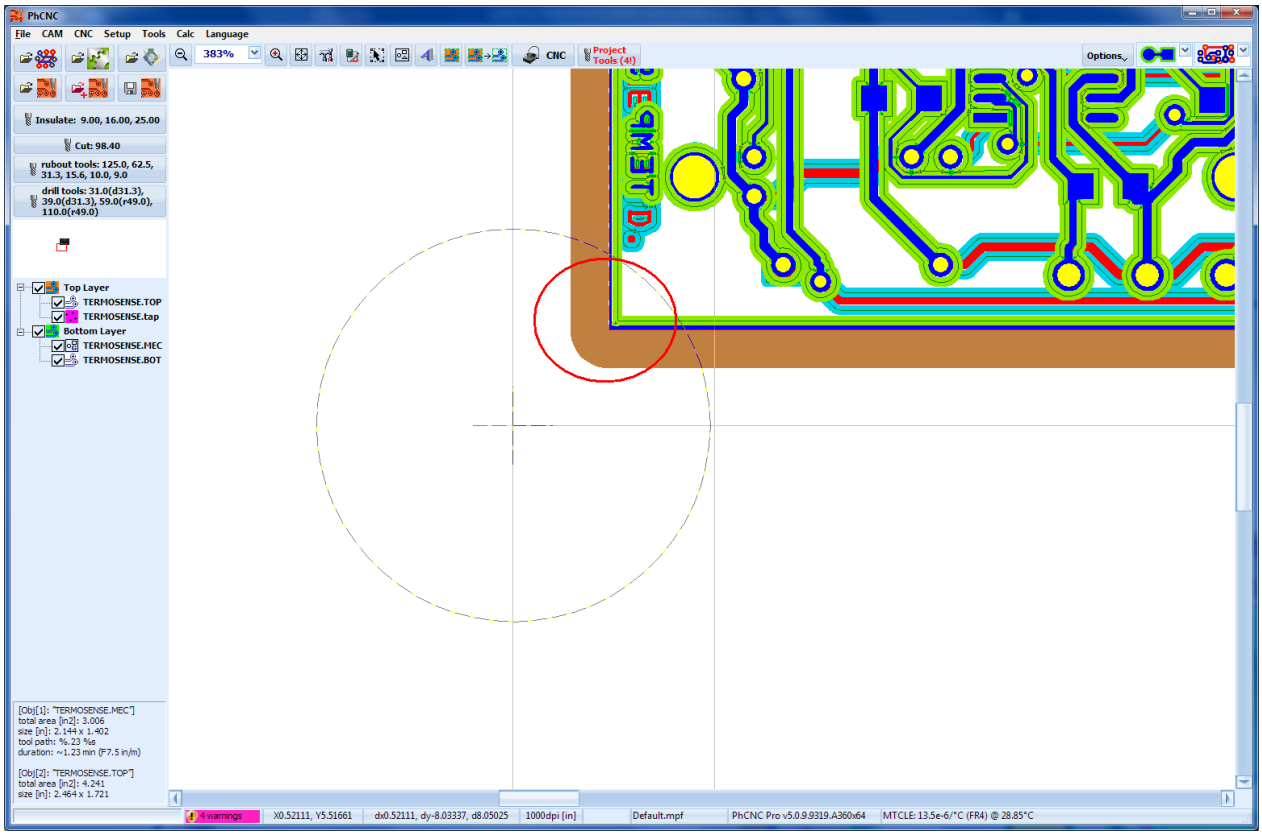
Below is the window that allows you to add/edit registration pins and thumbtacks:



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 example above we have 2 registration pins at X=0.0000 and X=7.5000 shown below.



Here is an example for an improperly set object:



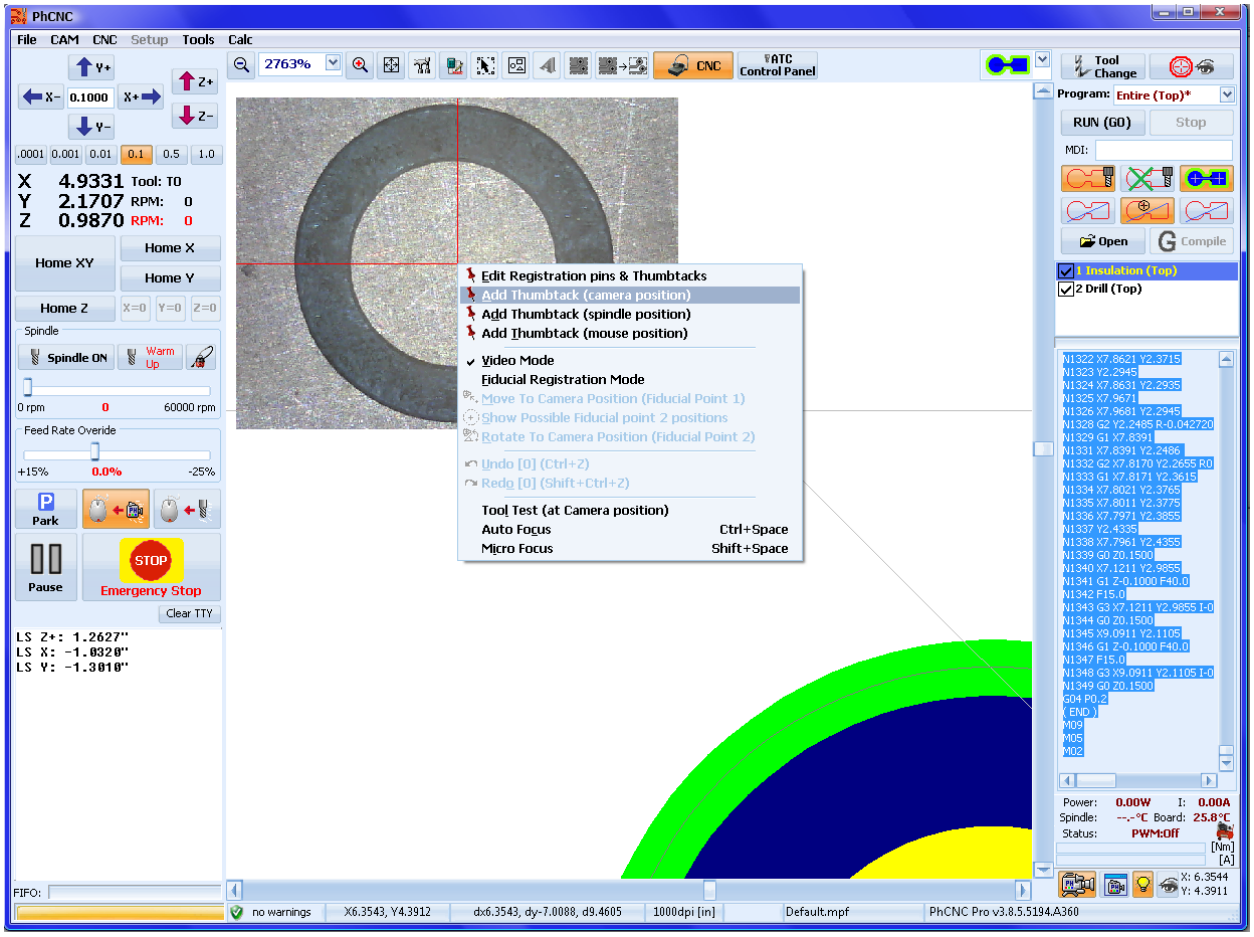
The area above in the red circle is in violation of restricted space. The depth limiter ring will hit the pin during the machining process.

2.14.2 Thumbtacks.

Thumbtacks are markers visible in CAM and CNC mode. They are very convenient tool during the setup. Thumbtacks are markers visible in CAM and CNC mode. They are a very convenient tool during the setup. Thumbtacks transfer their position between the two modes (CAM, CNC) without errors. You can set their position using one of the following ways:

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

Below is an example of adding a thumbtack to the position of the camera in CNC mode:



2.15 Insulating copper rubout (Pro version only).

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.

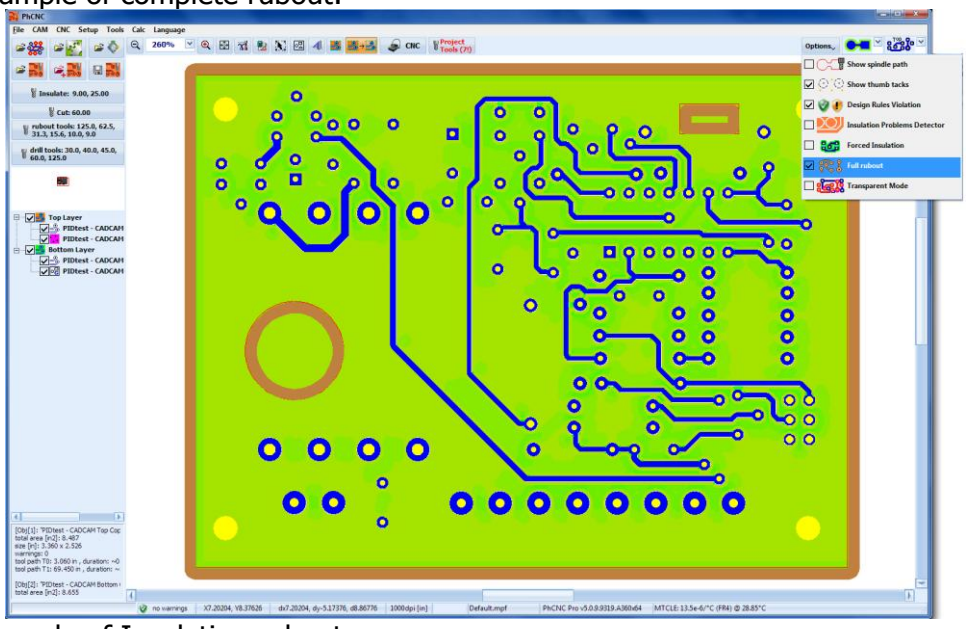
Rubout has two modes:

- 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.

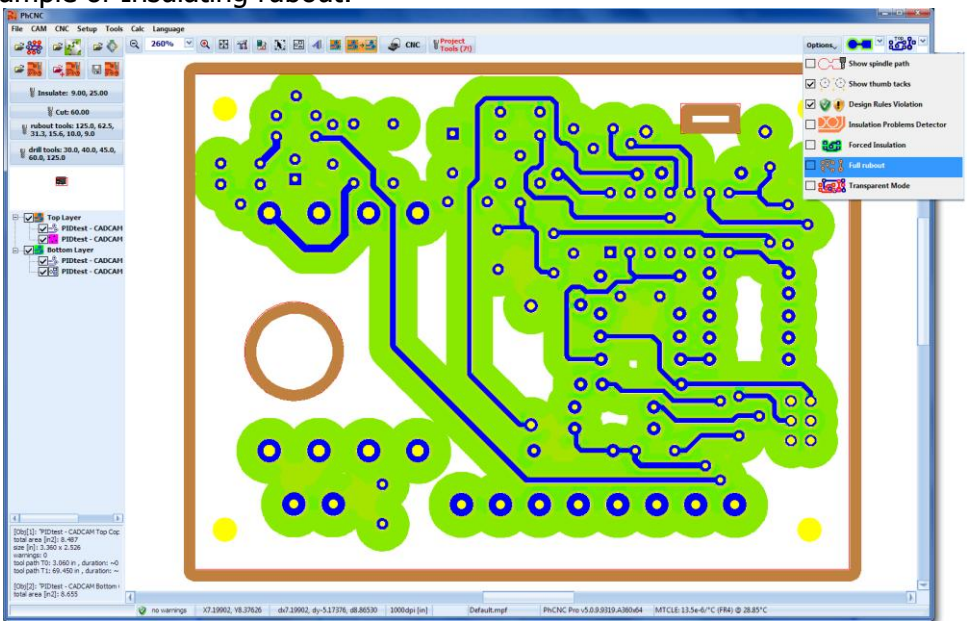
Complete or insulating rubout can be selected by this toggling button

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

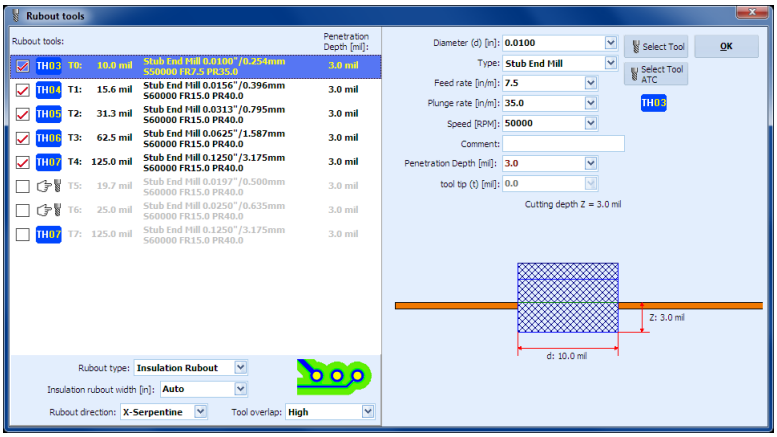
Below is an example of complete rubout:



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 then 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 can not penetrate (without interfering with design features).

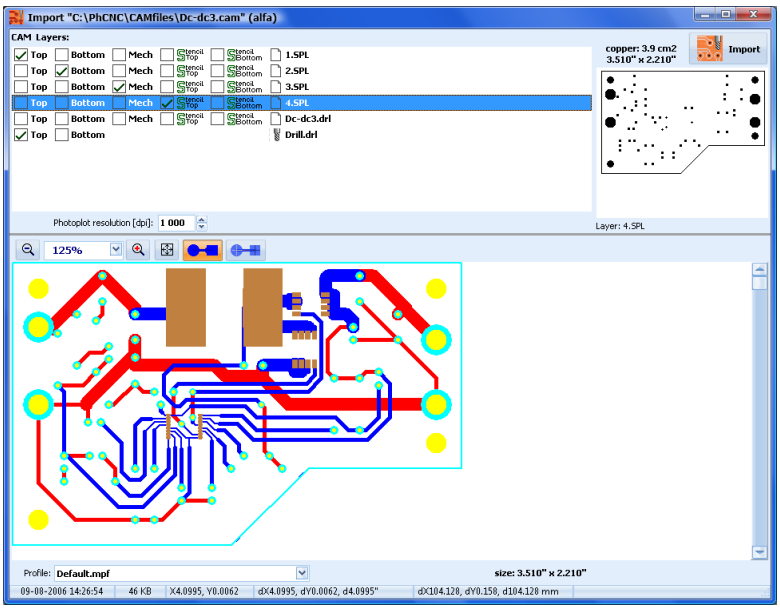
2.16 CAM350 import (Pro version only)

PhCNC Pro is able to import CAM350 files without the need to export them (from CAM 350) in Gerber and/or Excellon formats.

Notes:

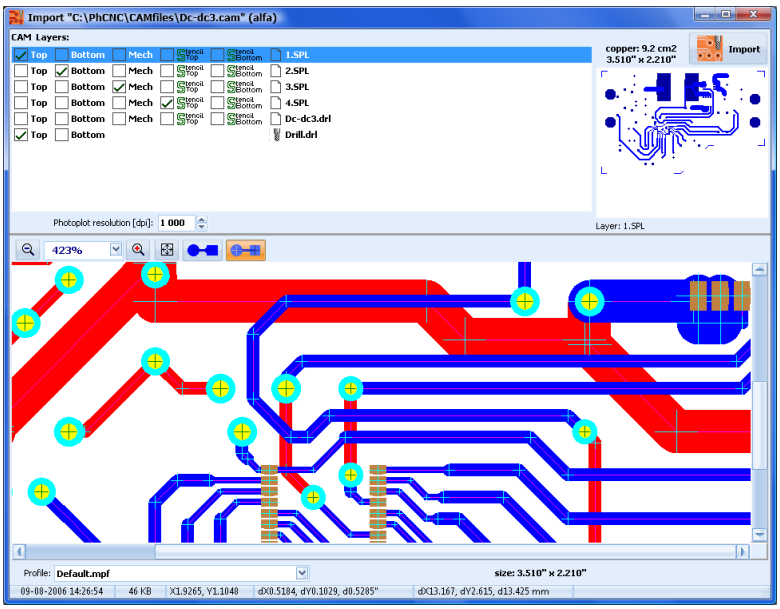
1. CAM350 ® is a registered mark of Downstream Technologies, LLC.
2. Files generated by CAM 350 do not have released specifications. We consider this option an "Alfa" revision and we are not responsible for errors.

Button will allow you to select a CAM350 file (.CAM).
Once you select a .CAM file the "CAM layers" window will appear. In the list are the layers of loaded .CAM file:



Similar to importing Gerber/Drill files, the operator should select layers for import. The drill file is generated from the information embedded in CAM file. A click over the layer shows the graphics in the right window. Selected layers appeared in the lower window.

By default PhCNC selects the first 3 layers as Top (copper), Bottom (copper), and Mech (mechanical). The operator should check if this order is correct. Similar to Gerber/Drill import you can use the lower window for detailed inspection and measurement of the files. (See 2.1.3)



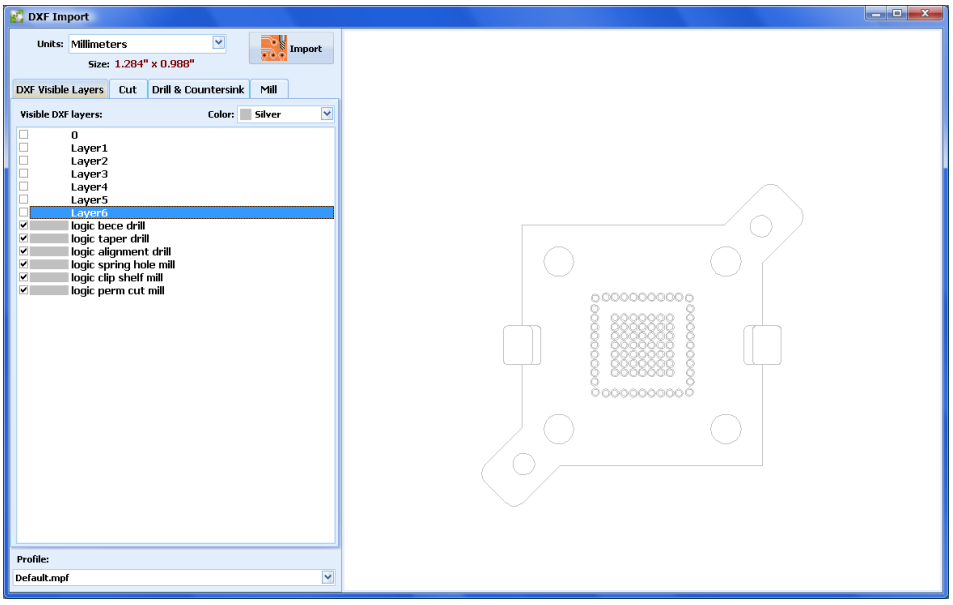
2.17 DXF import (Pro version only)

PhCNC Pro imports 2D DXF files (AutoCAD).

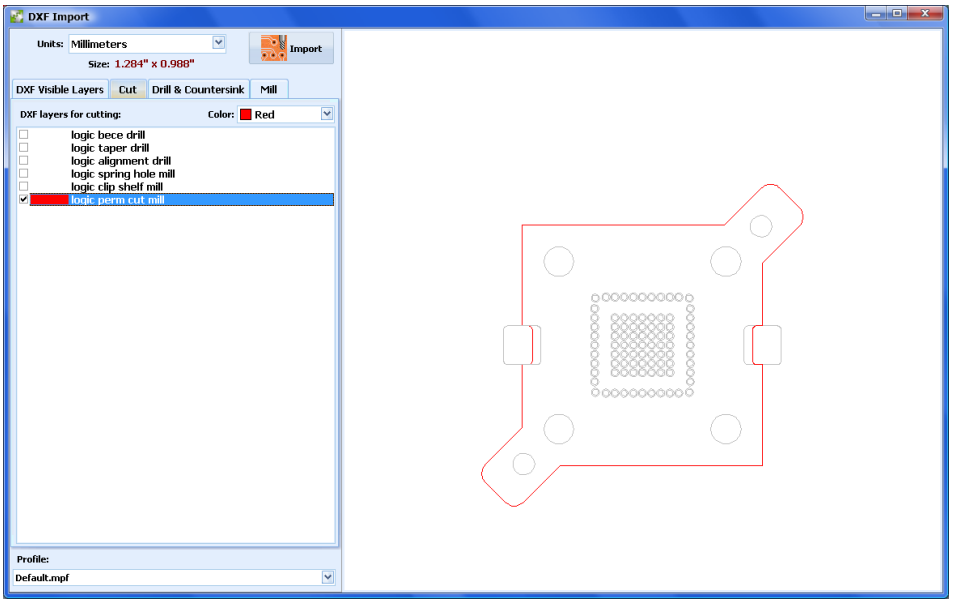
Button initiates DXF files import in PhCNC Pro. After DXF file selection a window for layers selection (for machining) comes up. Here are the conditions that are related to import of DXF files in PhCNC.

- 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.

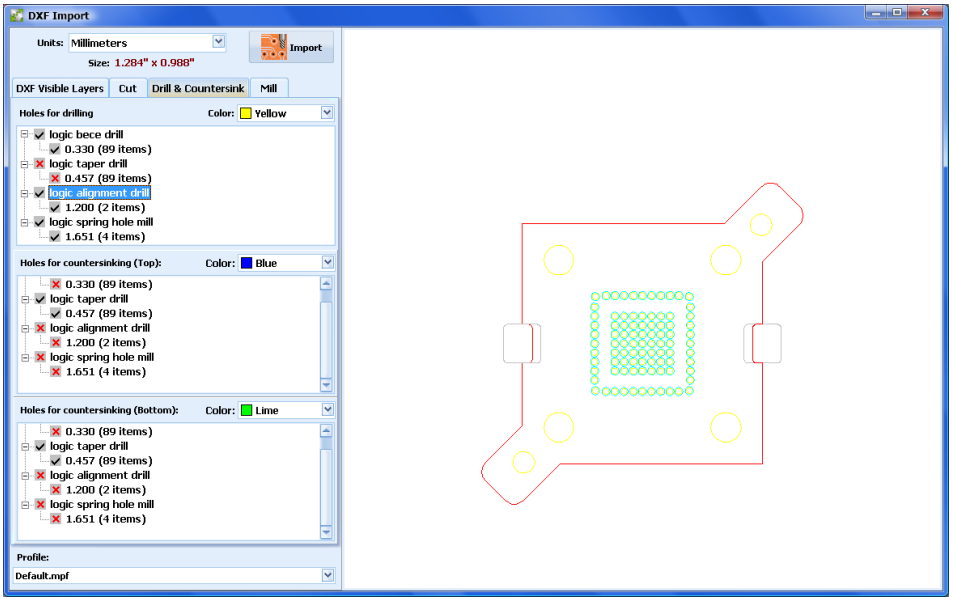


2.17.1 Selecting DXF layers for "Cut" operation



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)

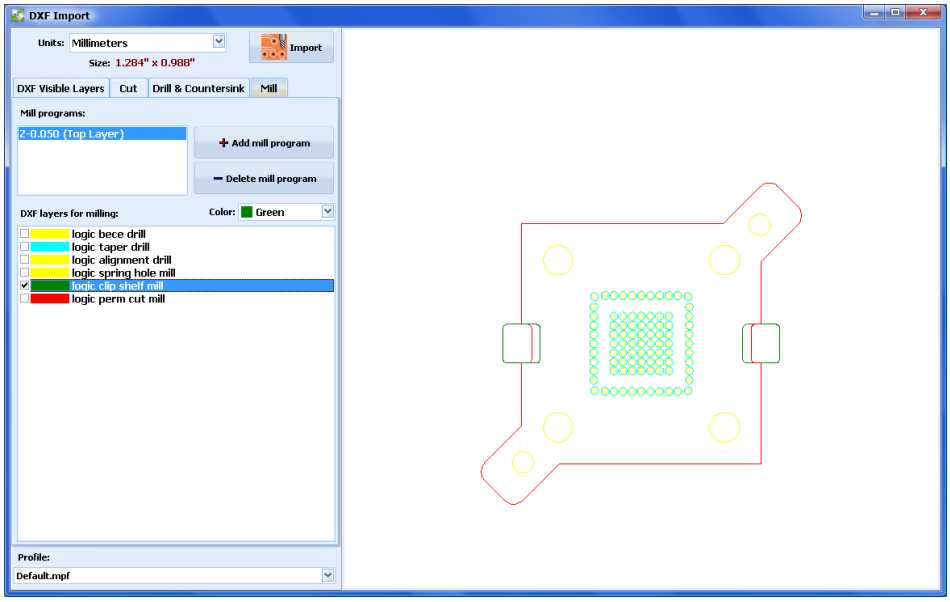
PhCNC is able to mill closed contours at pre-selected Z axis position (depth). For each different depth and each different layer (Top/Bottom) you have to set separate programs.

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.

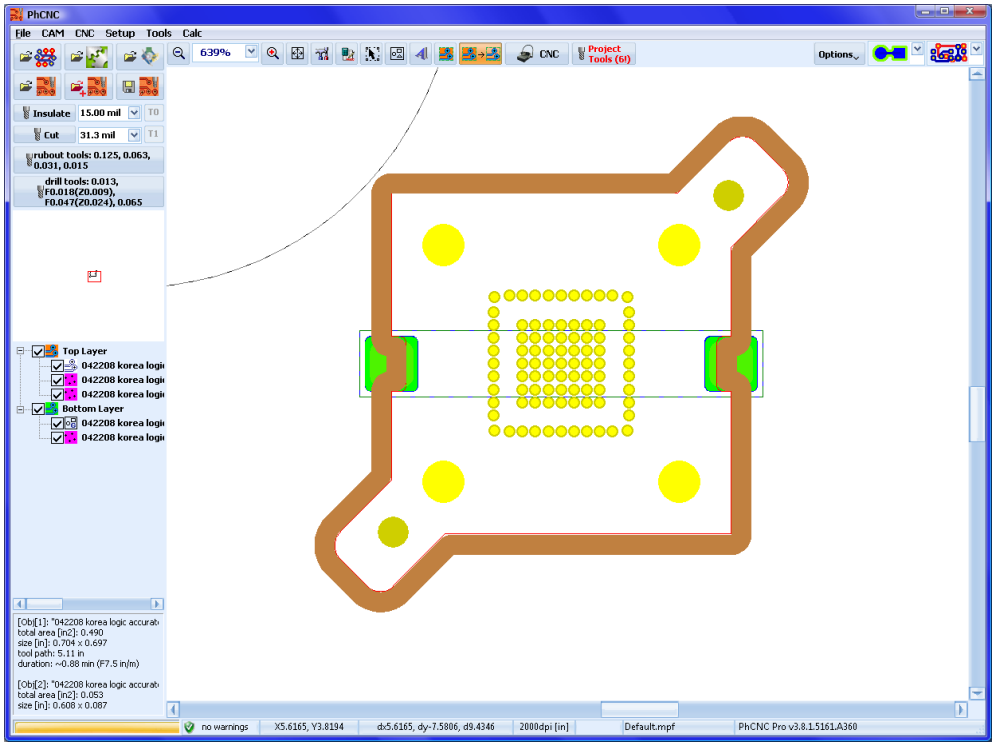
Once a program (Z depth/layer) is created it is necessary to select a corresponding DXF layer that will be machined with these settings (a DXF layer can be selected in more than one program).
First select the program from the list "Mill programs" and then select the DXF layer(s) that correspond to the settings.



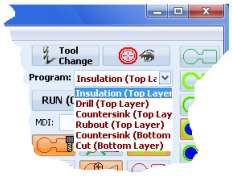
Each layer can be used in more than one program. After the import in PhCNC they will generate programs for layer insulation and rubout in CNC mode. The execution of the programs depends on the machine type. With a standard model equipped with depth limiter, PhCNC software will ask the operator to reset 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 will generate all machining programs needed, according to your settings.

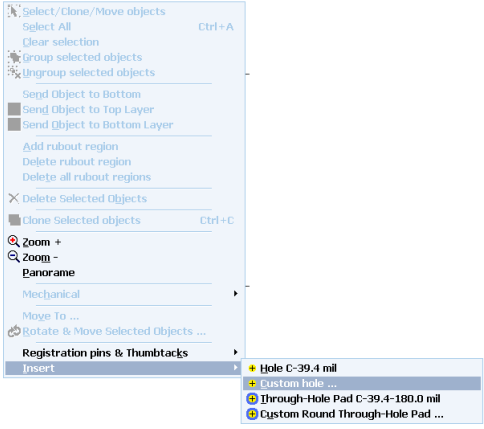


The machining programs will be one of the following:

- 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 (Pro version only)



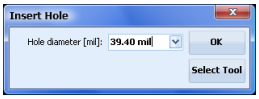
CAM mode of latest PhCNC Pro (v1.11.2+) allows you to insert additional holes and pads in order to meet the needs of fiducial registration process. This option can also be used for many other purposes like for instance when you need an additional hole or pad in your project. All inserted holes or pads are treated as separated objects and can be moved, grouped, selected…etc. as any object in CAM mode of PhCNC.

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

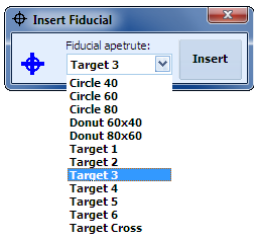
The process of inserting through-hole pads is the same as in 2.18.1
You have two choices, a current size or customized (that becomes current after usage).

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.

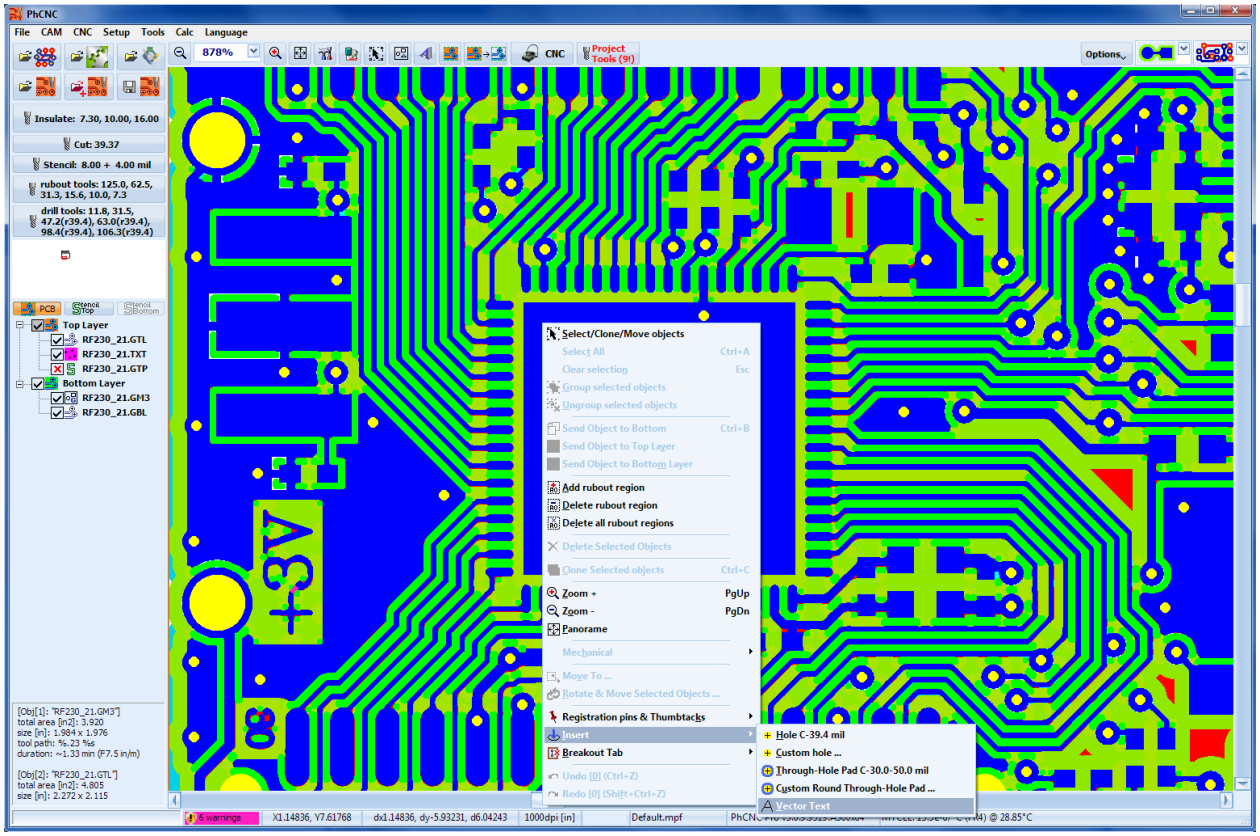


All apertures are predefined but adding new ones can be done by request.

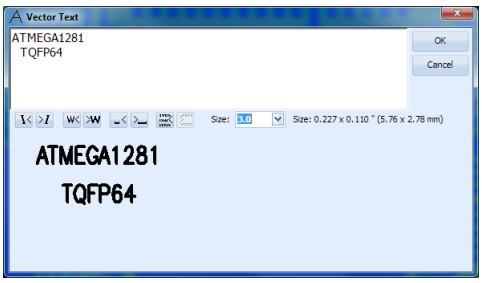
2.19 Vector Text insertion function (Pro version only)

PhCNC Pro (v1.13.2+) 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:



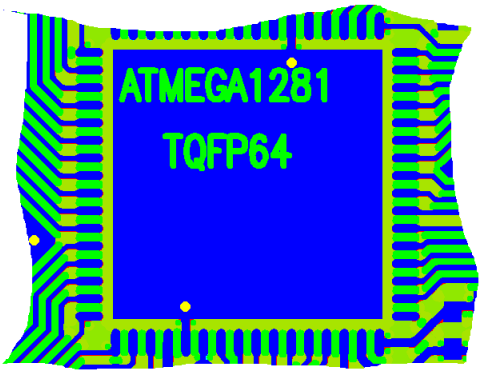
You will then be presented with the vector text window as seen below:



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.

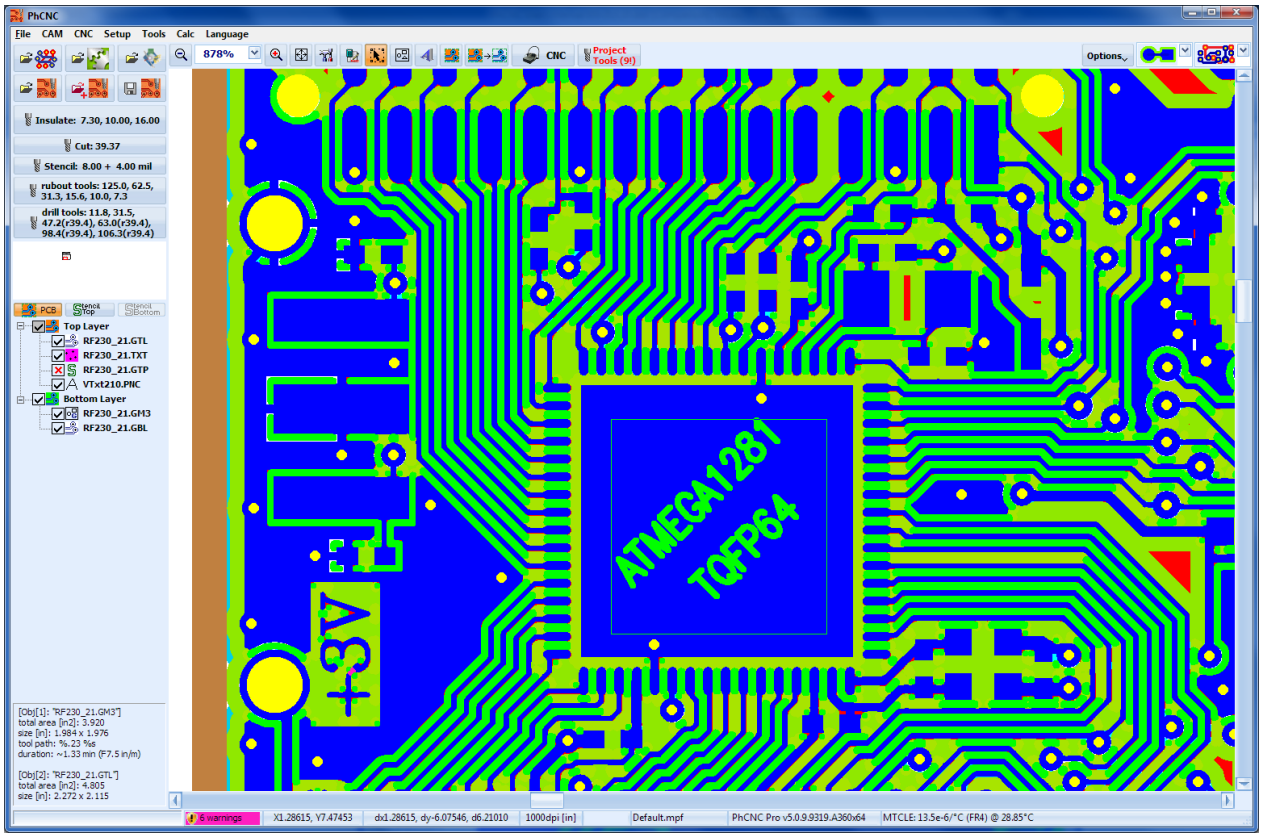
Buttons are for text slant (left or right).
Buttons are to change symbols width.
Buttons are to change the spacing between the symbols.
Buttons are to change the spacing between the lines (if there is more than one).

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.



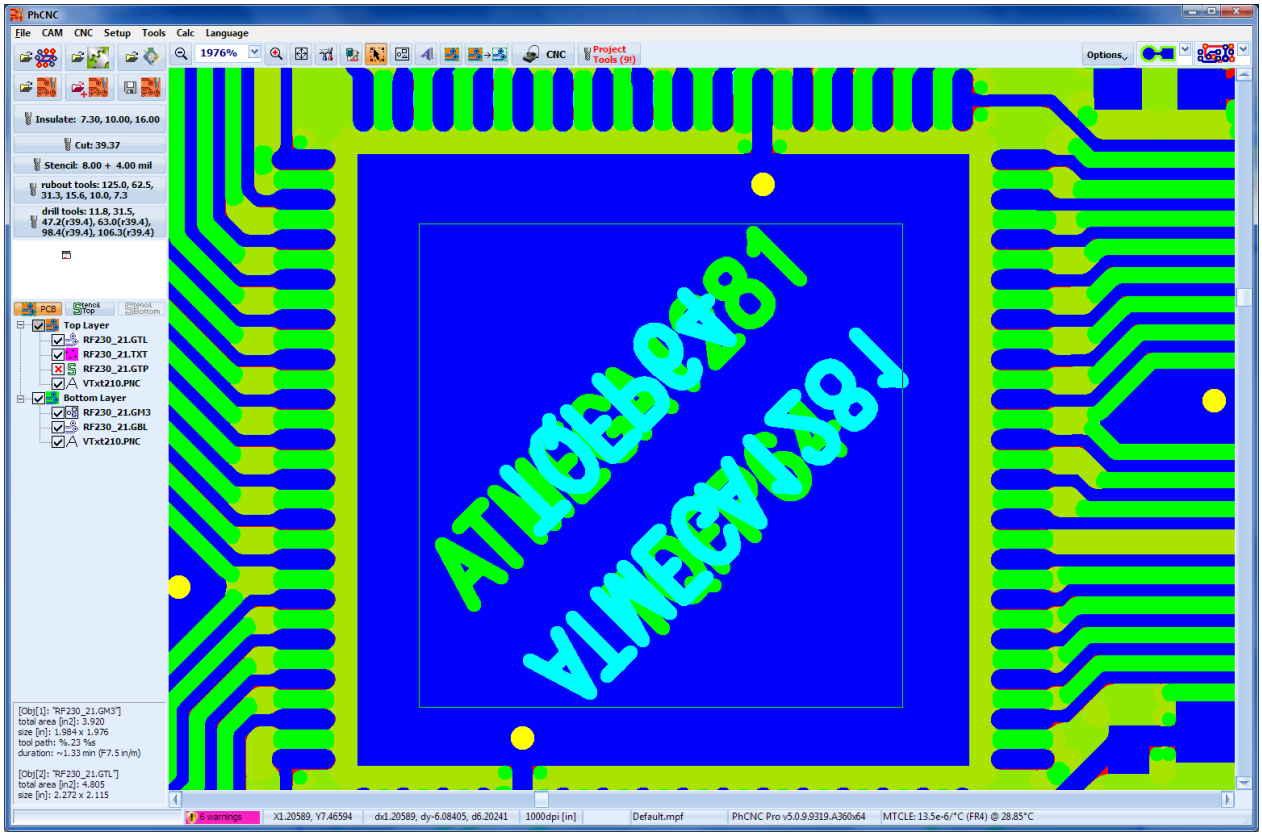
Inserted text objects can be manipulated just like other objects can they can be cloned, rotated, moved, grouped etc.

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

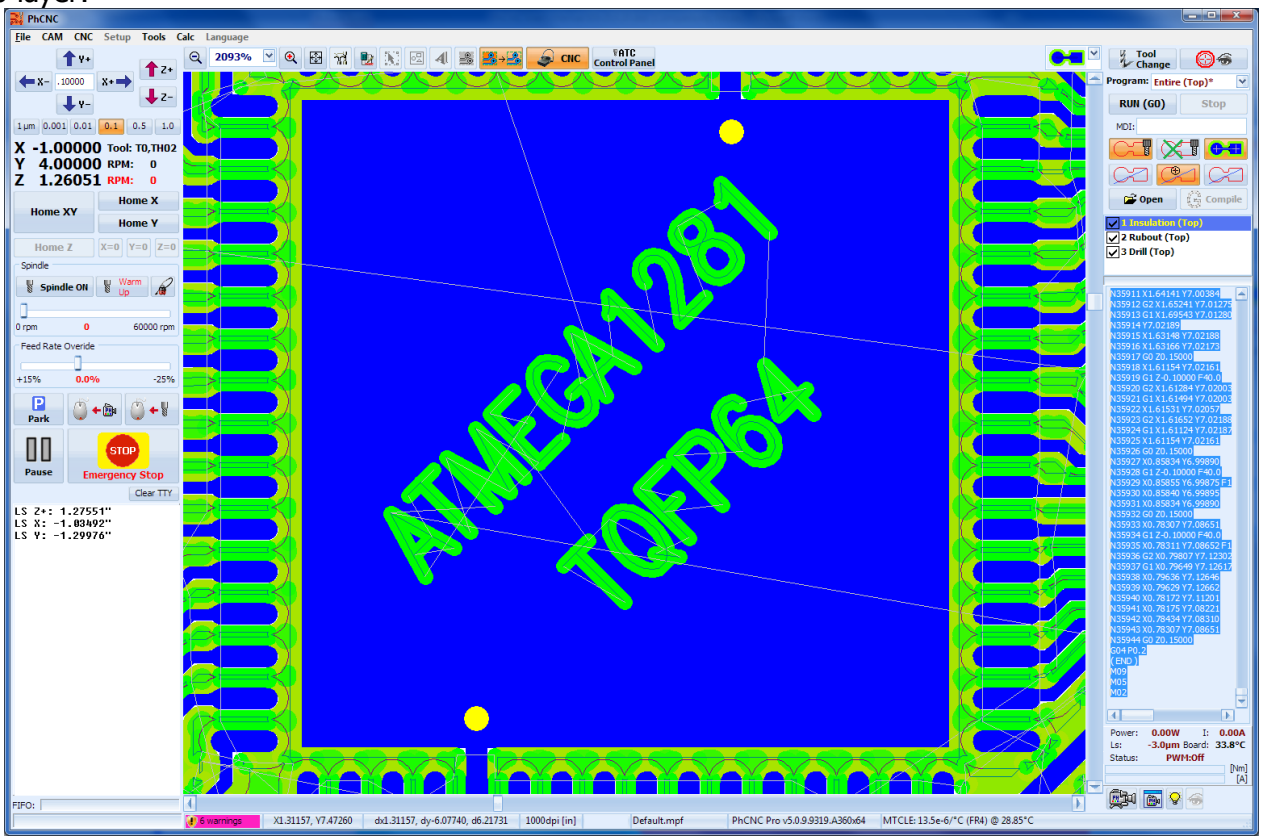


If you need to insert text in bottom layer, you have to use the (drag & drop) function.

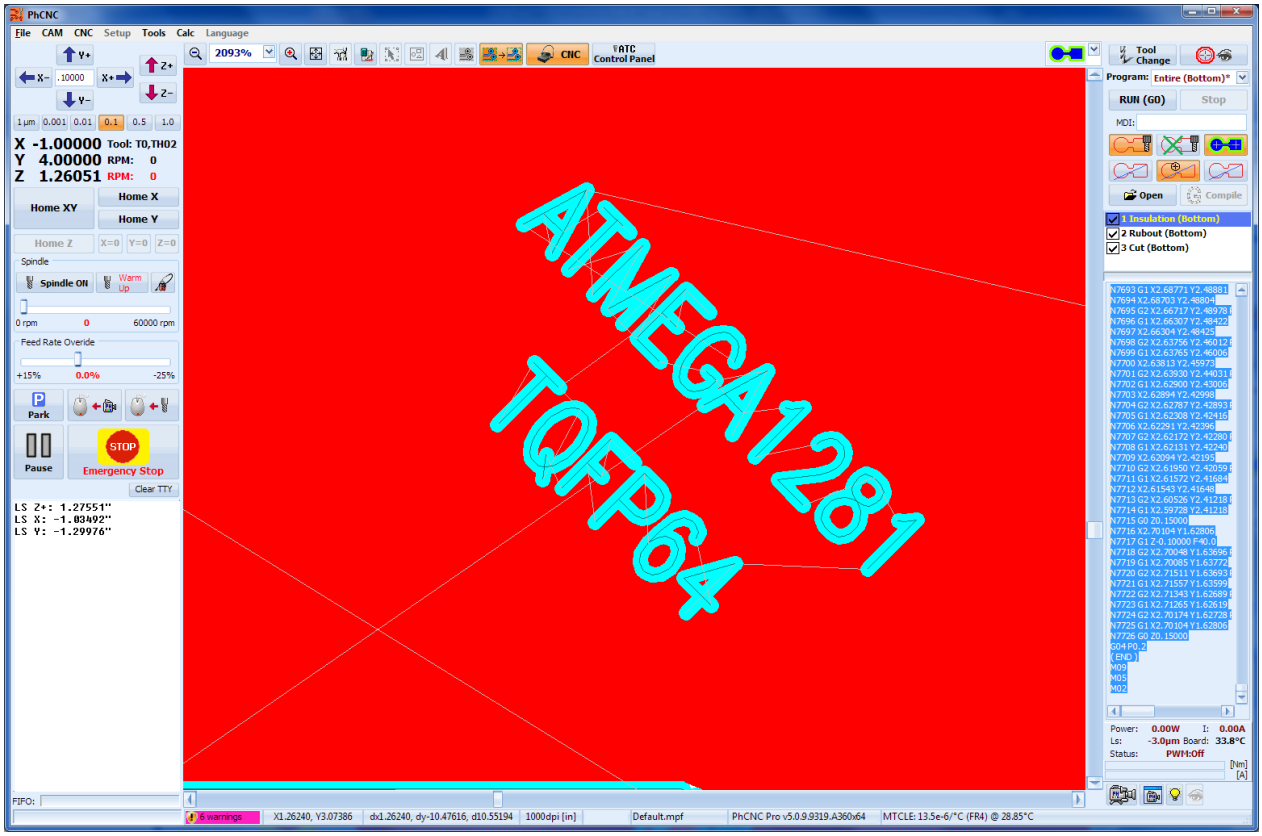
In the sample below text from the sample above has been cloned and moved to bottom layer.



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:



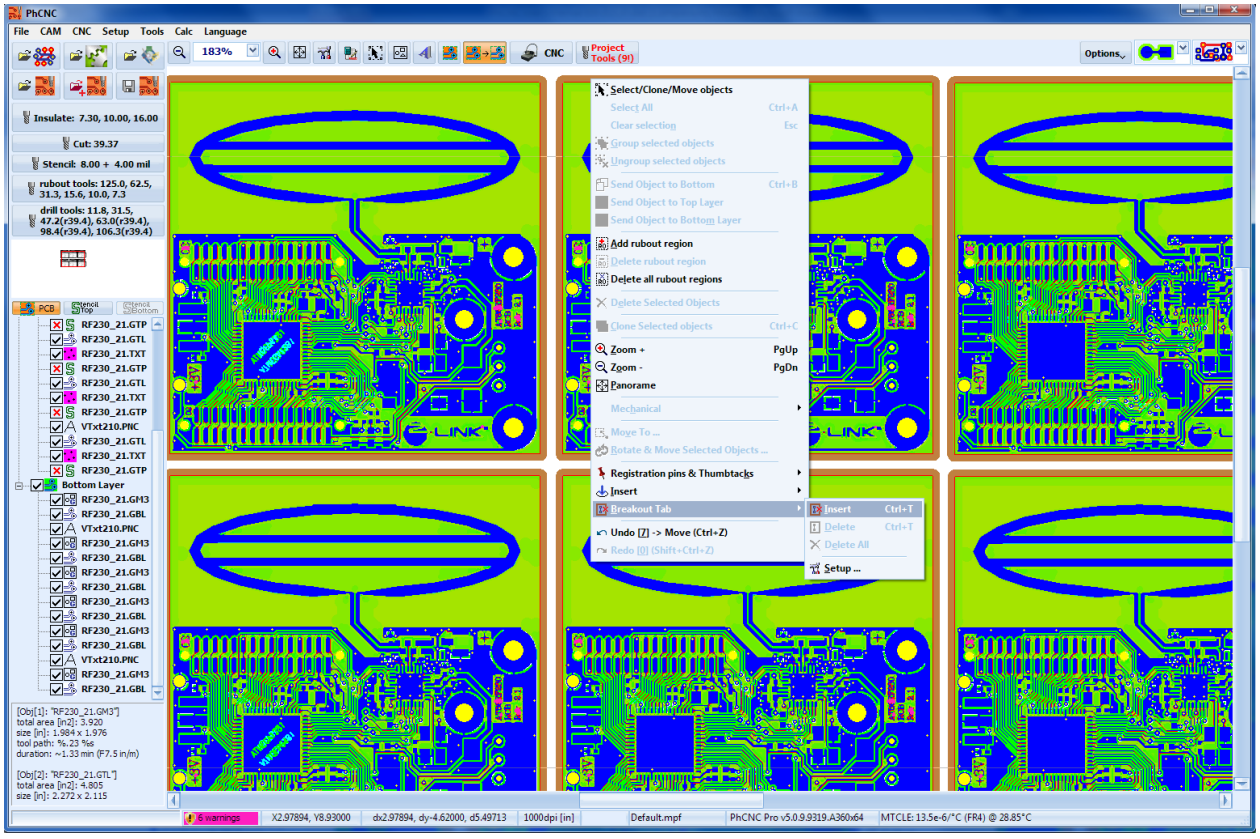
2.20 Breakout Tabs (Pro version only)

PhCNC Pro (1.14.4+) 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.

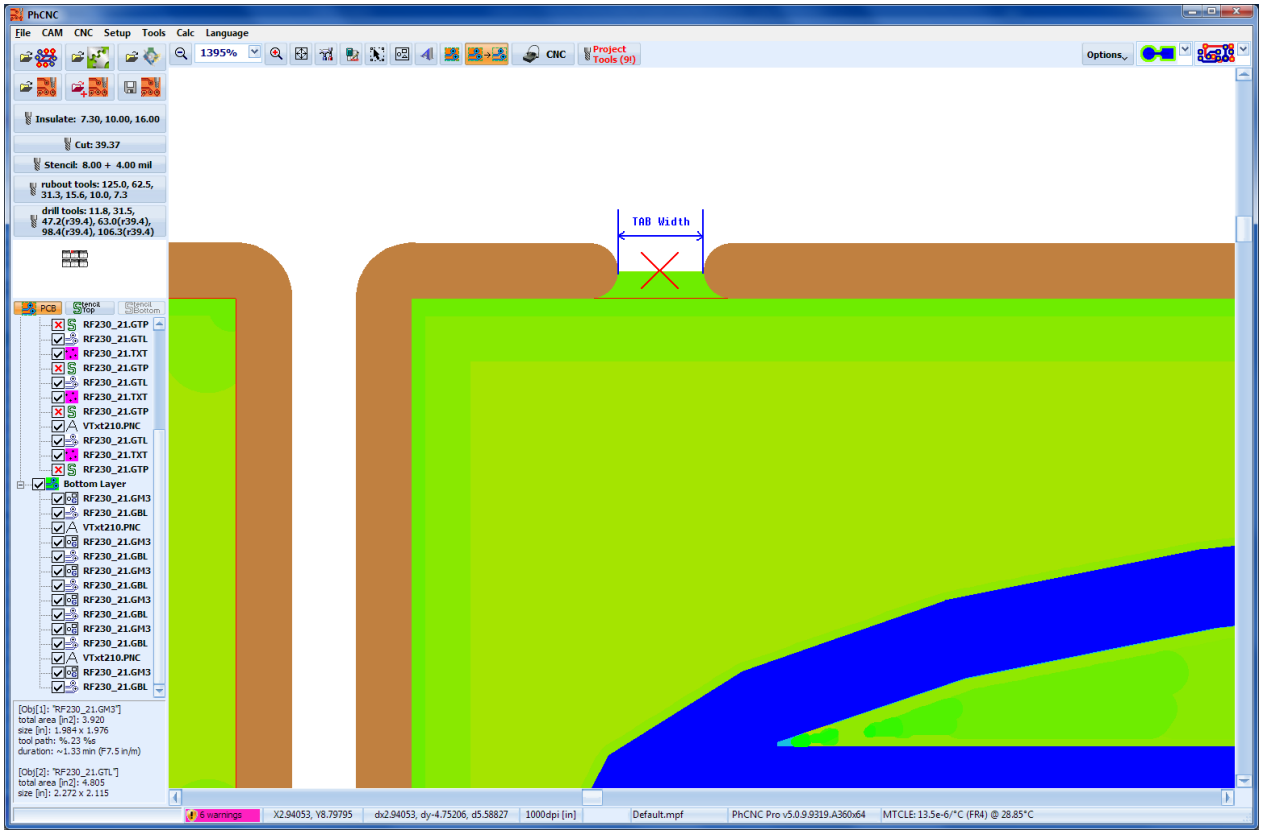
2.20.1 Breakout tabs insertion

Breakout tab insertions are possible in PhCNC 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:



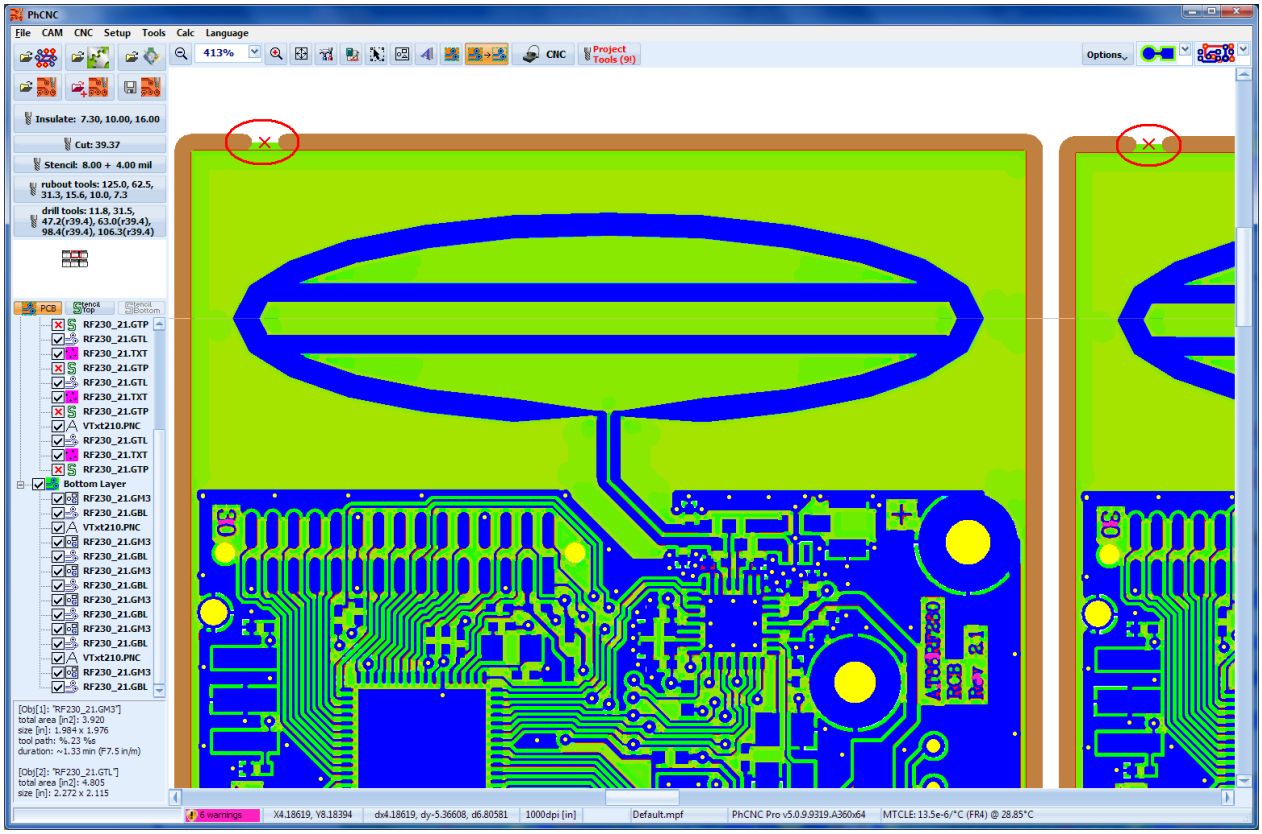
The "Insert" option is allowed only when the mouse pointer is on a cutout contour (mechanical layer).

Shown below is the inserted breakout tab:



Tab width is calculated by PhCNC 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 can not be inserted over another existing tab, or placed were 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

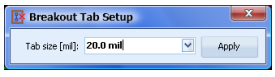
To remove a tab place the mouse pointer over the small "x" (tab center) and make a right click. A popup menu will then appear. Select option "Breakout tabs" from the menu then select option "Delete" from sub menu.

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 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 will remove all tabs. Therefore it is recommended to insert tabs after the selection of a cutout tool.

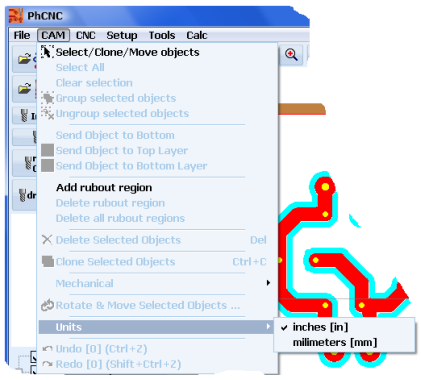
2.20.4 Breakout tabs complete removal.

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

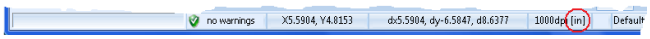
2.21 Units of measurement [mm]/[in]

PhCNC (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 (1.15.4+) now has an UNDO/REDO function. The size of UNDO/REDO buffer is 16 commands deep for PhCNC Pro and 2 commands deep for PhCNC Std.

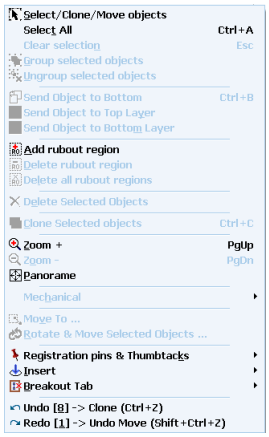
The following operations can use the UNDO/REDO function:

- 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 Pro only
- FIDUCIAL_ROTATE (rotating project - CNC mode – point two) available in PhCNC Pro 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:

- Standard PhCNC Undo/Redo commands are located in the CAM menu and can only be used with CAM functions.
- PhCNC Pro Undo/Redo commands are located in the CAM menu and the CNC popup menu for UNDO/REDO of Fiducial positioning commands.



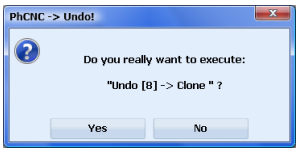
Each Undo/Redo command contains the information for operation that is reversed, and the digit in square brackets shows the number of commands in the corresponding buffer.

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!

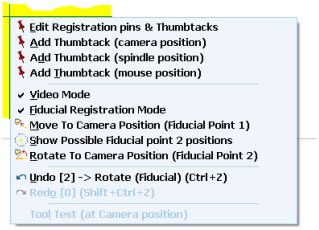
REDO reverses the last UNDO command, but if you have another command that is in the UNDO list, the REDO buffer will be cleared.

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 Pro):



2.23 Forced Insulation

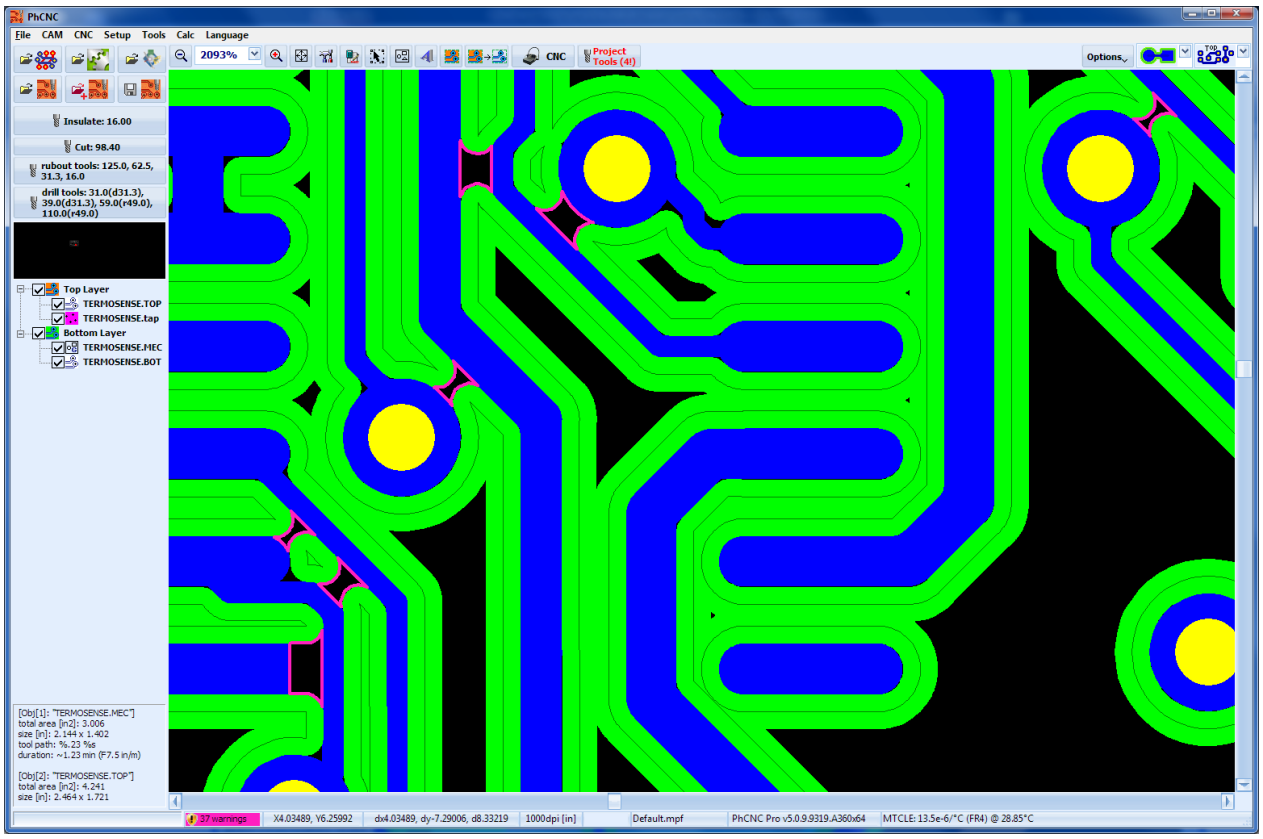
PhCNC has a function "Forced Insulation" that allows the system to handle sections of the design marked by DRV (See 2.13 Design Rules Violation) as impossible to insulate. The standard response of PhCNC tool path generation doesn't allow the tools to penetrate inside the traces/pads contours in this 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:

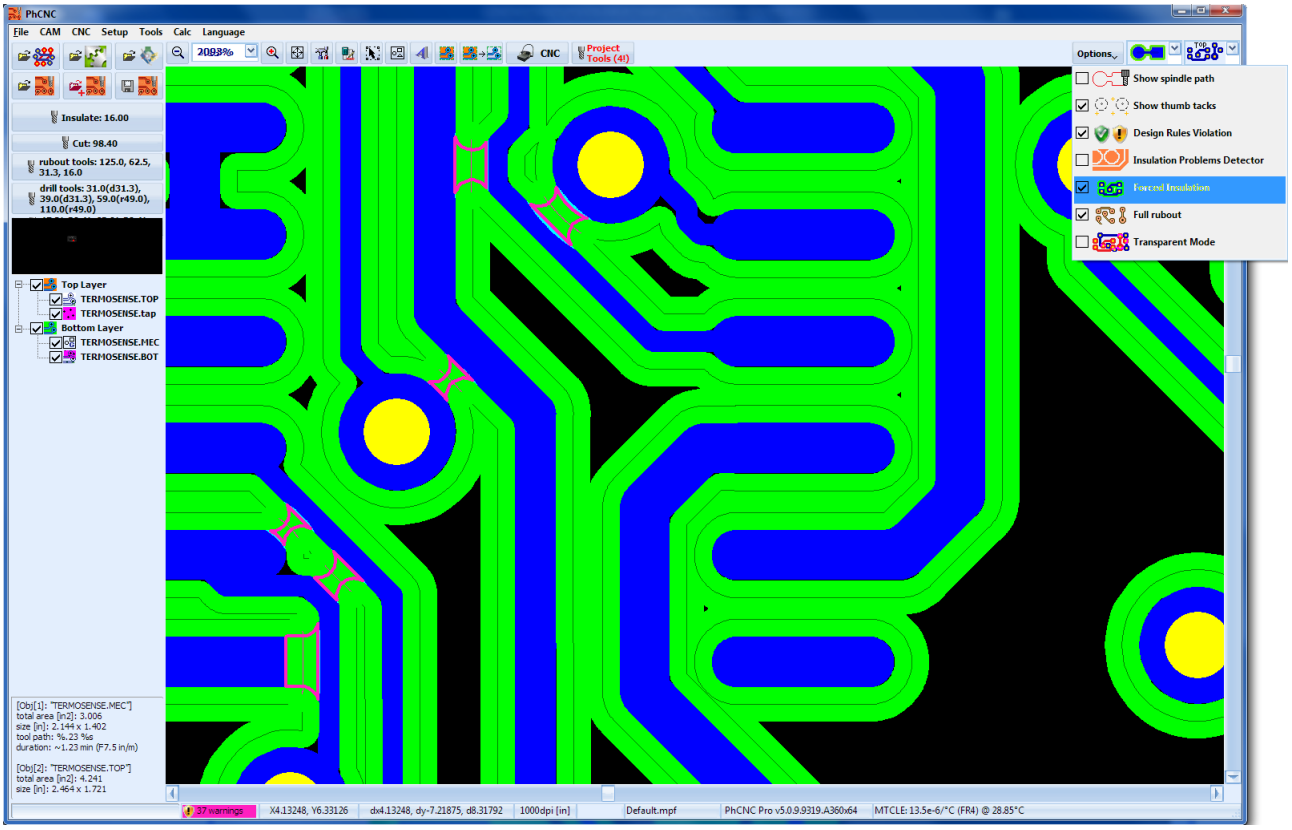
- Calculates tool paths used in "Forced Insulation". If the tool paths are never calculated the first activation will do so. If DRV function wasn't activated it will start it.
- Shows/hides tool paths related to "Forced Insulation".

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:

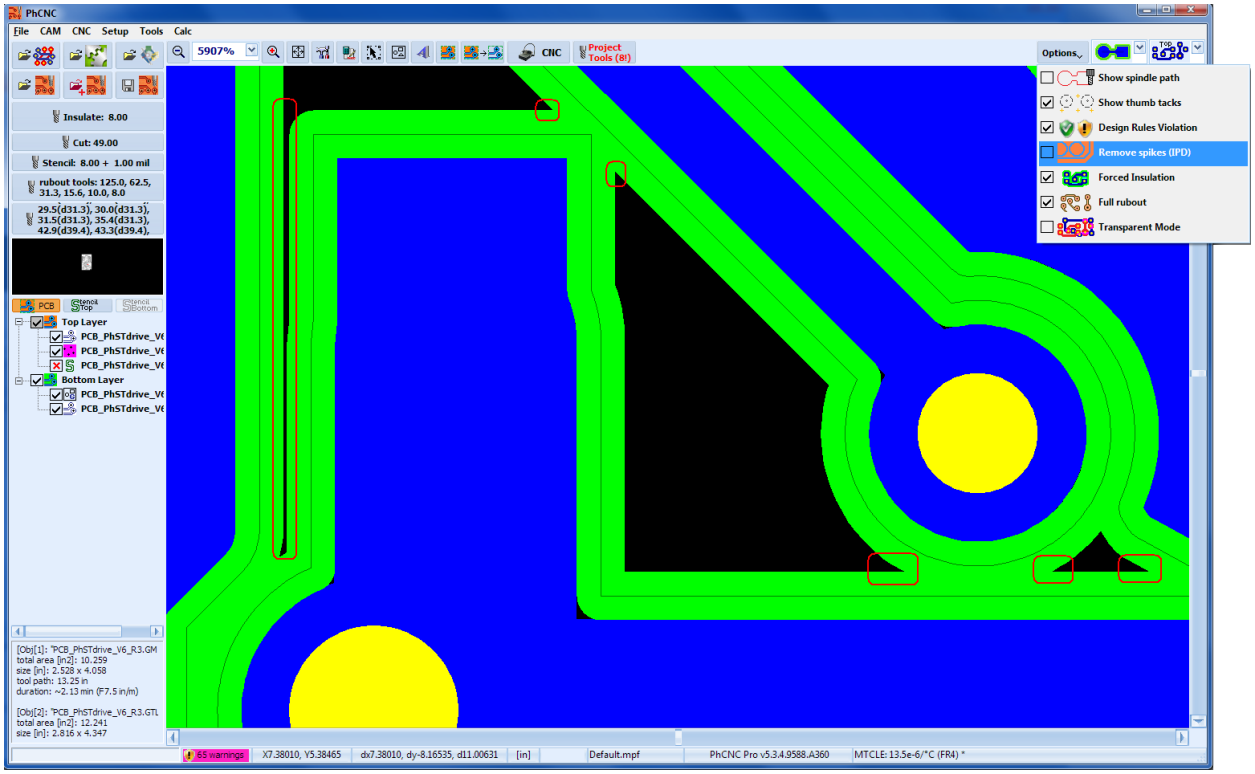


Light blue (default) strips shows additional copper removed from the design features when zoomed in to view the "Forced Insulation" paths.

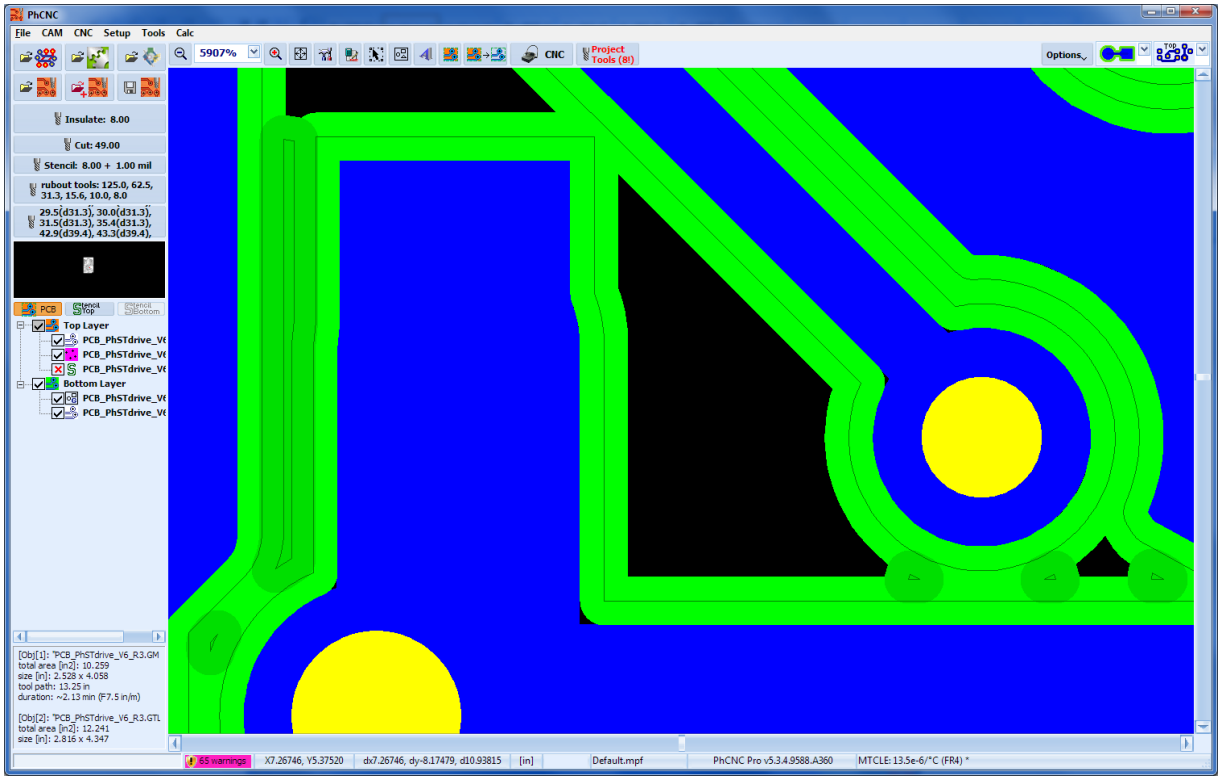
Button will also serve as a switch to include/exclude the problematic sections in the machining process. "Forced insulation" tool paths are not automatically recalculated when you change the insulation tool diameter. If you change the insulation tool diameter you need to run the DRV function again to 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)

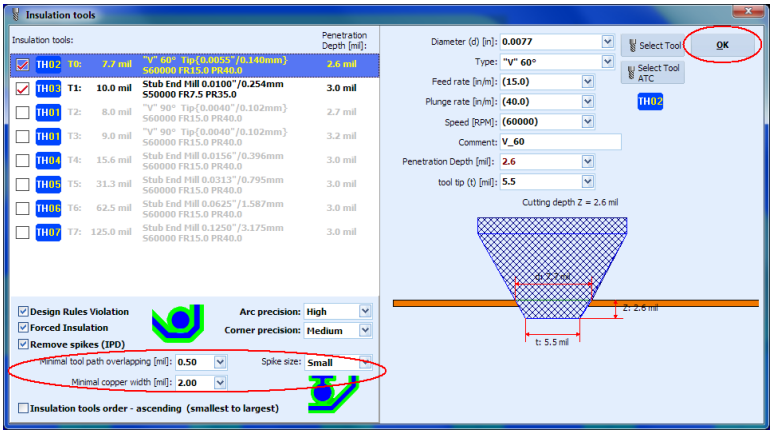
When the insulation only is used (not full copper rubout) very tiny copper strips may be left over from machining (hair sized). They do not have enough foundation to the board substrate and usually become free moving copper hairs that can create a condition for fine (hard to see) shorts on the board. 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 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:



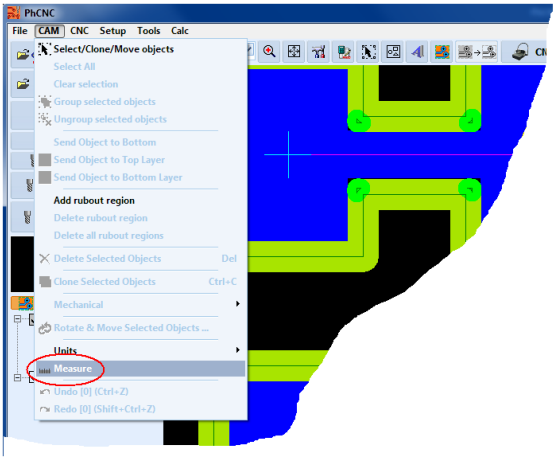
The parameters you may set are:

- 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).

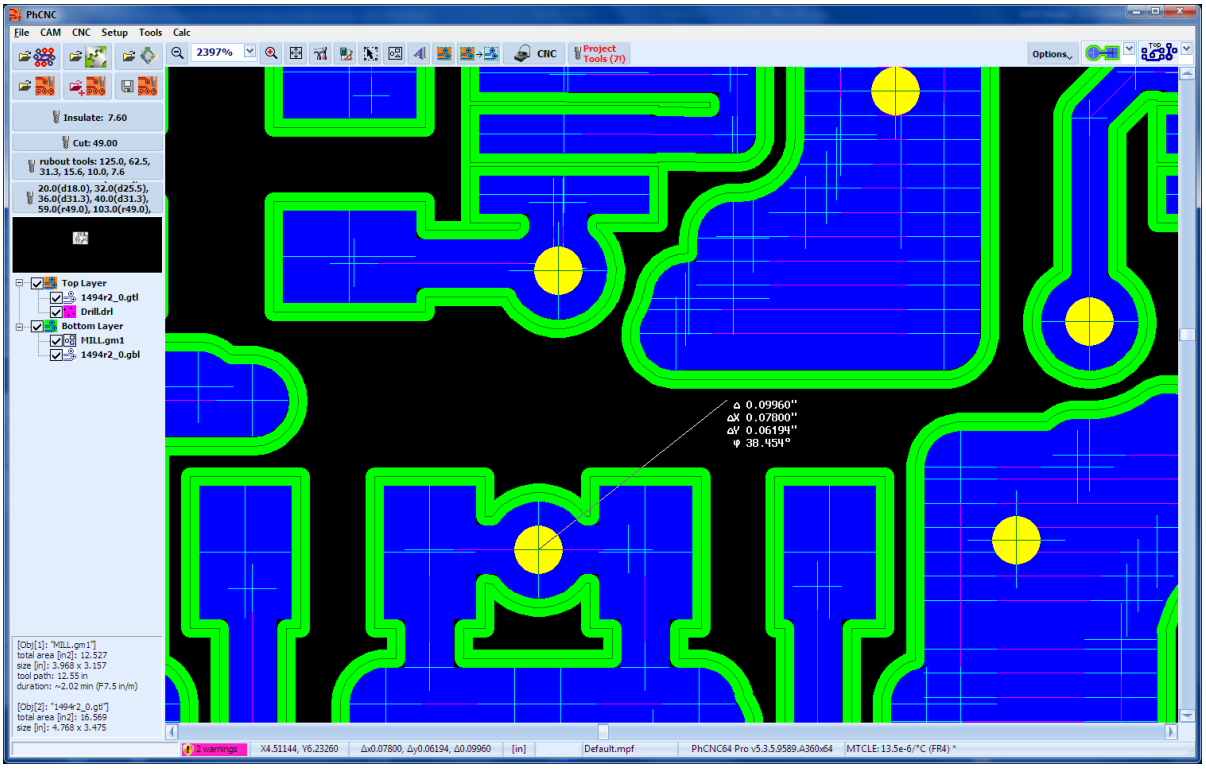
As a bottom line, using IPD when you do not use full copper rubout is very recommended.

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.

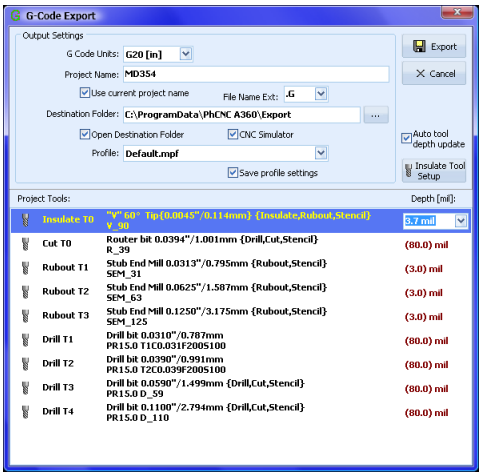


On screen information contains the distance, X difference, Y difference and the angle related to X axis. It is shown in the measurement units selected (inch/mm).

2.26 G-Code Export (PhCNC LS/LSSE)

The software revision named PhCNC LS/LSSE (Std & Pro) generates G-Code that can be executed on almost any CNC machine that accepts G & M codes.

The button in PhCNC LS/LSSE calls a window for G-Code export:



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

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

* Only with PhCNC LS/LSSE Pro

Basic parameters that you must determinate are:

G-Code Units:
Units of measurement that will be generated in G-Code program(s), G20 ([in]) or G21 ([mm]). By default measuring units are in [in].

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 shown in (APPENDIX-D) 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).

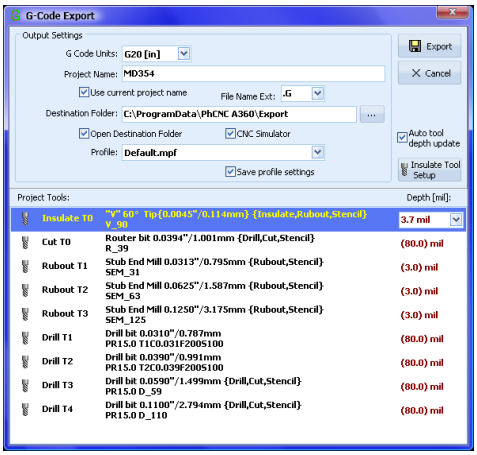
The operator will be able to make changes (if needed) in the selected tools penetration depth (see paragraph 2.4.6 Project Tools)

Insulation tool Z depth:
Systems with depth limiter:
Z depth penetration is set in the Setup menu, but the actual penetration is the one set by the limiter micrometer.
Systems without depth limiter;
When the V tool is chosen, the tool penetration is calculated by PhCNC, 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.

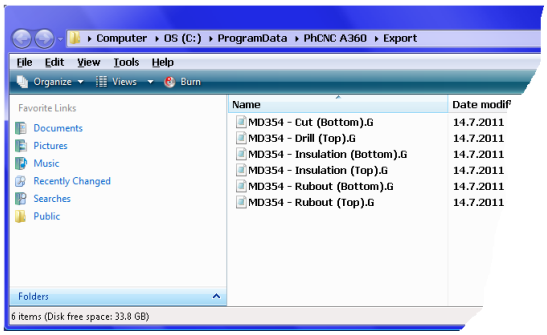
Rubout tools Z depth:
This sets the penetration depth of rubout tools.
Important:
Our rubout program always uses the insulation tool as the first tool for rubout. The following tools are normally stub end mills and have to have the same penetration settings.

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

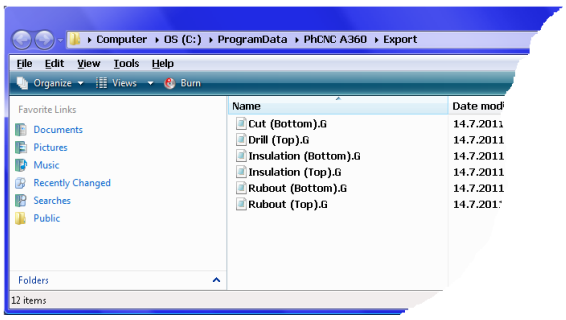
Example:
Project name: MD354,



Generated files are:



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

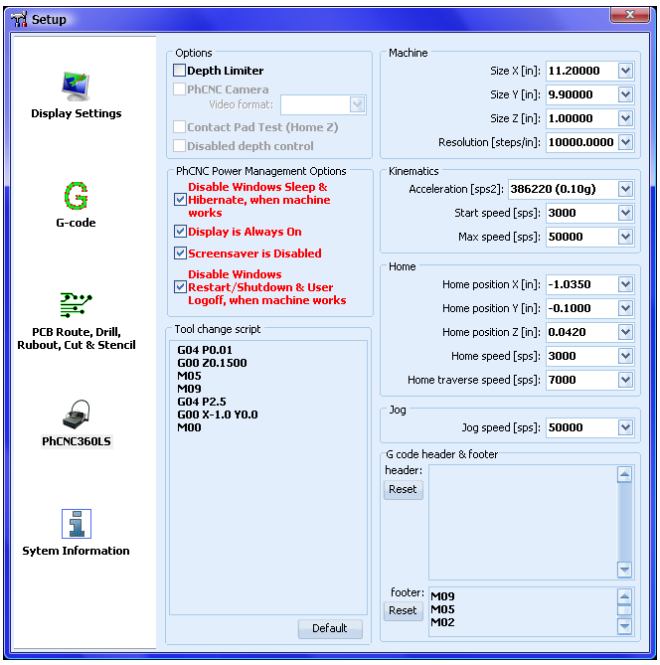


If "Use Project Name" is selected, the project name will be the name of current project (if any). The files extension name is .GNC (by default) but can be changed if needed.

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

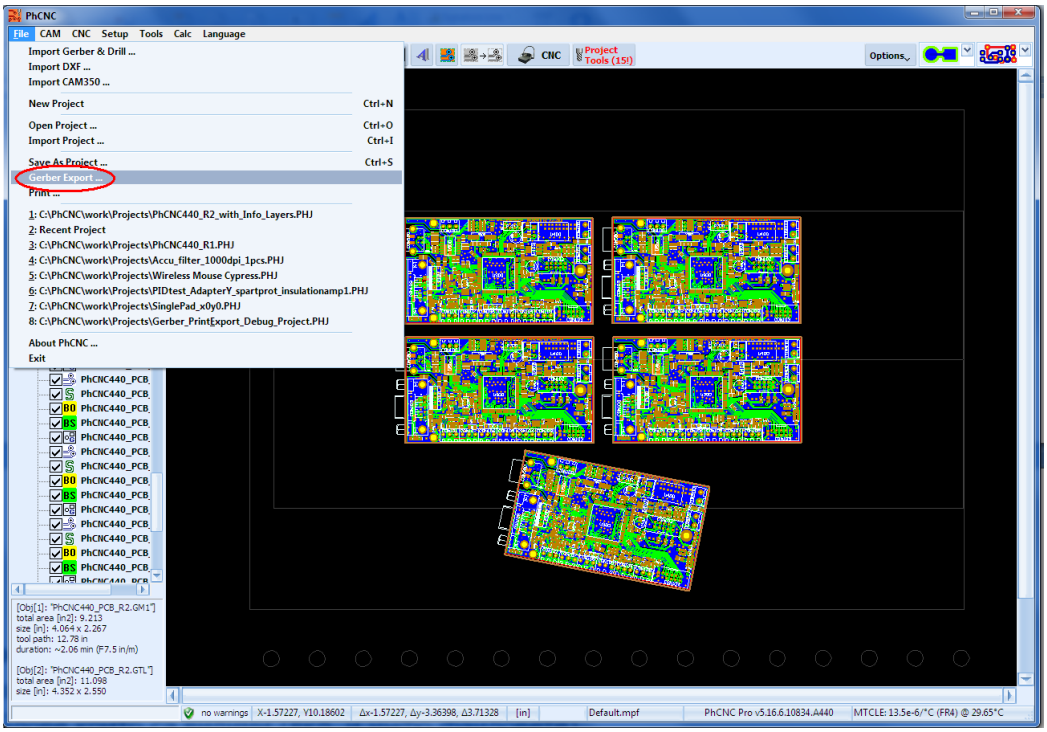
"CNC Simulator"
If this option is selected the system goes into CNC simulation mode after the G-Code programs are generated. In this mode all programs can be started.
The CNC Simulator – for PhCNC LS/LSSE is a hardware USB dongle that is supplied together with the PhCNC LS/LSSE and LS/LSSE Pro software.

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

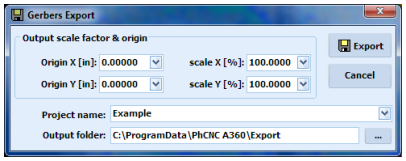

Setup
Display Settings
G-code
PCB Route, Drill, Rubout, Cut & Stencil
PhCNC360L5
Sytem Information
Options
Depth Limiter
PhCNC Camera
Video format:
Contact Pad Test (Home Z)
Disabled depth control
PhCNC Power Management Options
Disable Windows Sleep & Hibernate, when machine works
Display is Always On
Screensaver is Disabled
Disable Windows Restart/Shutdown & User Logoff, when machine works
Tool change script
G04 P0.01
G00 Z0.1500
M05
M09
G04 P2.5
G00 X-1.0 Y0.0
M00
Default
Machine
Size X [in]: 11.20000
Size Y [in]: 9.90000
Size Z [in]: 1.00000
Resolution [steps/in]: 10000.0000
Kinematics
Acceleration [sps2]: 386220 (0.10g)
Start speed [sps]: 3000
Max speed [sps]: 50000
Home
Home position X [in]: -1.0350
Home position Y [in]: -0.1000
Home position Z [in]: 0.0420
Home speed [sps]: 3000
Home traverse speed [sps]: 7000
Jog
Jog speed [sps]: 50000
G code header & footer
header:
Reset
footer:
Reset
M09
M05
M02

2.27 Gerber Export ...

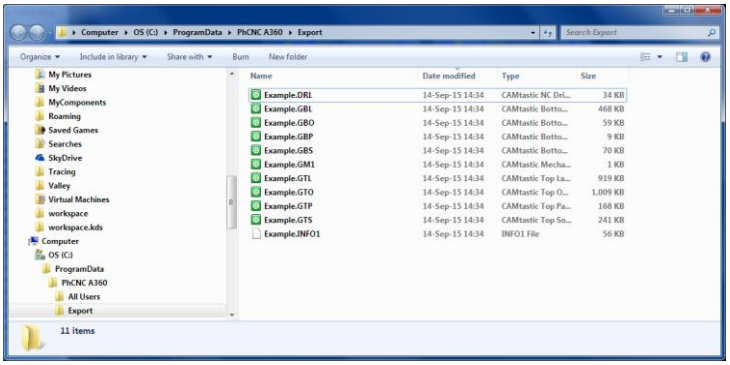
PhCNC now allows an export of all Gerber and Excellon drill files in the current project . All visible files are exported including the info files that are not involved in the machining process.



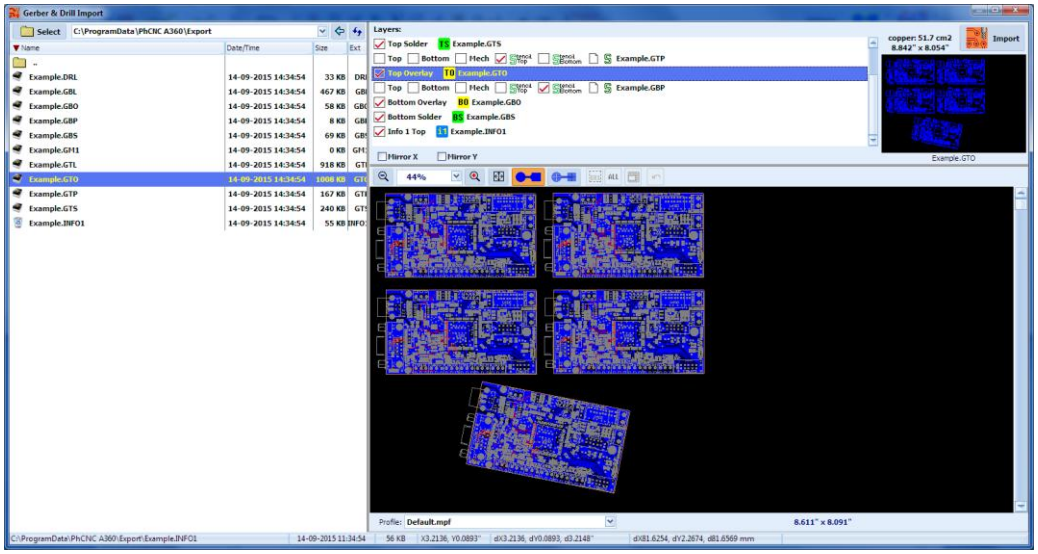
Gerber Export window allows to scale the files exported in Gerber and Excellon format (if needed).



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



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

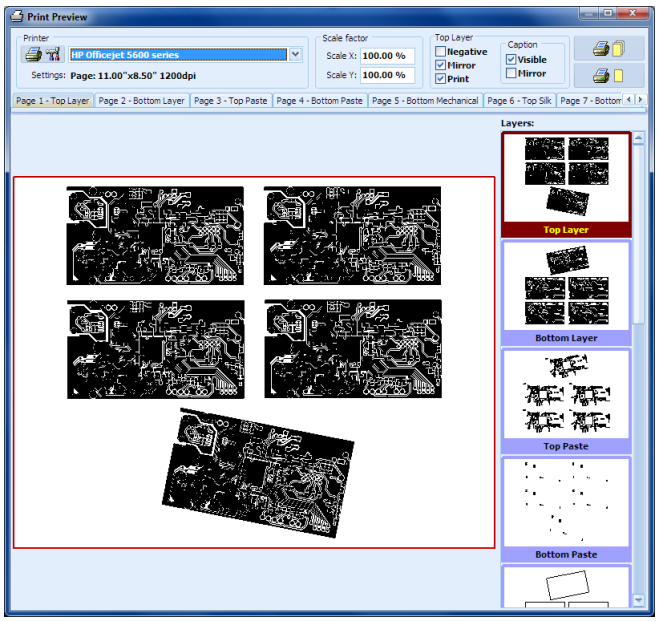


Import & Export Gerbers & Info layers - video:
https://youtu.be/huY_sy04yi4

2.28 Print

PhCNC now 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



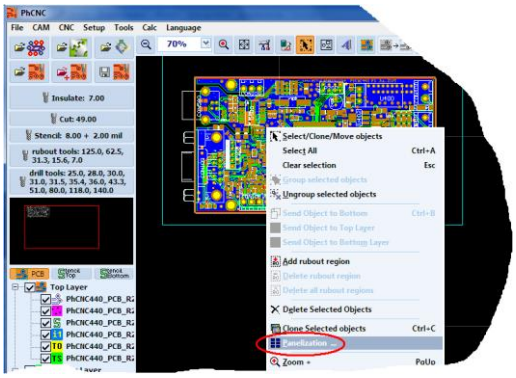
Layers list include all visible layers in the project. It is advisable to disable (hide) all layers that you don't have an intention to print. In the center window you will see the selected layer.
Using left double click on the layers name is the fastest way to enable/disable the layer for printing.
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 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 it's settings, missing DLL file
or non existing network printer or other problems related with the printer - PC connection.

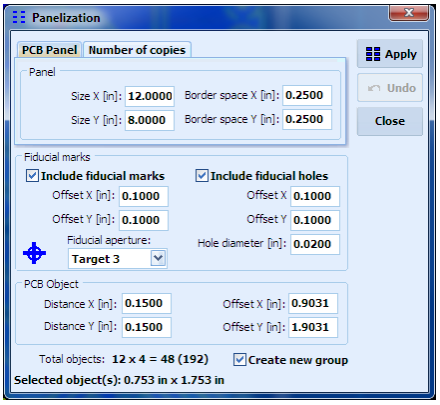
Print preview video:
https://youtu.be/zFNuUN_GkVo

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 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

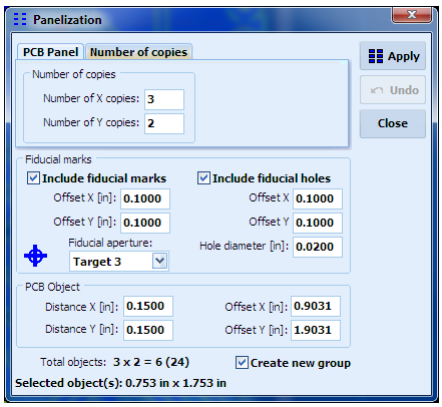
"PCB Panel" mode allows placing the object(s) in a panel with predefined size (Panel->Size X/Y) and on certain space from the physical edges of the panel (Panel->Border space X/Y). The number of copies is calculated based on:

- 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

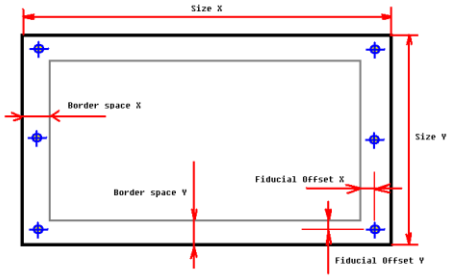
This is the easiest way to create the desired number of copies of your project entering the repeats on X and Y axes.



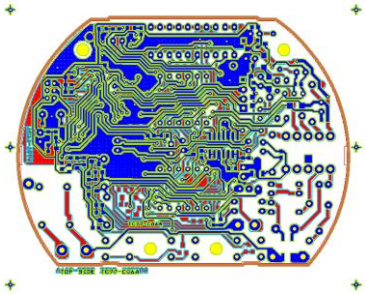
2.29.3 Fiducial marks

PhCNC allows inserting of targets & holes as a 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.

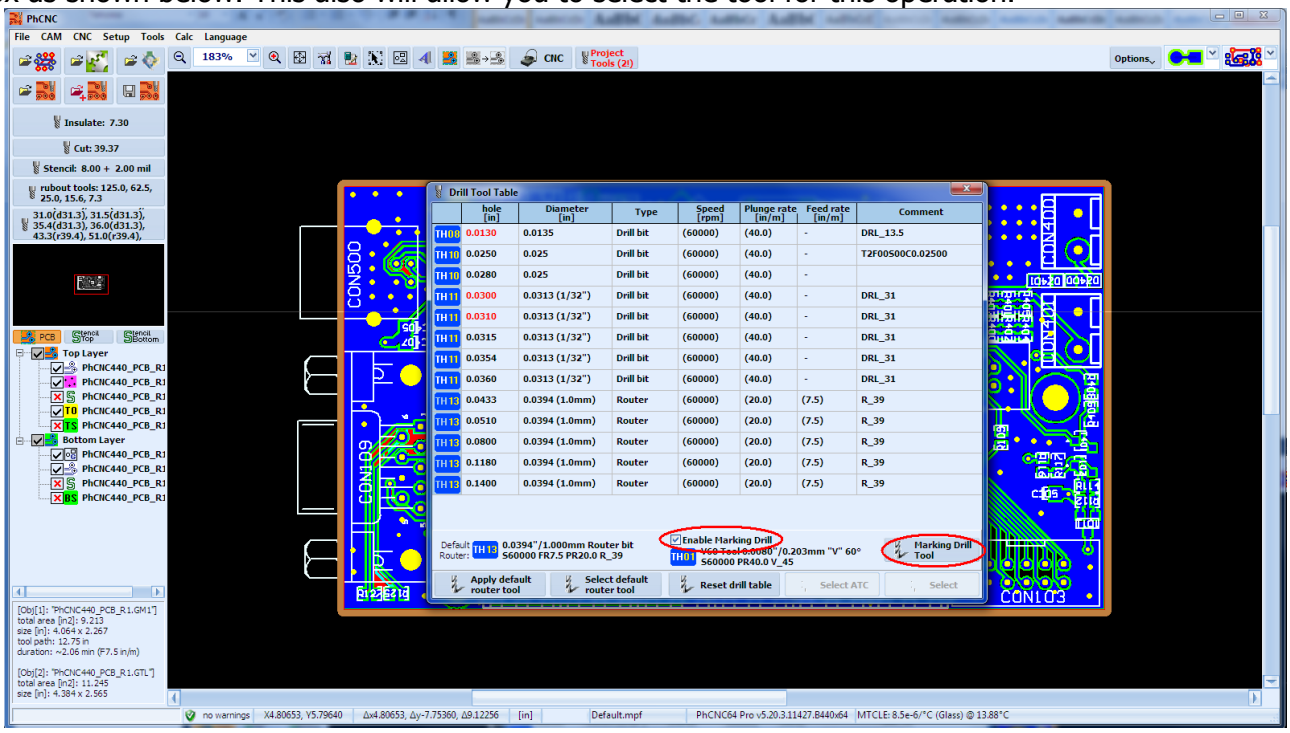
2.29.5 Create new Group

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).

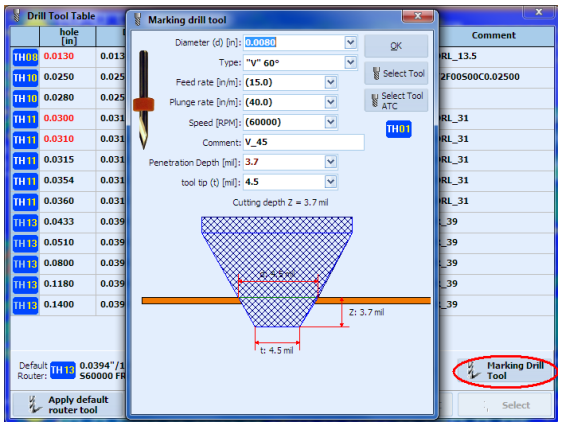
Panelization video:
https://youtu.be/O2ziZLAYVwg

2.30 Marking Drill

PhCNC 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.

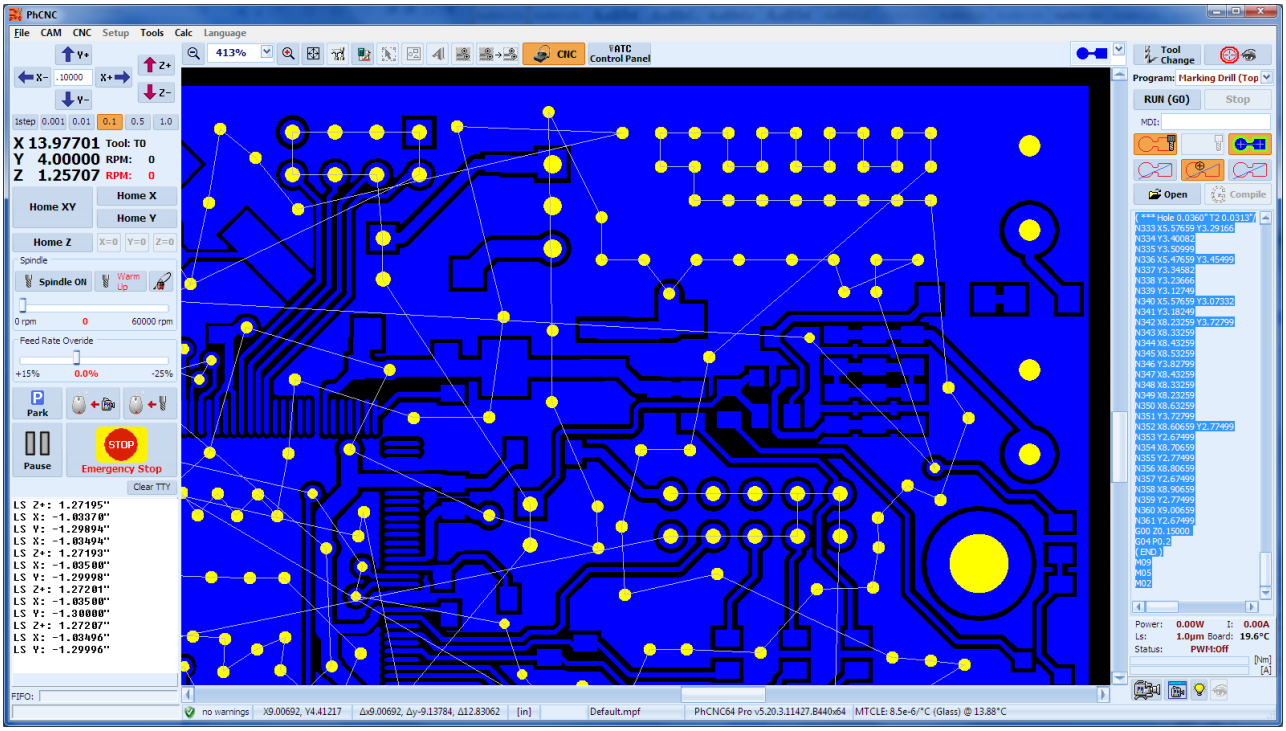


Marking Drill (Marking Drill Tool Setup):

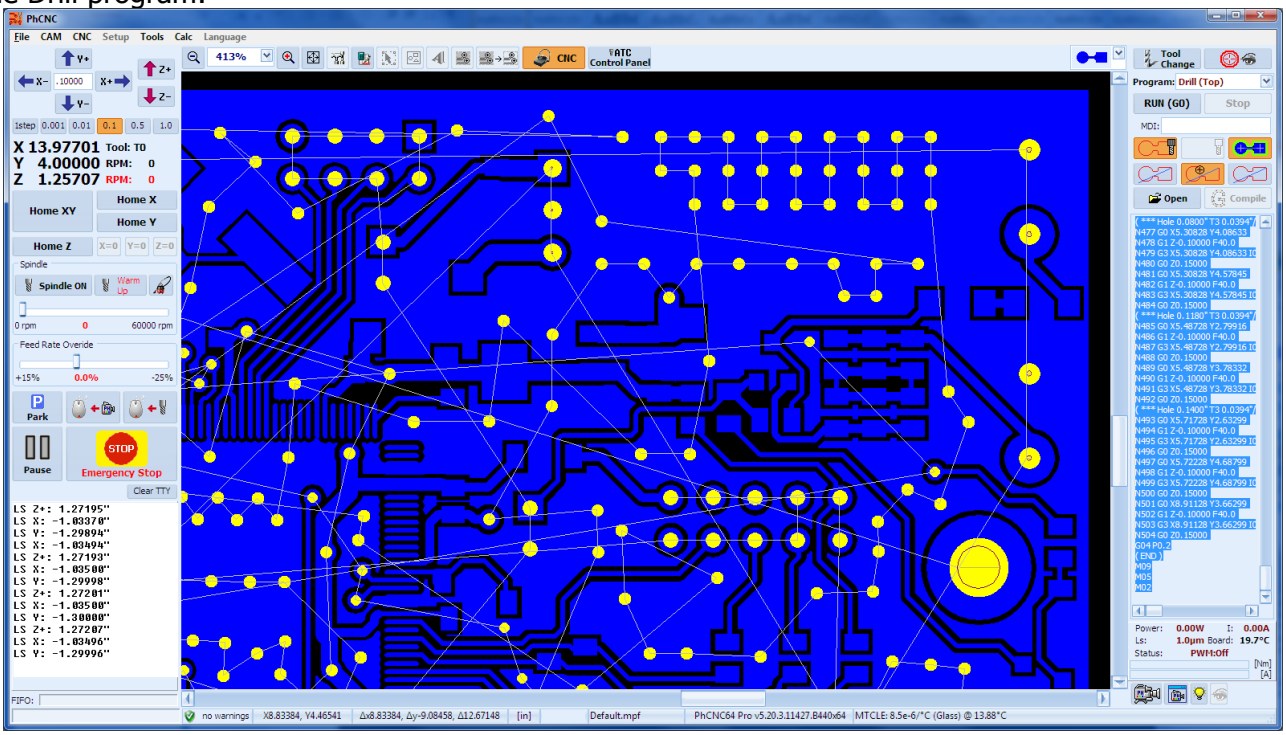


Notes:
The Marking Drill program will be applied only for the holes, for which drill bit will be used. The (usually larger) holes for which router bit is selected and the slot holes are excluded from the Marking Drill program.

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



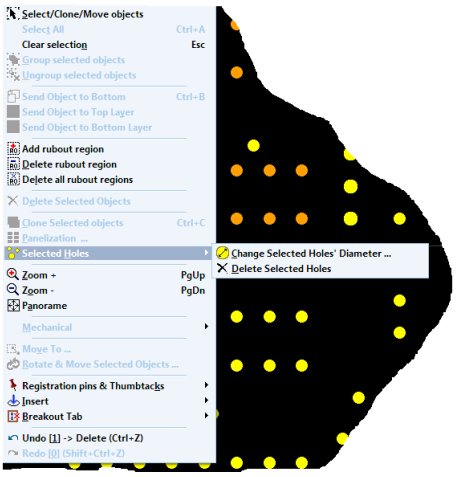
The Drill program:



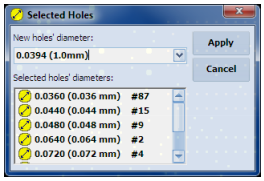
In the "Entire (Top)" & "Entire (Bottom)" ATC programs, the Marking Drill program will be executed before Drill program.

2.31 Holes Properties

PhCNC now allows selected holes diameter change or removal.
In CAM mode, the operator have to push and hold "R" key on the keyboard and using the mouse can create multiple rectangle regions. All holes inside the regions will be selected and will appear in different color. Using the "Selected Holes" from popup menu the operator can change the diameter or remove them.

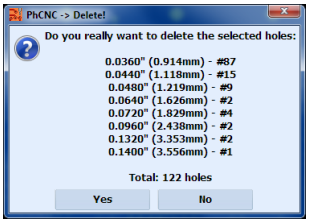


- If the diameter change is selected an additional window will appear to enter the changes.



The window contains all selected holes and allows a manual diameter changes or link to the diameter of the existing selected holes.

- If the holes removal is selected PhCNC will require additional conformation for the process.
A list of selected holes will appear including current diameter and the number of them.



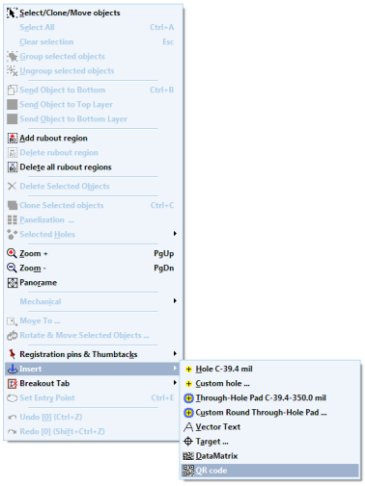
Notes:
- All changes can be reversed using Undo (Ctrl+Z)
- If the project has multiple copies the changes will affect all of them, PhCNC does not allows the copies of the same project to be different.

Video #1 - holes & regions selection:
https://youtu.be/j3JyJfE8SVU

Video #2 - holes selection & panelization:
https://youtu.be/nFny2D8eqBo

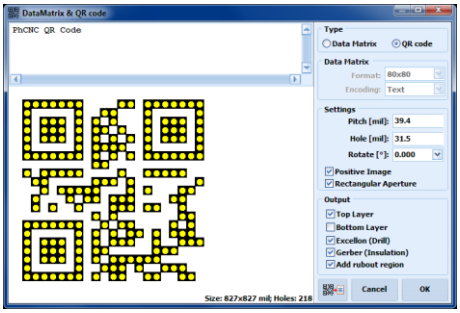
2.32 QR Code

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



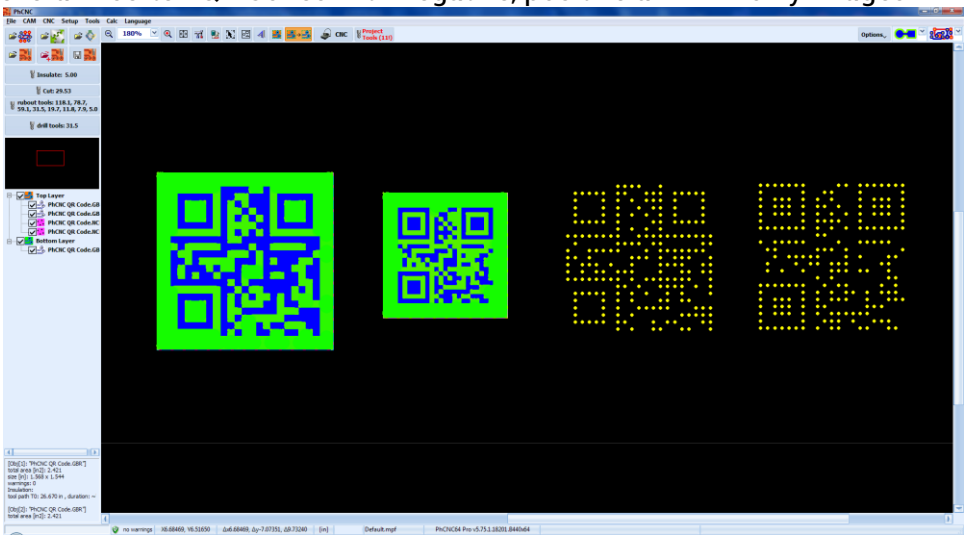
A lot of options are available depending on the methodology of manufacturing and the pitch size:

Positive image QR Code with Gerber & Drill:



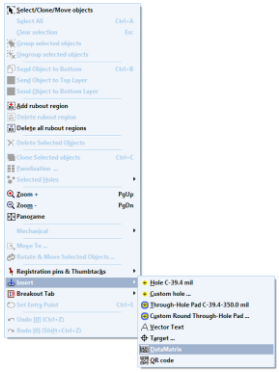
Positive image QR Code with Gerber (engraving / insulation)

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



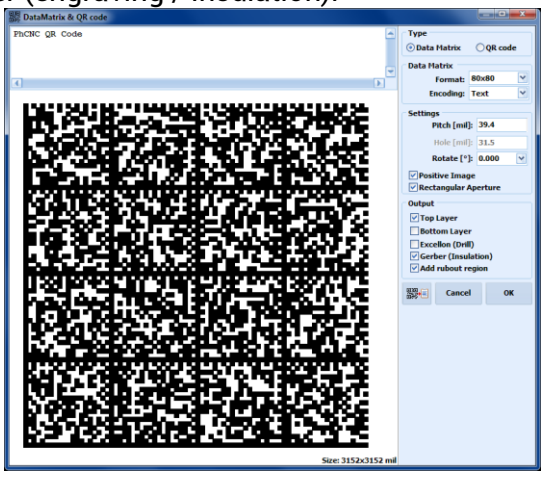
2.33 DataMatrix

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

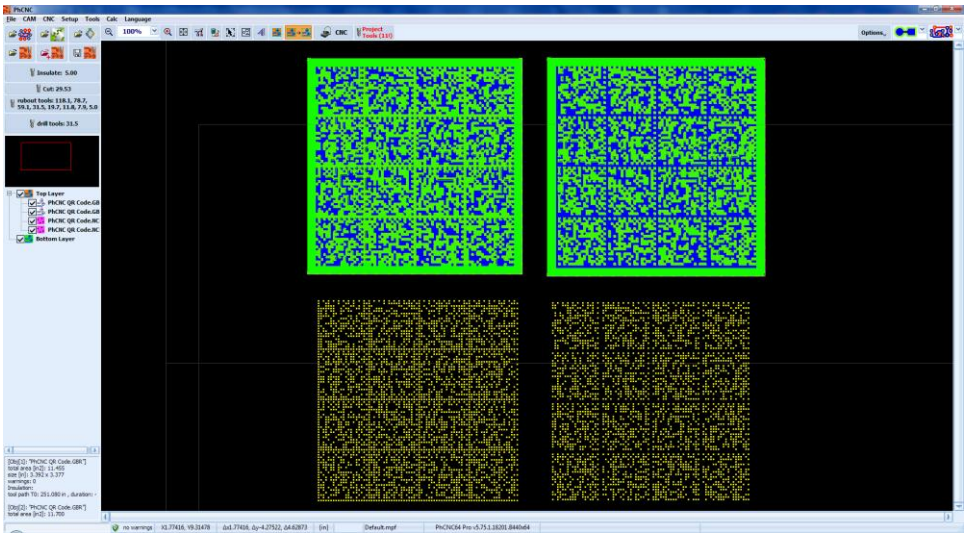


A lot of options are available depending on the methodology of manufacturing, the pitch size and the image size:

Positive DataMatrix with Gerber (engraving / insulation):



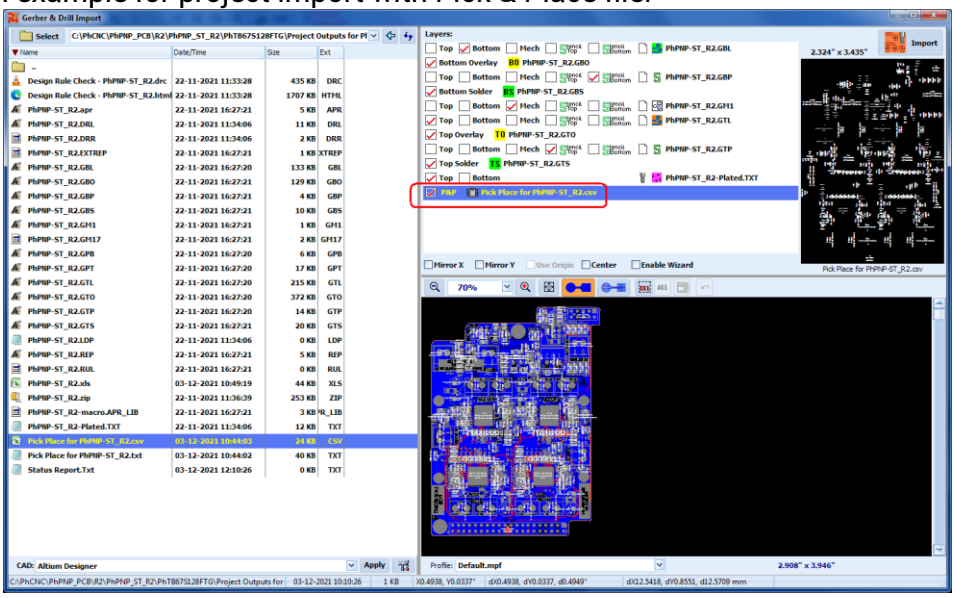
The result of several inserted DataMatrix with negative, positive and drill only images and with format 80x80:



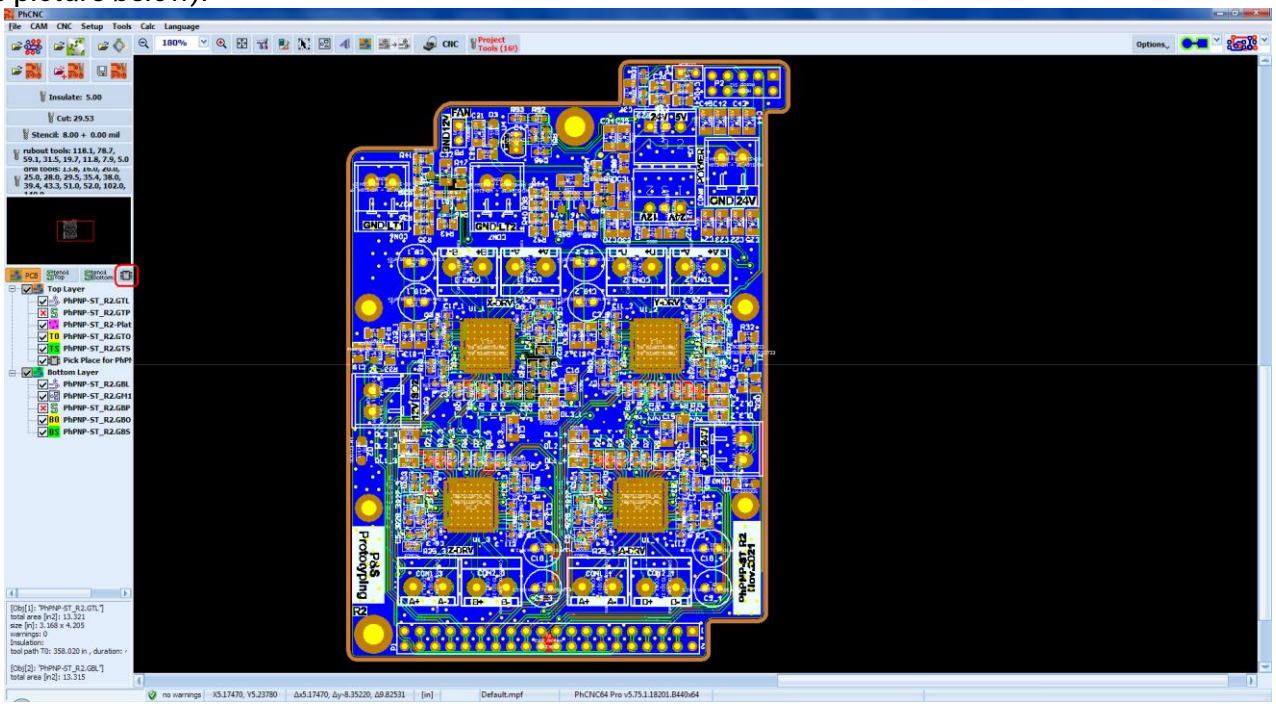
2.34 Pick & Place Helper

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

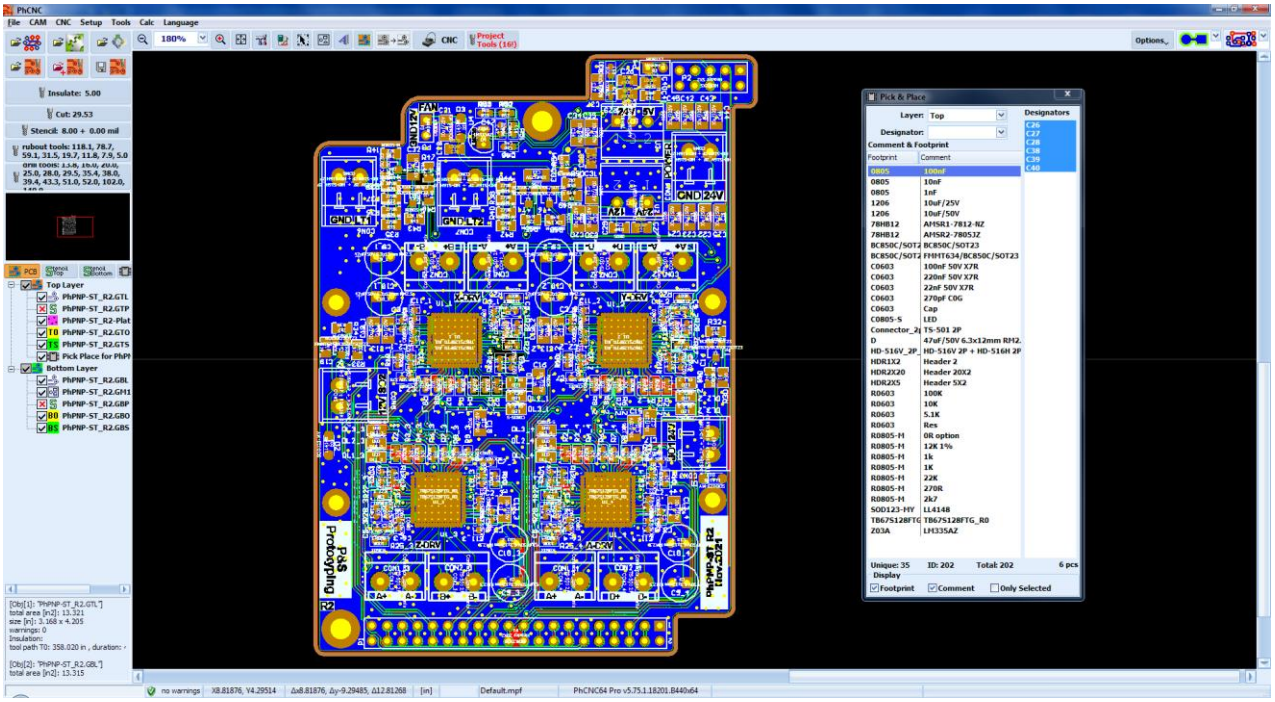
Here follows an example for project import with Pick & Place file:



When the project contains Pick & Place file, the button "Pick & Place" is visible (circled with red line on the picture below):

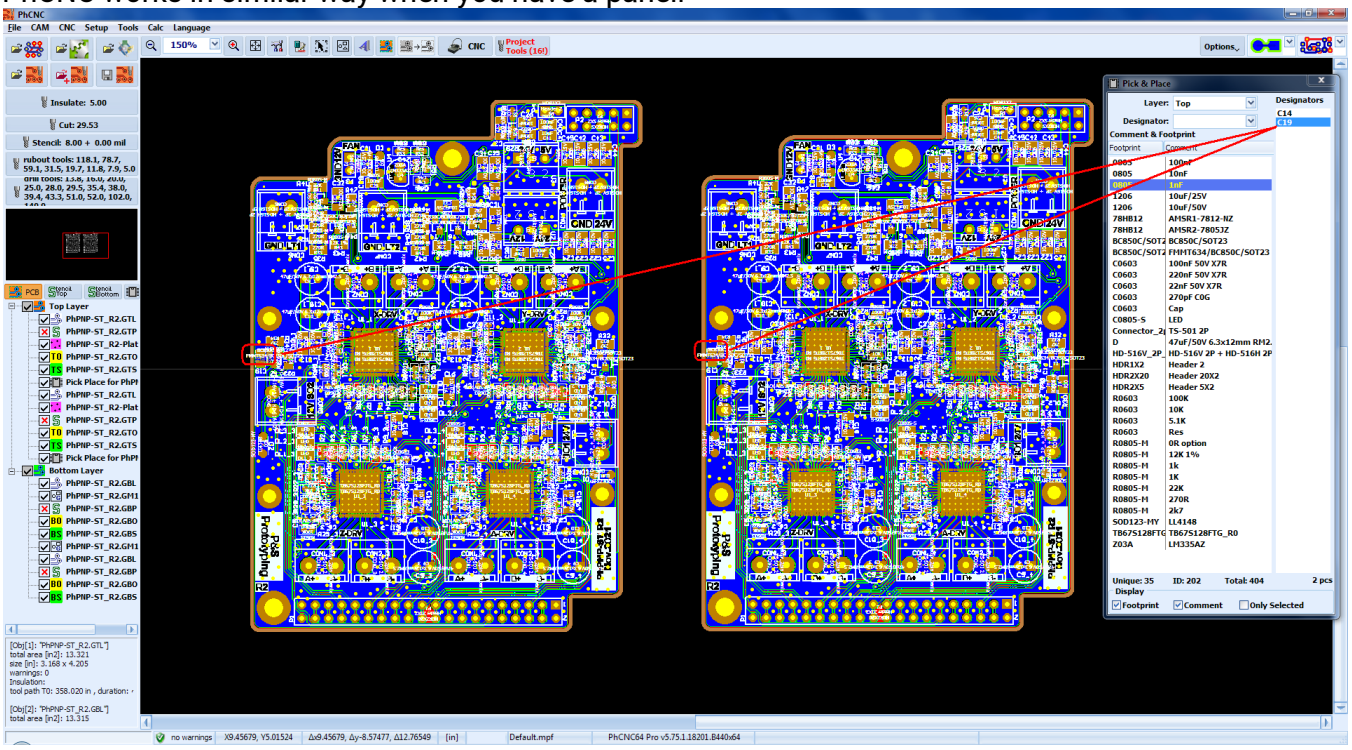


When the "Pick&Place" button is pressed, PhCNC shows the Pick & Place Helper window:



From this screen you can select any component, then PhCNC will show you where it is located on the PCB by making this component flashing. If only one component is selected, the project will be 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 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/1WqBs7E_AXA

3. Setup

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

3.1 Machine Setup

Listed below are the settings:

3.1.1 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 models. The default setting for "Depth Limiter" is checked. Standard models use closed loop feedback to limit the depth and should have this option checked

3.1.2 PhCNC Camera

This setting Enables or disables the PhCNC video module and allows you to select a video format/size such as 640 x 480.

• **Contact Pad Test (Home Z)**

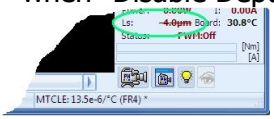
This setting Enables or disables the "Contact Pad Test" (See 4.1 for Z home procedure: button "Home Z").

3.1.3 Disabled depth control

This setting when checked disables CAST™ (Continuous Automatic Surface Tracing) allowing the user to keep the pressure foot off of the work piece and use CNC open loop depth control (Depth is controlled by the g-code only).
"Disabled Depth Control" option, allows the system to operate without pressure foot. This option is created for some special applications where the pressure foot can create problems.
When "Disabled Depth Control" is active the pressure foot is lifted at approximately 0.354 inches (9 mm). A specially designed clip is included in the machine accessories that can be placed and removed quickly. In this mode (without CAST) the tool penetration will be less accurate and will depend of material thickness variations and some additional factors.
Home Z procedure is extremely important for this mode of operation. The procedure is the same as with pressure foot (CAST mode), but require empty spindle collet. An ATC machine will place the tool back in the tool holder or ask the operator to remove it manually. Same apply for machines with manual tool change (MTC).

Notes:

- when "Disable Depth Control" option is active, PhCNC does not generate a warning message for the inappropriate tool installation. In this mode the tool should be installed the same way as in CAST mode, but the automatic evaluation is impossible.
- the switching between CAST mode and "Disable Depth Control" mode can be done multiple times without a need to re-home the machine. It is an important operator obligation to set the pressure foot in appropriate position (lifted when "Disable Depth Control" is active).
- when "Disable Depth Control" is active **Ls** reading is shown on the screen with a line across it:



Important:

This option is designed for special operations only, please consult with us in advance.

3.1.4 PhCNC Vacuum Control Options

Manual Vacuum Control - Designed for the cases that are using the debris collecting vacuum as a source of vacuum for the table. Possible solution, but definitely not recommended by us.
Auto ON - This option starts the vacuum automatically at the start of the machining process. Turning the vacuum off is operator's action. Designed to serve the application mentioned above. IMPORTANT! Any interruptions in the vacuum supply for the table between the operations (insulation, rubout, drilling etc.) may create significant error due to the board position changes.

3.1.5 PhCNC Power Management Options

PhCNC has embedded Power Management Options (PMO) that controls the systems operations. When a request is placed outside (of PhCNC), in the operating system, for Sleep/Hibernate, Windows Restart and User Logoff, PhCNC controls those requests with its (PMO) to keep from being interrupted at the wrong time. All options are selected by default. Windows operating system versions below Windows Vista and Windows 7/8 allow PhCNC full control of these features. Unfortunately Windows Vista and Windows 7/8 do not. Listed below are the options and the differences in compatibility between windows versions:

- **Disable Windows Sleep/Hibernate**:

When machine is working When this option is selected, PhCNC rejects the requests for Sleep/Hibernate during the machine operations (program execution, active Home procedure, active Jog etc.). This option is only effective for windows versions before Vista. Under Windows Vista/Win7/Win8 (most likely also in the future revisions of Windows) PhCNC can not prevent Sleep/Hibernation mode. The user must disable this in windows control panel. In any case PhCNC will put the machine in pause mode before the Sleep/Hibernate mode is activated. After leaving this mode is able to continue from the pause point (if the machine was not turned off).

- **Display Always On**:

This option prevents the operating system from turning the display off during machine active operations. The display control returns to the operating system when the machine is in idle mode or off. This option currently works on all windows revisions.

• **Screensaver Always Off**

This option if selected prevents the operating system from switching to a screen saver (if one is assigned) during the machine operations. When the machine is in idle mode or off the control of the screen goes back to the operating system. This option currently works on all windows revisions.

• **Disable Windows Restart/Shutdown & User Logoff**:

This option if selected rejects requests for shut down, restart, etc. (except Vista/Win7/Win8) during machine active operations. Under Windows Vista/Win7/Win8 a "Forced Shutdown" is allowed. In this case you will see a prompt similar to one shown below.

If you select "Log off now" ("Shut down now", "Restart now"), PhCNC will cancel the program execution and save the current project in "Recent Project" and will shut down. If the machine is not active it will save the current project and shut down.

Important:

In all cases of OS requests to shut down PhCNC it will save the current project in "Recent Project".

3.2 CNC process control settings

Important:

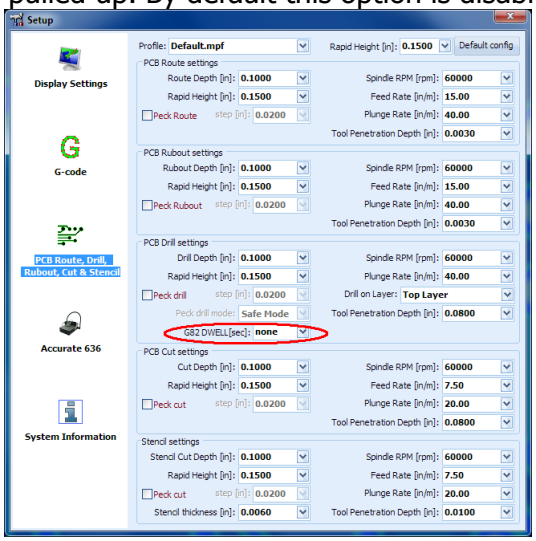
The parameters included in this table are used in creating the G-Code programs. Therefore, changes made after you enter CNC mode will not be effective. In the latest revisions of PhCNC, the access of "Setup" window is disabled in CNC mode.

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.

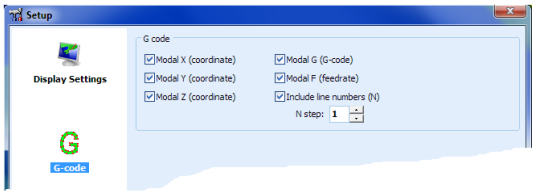


If this option is selected it applies only for the use of drill bit. If a router bit is selected it is disabled automatically.

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 adjustments and selections

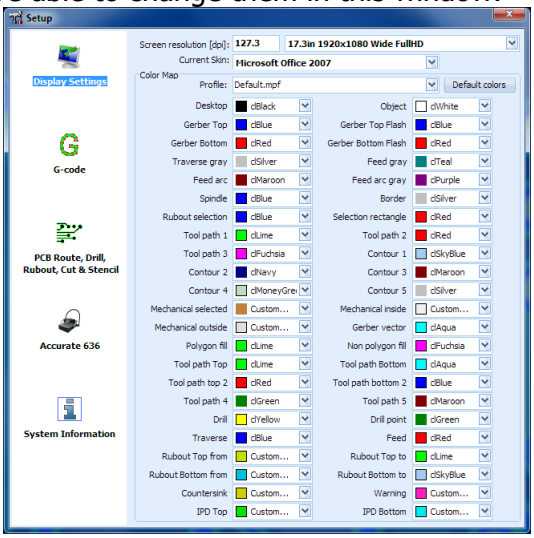
The selections in the table below will affect the size and readability of the G-Code generated by PhCNC, but they do not affect machine control.



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%.

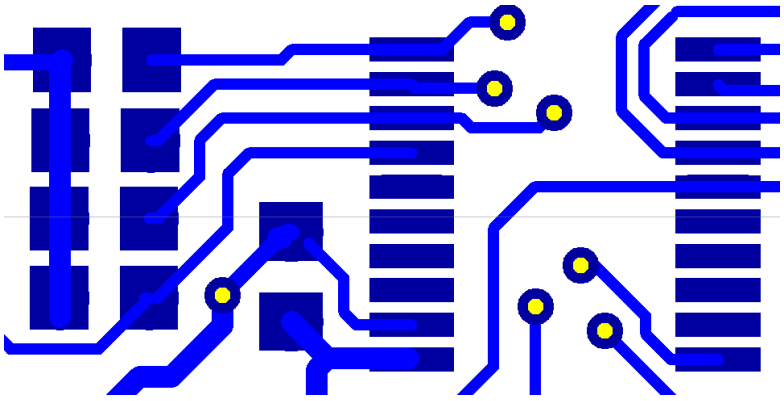
PhCNC has 7 built in skins, you are able to change them in this window.



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 & PhCNC64

PhCNC does extraordinary volume of calculations during the detrermination of tool paths in Insulation, Rubout, Remove spikes (IPD), Forced Insulation, ...
Beginnig 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 paralel processing is in use, **PhCNC** will need larger size of RAM. This problem does not exist in **PhCNC64** - the 64 bit version of **PhCNC**. 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 is fully identical to **PhCNC**. The 64 bit version simply is capable of full utilization of the capabilities of the new x64 OSs.
PhCNC64 and **PhCNC** can be installed on the same computer, but only one can be run at a time.
Ther is an option for manual setting of the maximum number of paralel threads in the Setup Window.

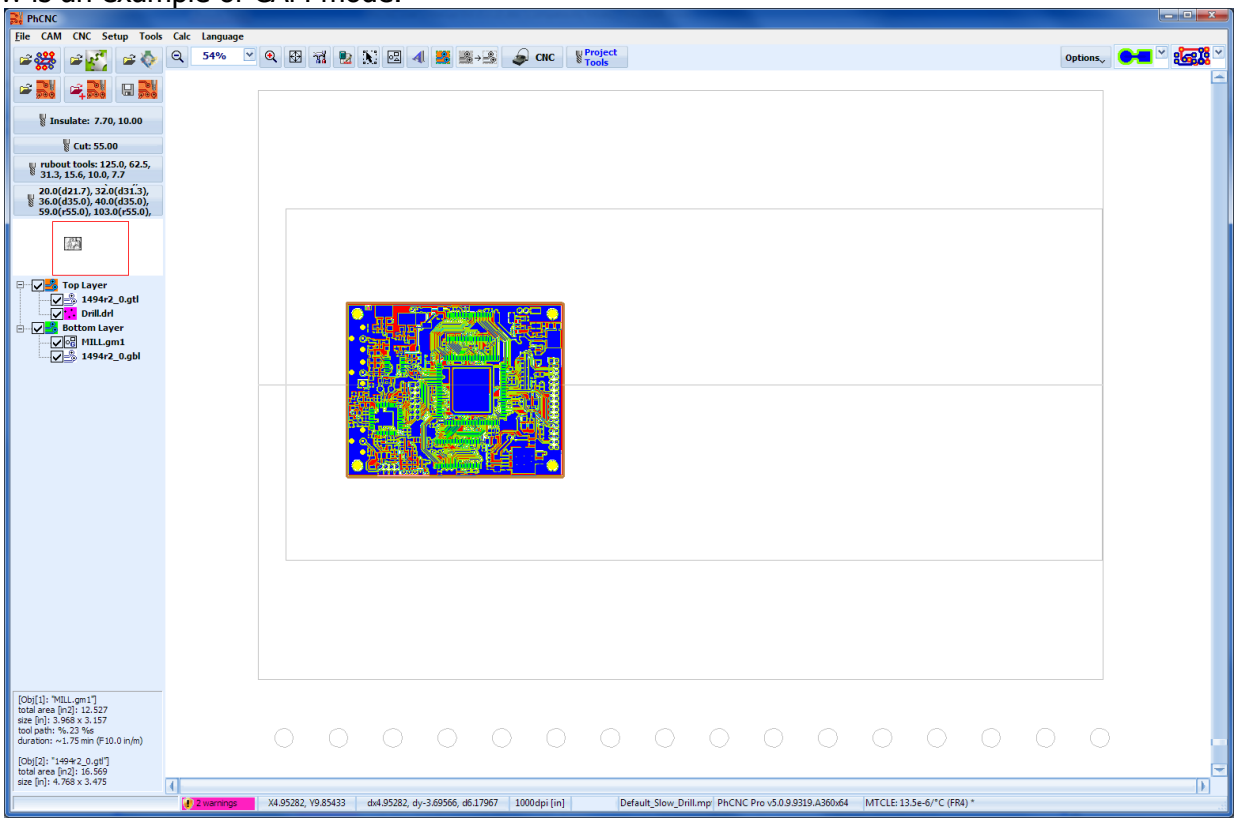
By default the maximum number of parallel threads is equal to the number of logical cores in the CPU(s) of the computer in use.
PhCNC will utilize the maximum number of parallel threads when it is posible or needed.

4. CNC mode

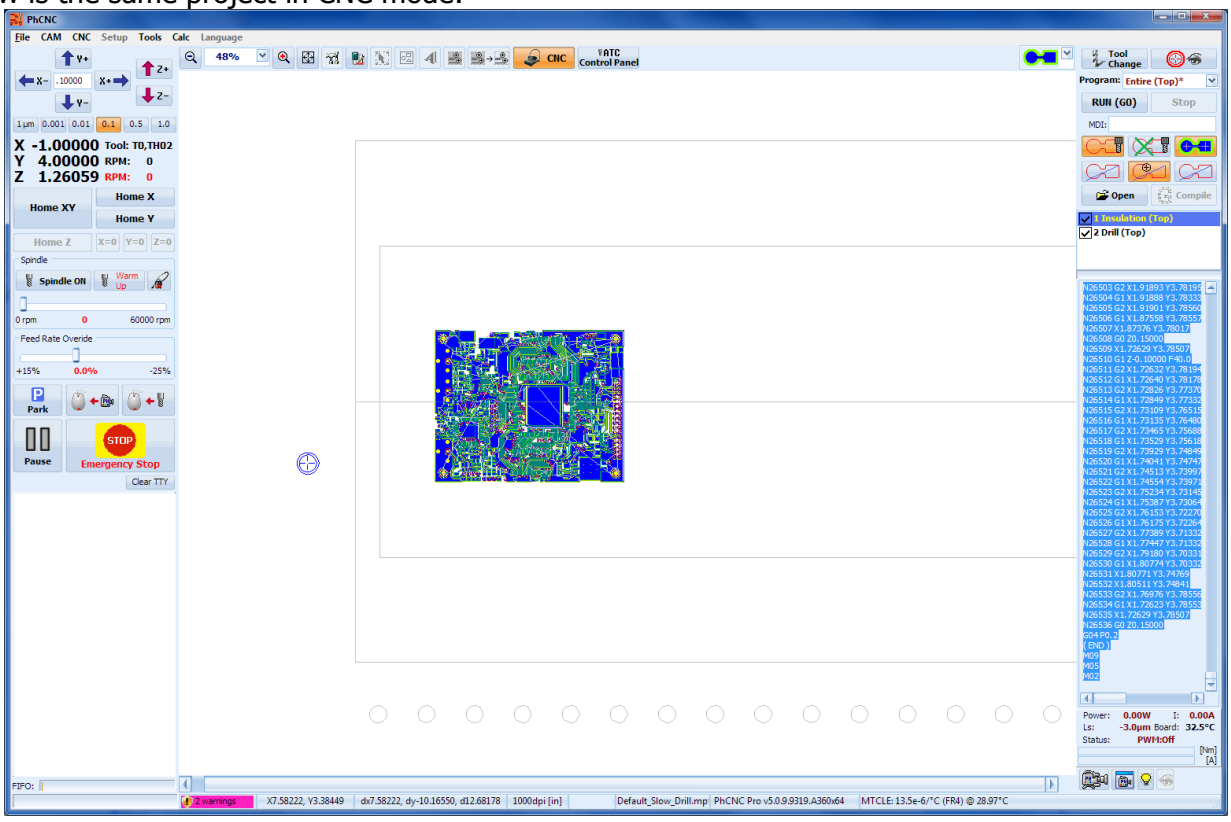
In CNC mode you will be able to produce the boards with your design, as edited in CAM mode. To enter CNC mode, click on the CNC button.

Important:
PhCNC 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:



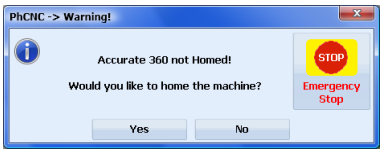
On the left side of your screen are all the manual controls, and on the right side are all the program execution controls.

Every time PhCNC 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

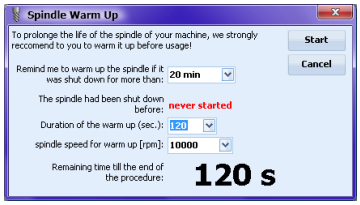
button.

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.

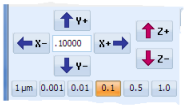
The next window will prompt you to warm up the spindle. PhCNC keeps track of spindle usage and will issue this prompt depending on the settings specified in this window. It is recommended to accept and push the Start button. This will prolong the life of your spindle.



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 (beginning with PhCNC v1.11+) "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 (1um).

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 will record your new "stack" height. In every future tool tip calibration PhCNC 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 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

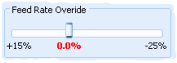
starts the Spindle warm up procedure.

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. In order to use this option, you have to have an Automatic vacuum starter (shown in the picture), or use one of our vacuums (all have this unit integrated).

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

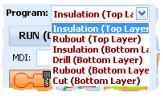
This button moves the machine head to the corner opposite to the machine zero. It is designed to provide access to the work piece and the tool magazine in ATC models.

4.2 Program execution in CNC mode

When you enter CNC mode, PhCNC 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 has generated these 6 programs:

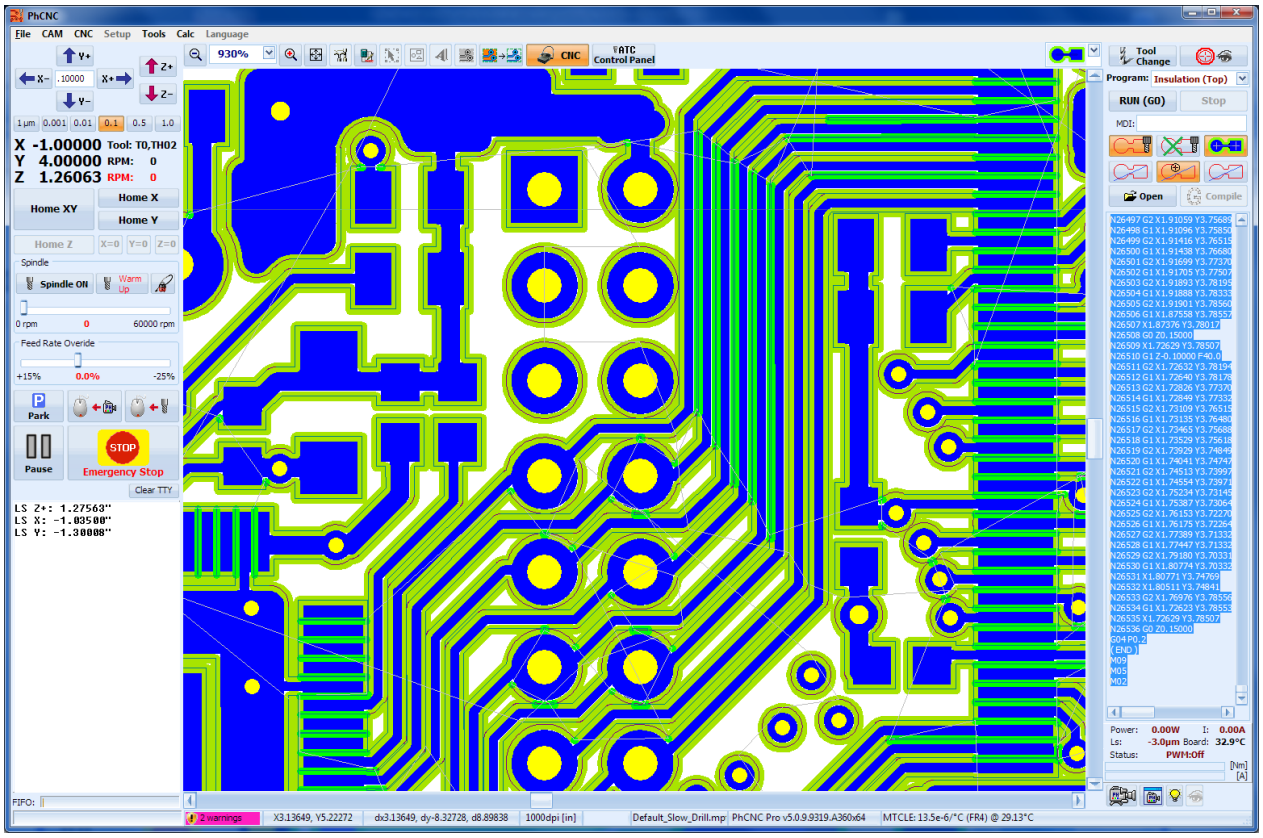


The operator sets the order of execution as desired.

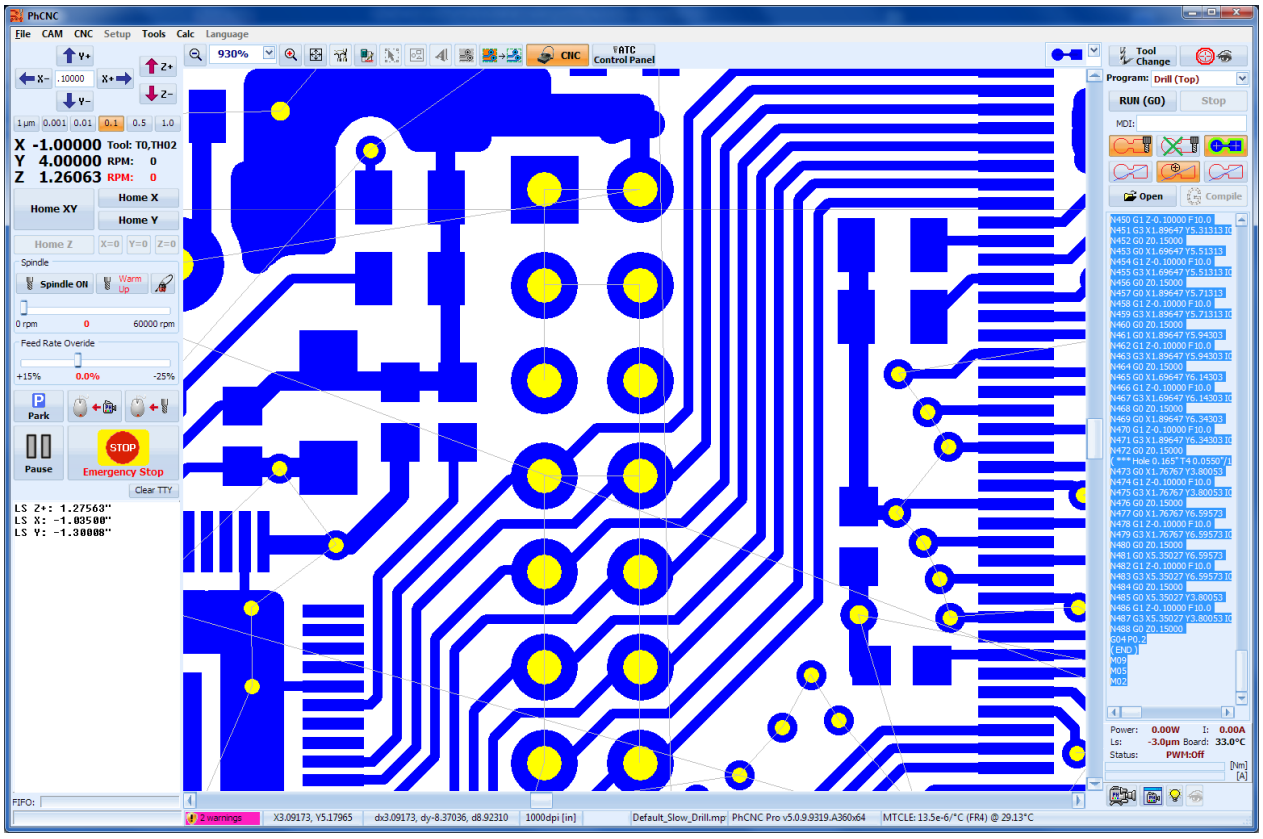
Note that the cutout must be the last!

Shown below are the tool paths of each program:

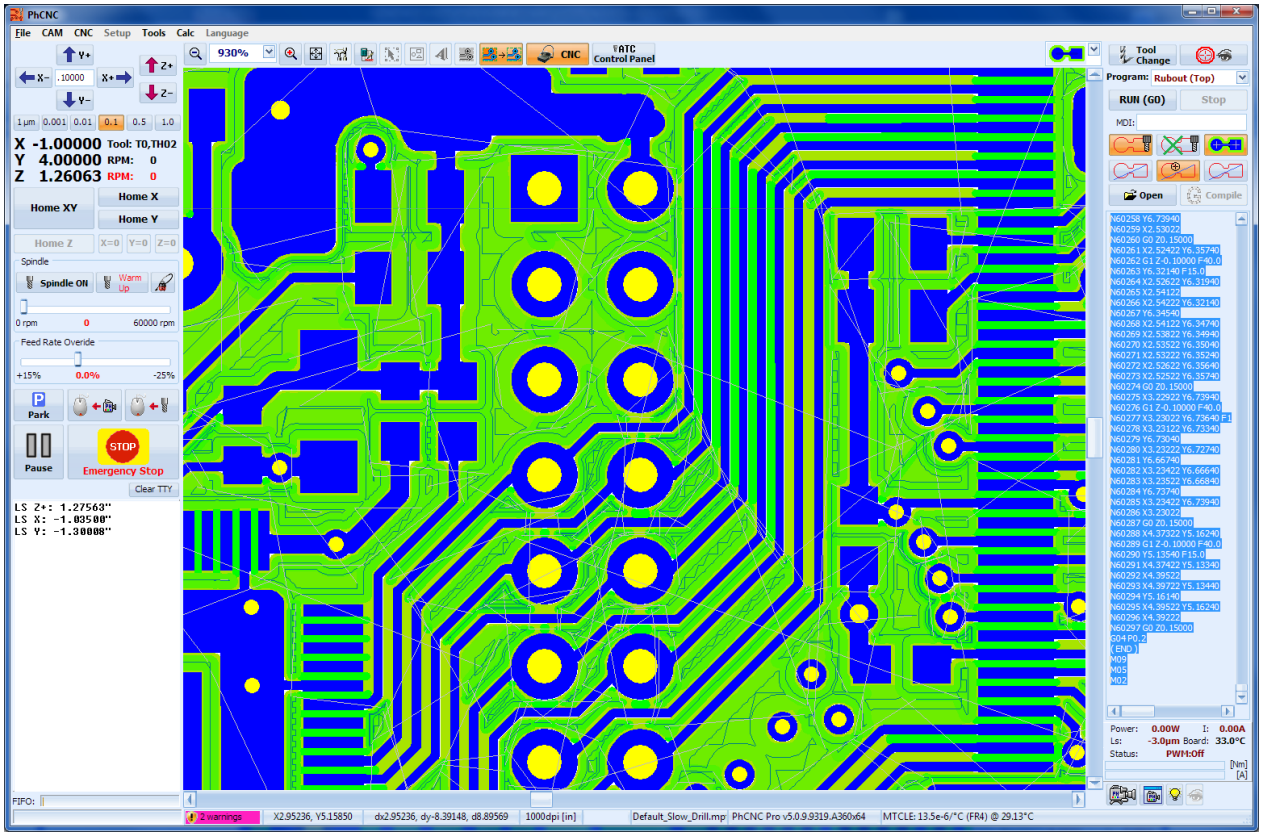
"Insulation (Top Layer)":



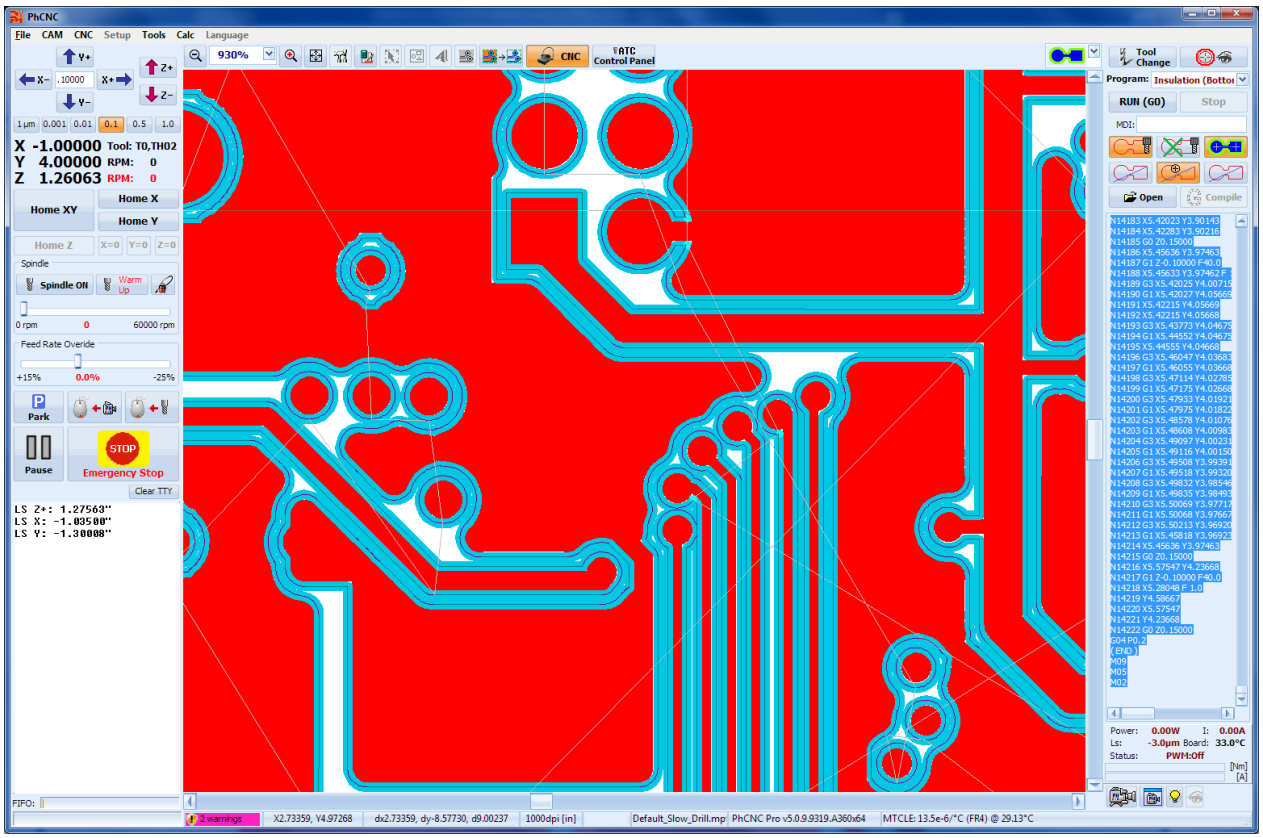
"Drill (Top Layer)":



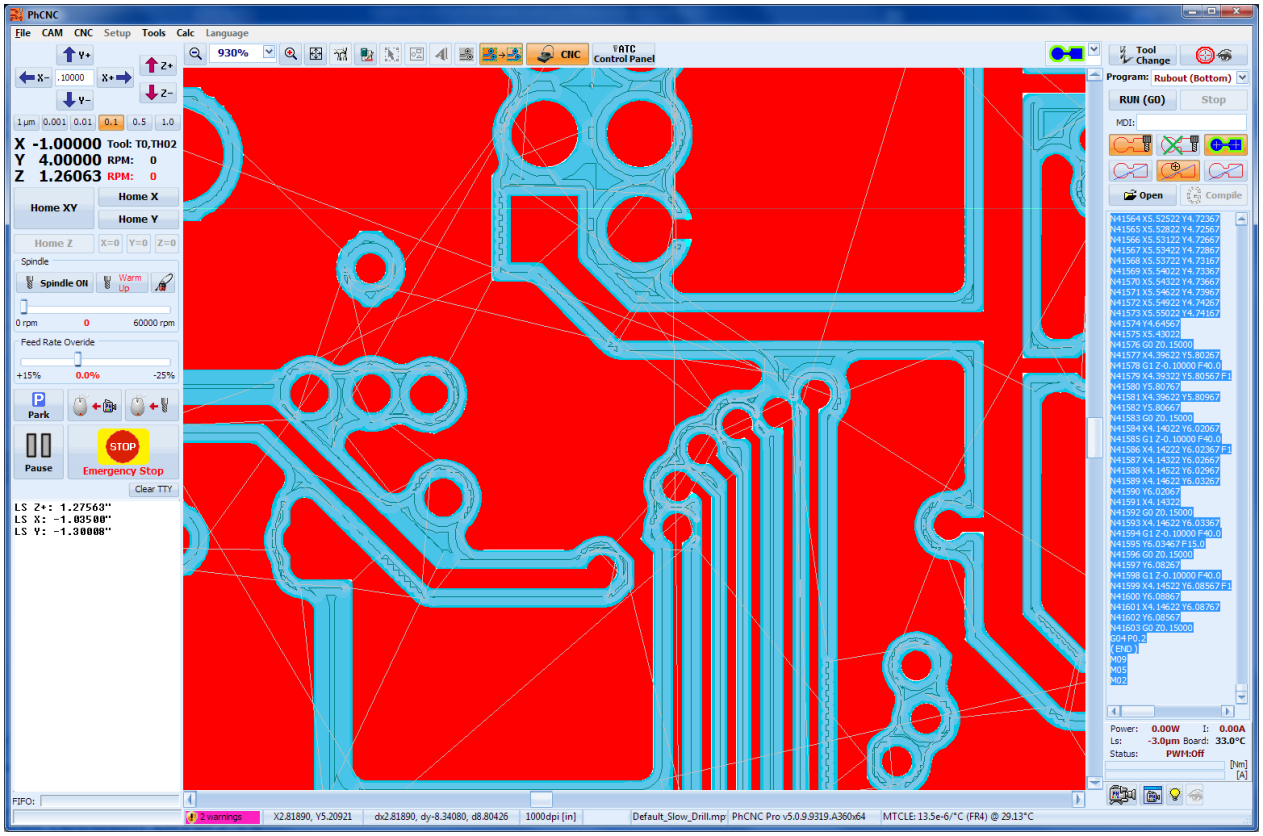
"Rubout (Top Layer)":



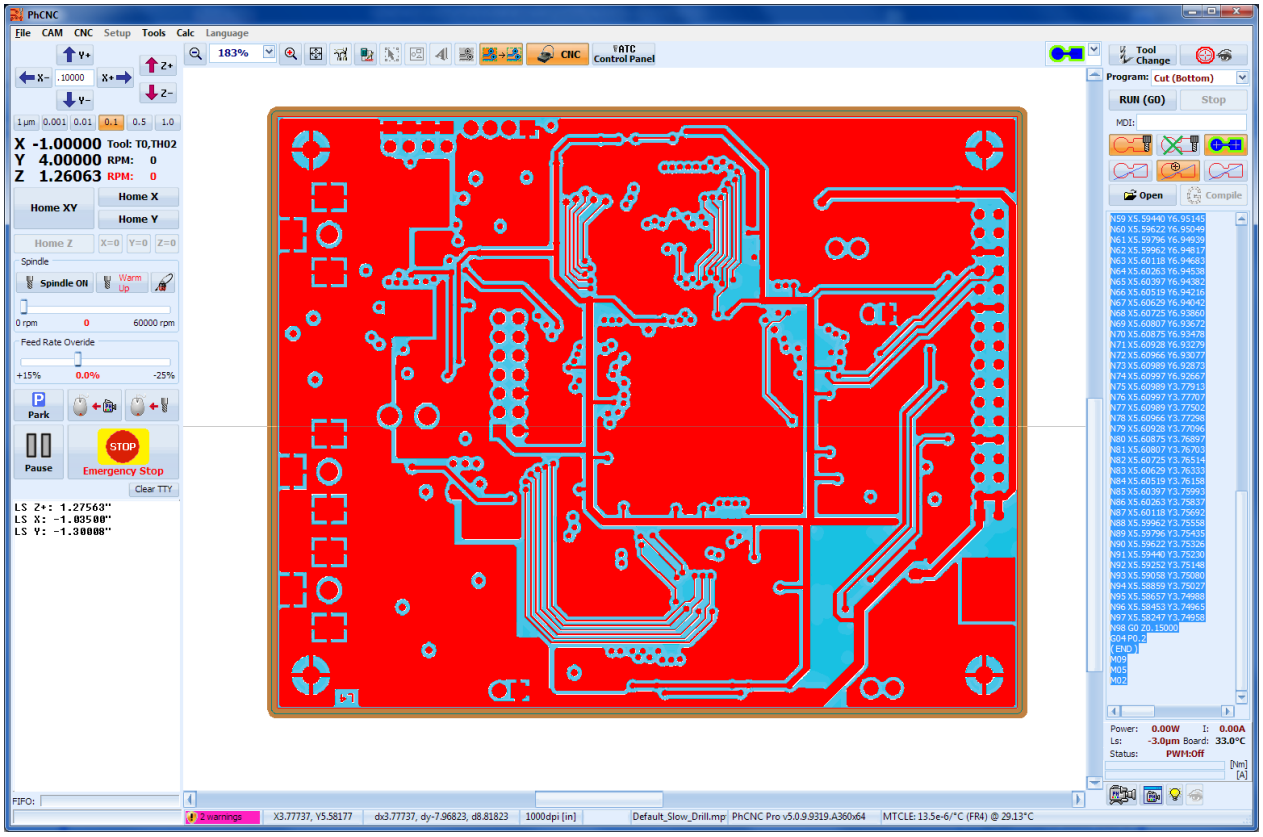
"Insulation (Bottom Layer)" - insulation:



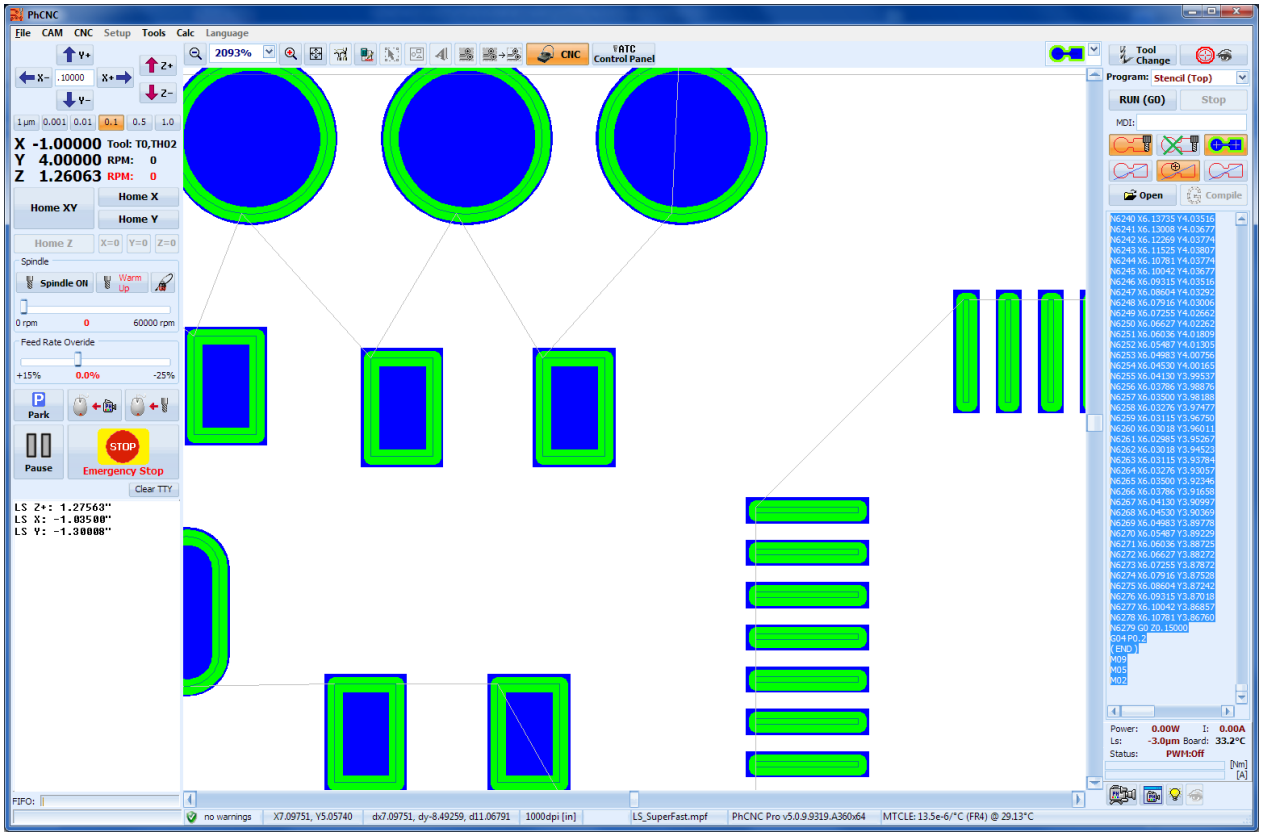
"Rubout (Bottom Layer)":



"Cut (Bottom Layer)":



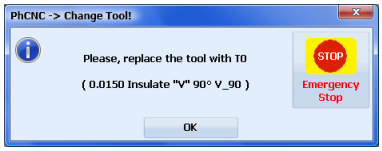
If during the import in CAM mode you have layers declared as a stencil for solder paste (usually only on the top layer) these additional machining programs will appear - "Stencil (Top Layer)" and "Stencil (Bottom Layer)" (if any).



4.2.1 Control buttons for G-Code programs execution

4.2.1.1 Tool Changes

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



Button Tool Change 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 RUN (G0)

Starts the execution of the selected G-Code program.

Important:

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

4.2.1.3 Stop

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!

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

Examples of MDI usage:

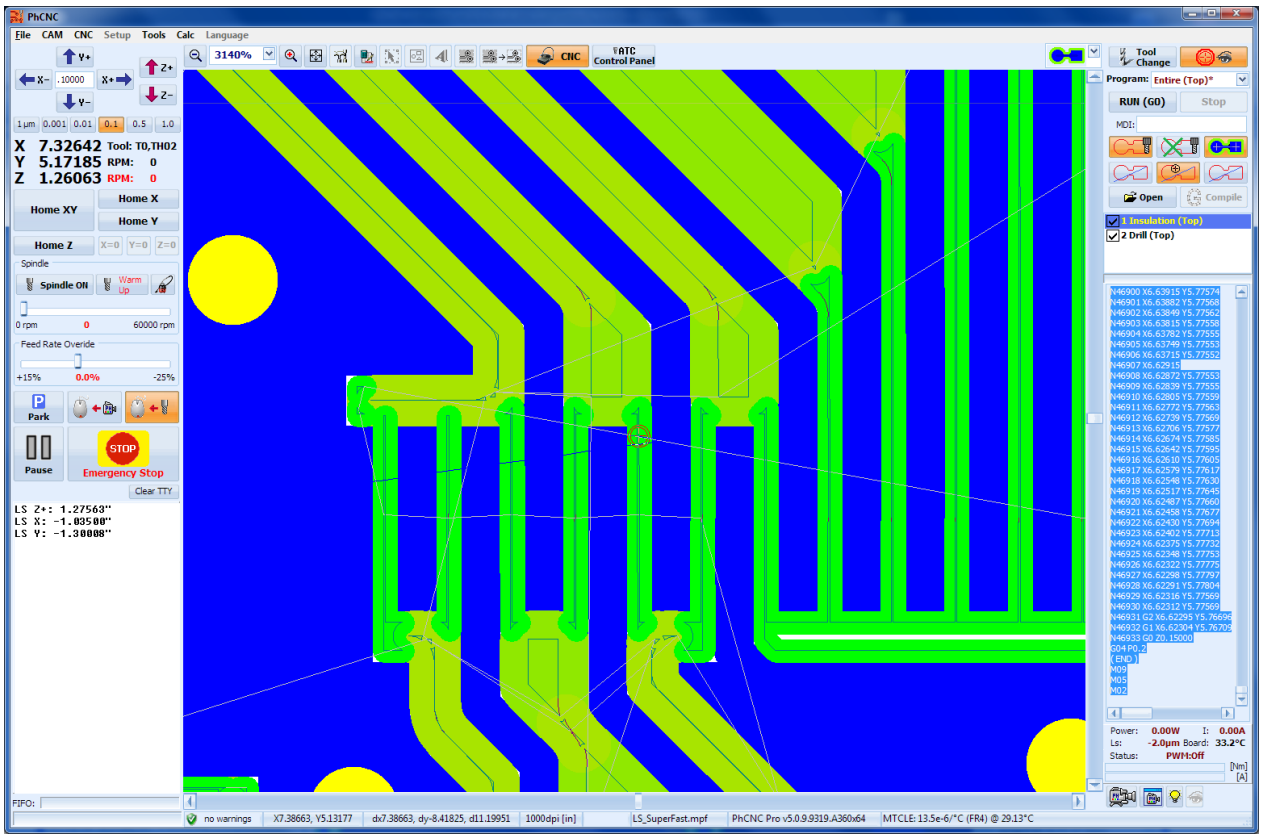
```
G0 X0 Y4.950
G1 X7.5 Y4.950 F15
```

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

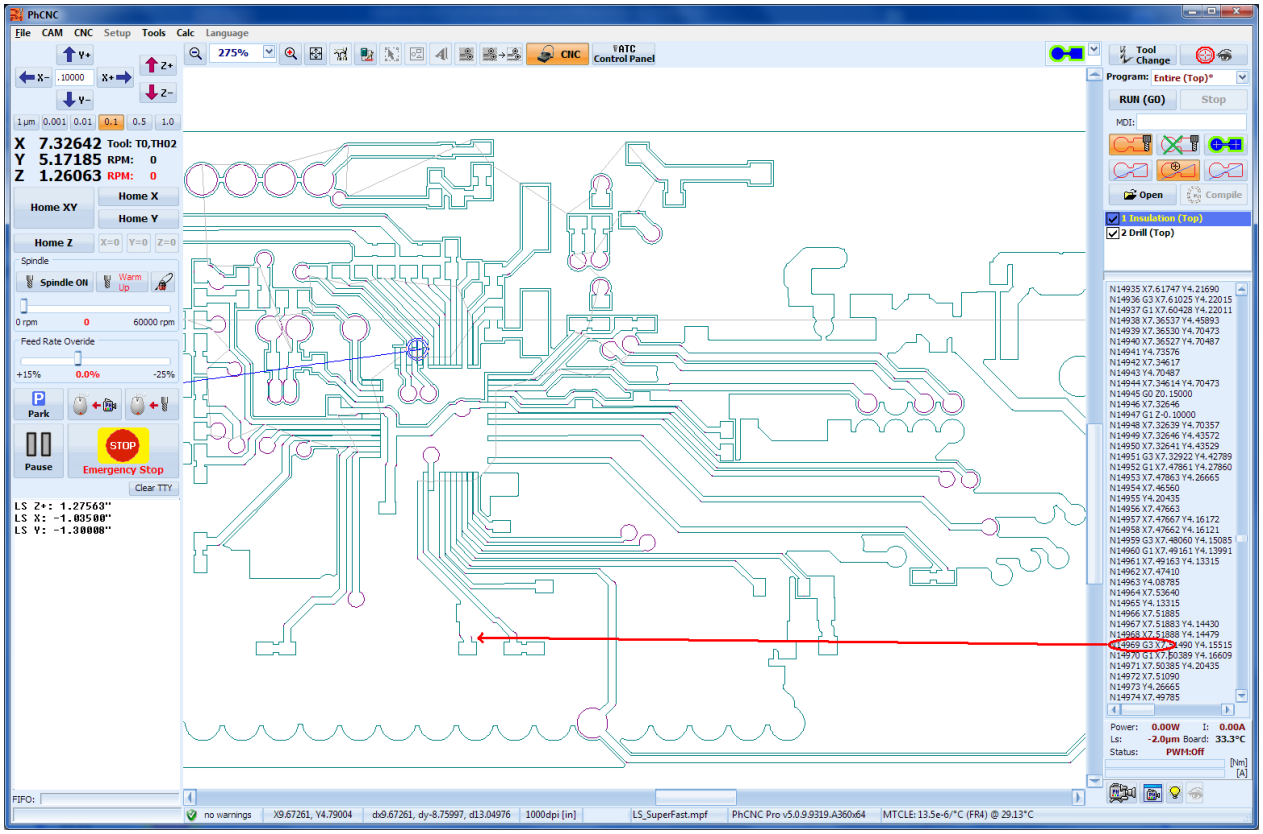
This button shows/hides accumulated RAW information generated during the spindle movements.
Red lines indicate negative Z coordinates (tool movements penetrated into the work piece), blue ones - positive or zero (usually movements above the surface of work piece).

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.

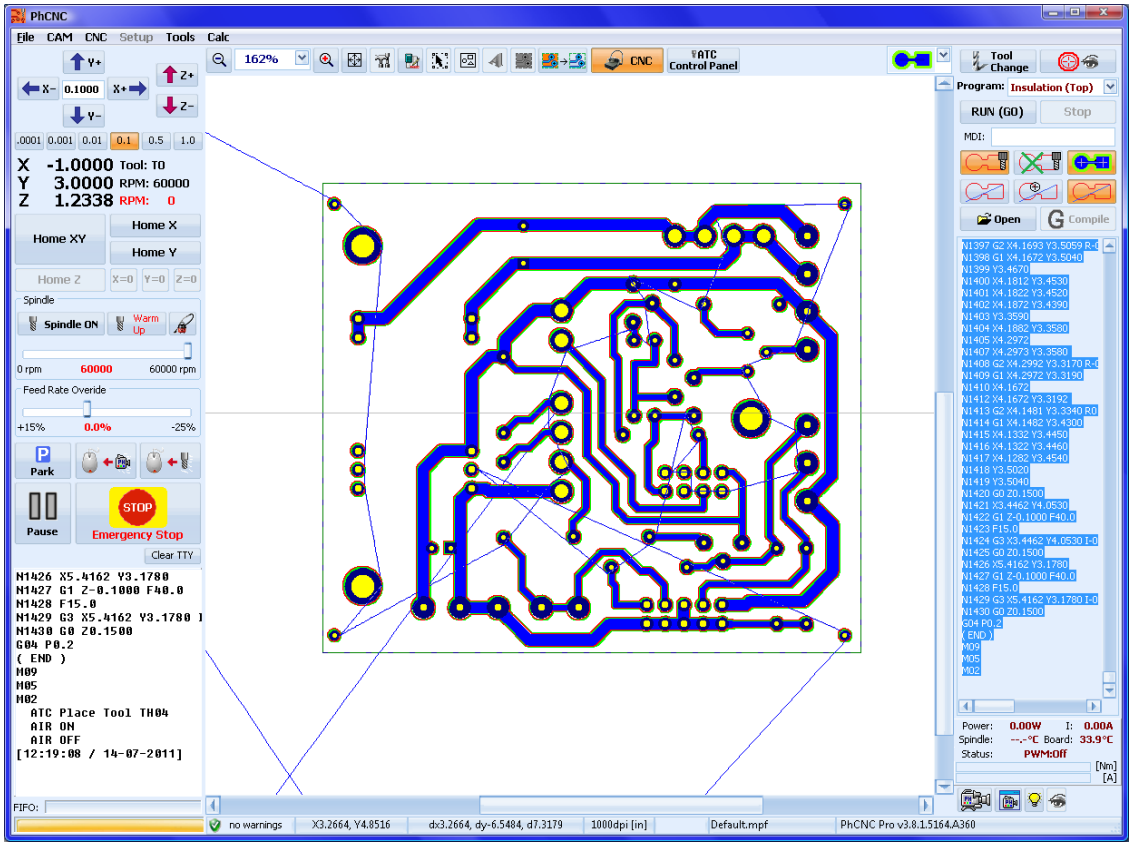
4.2.2.5

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 (G0) , PhCNC automatically recompiles the G-Code.

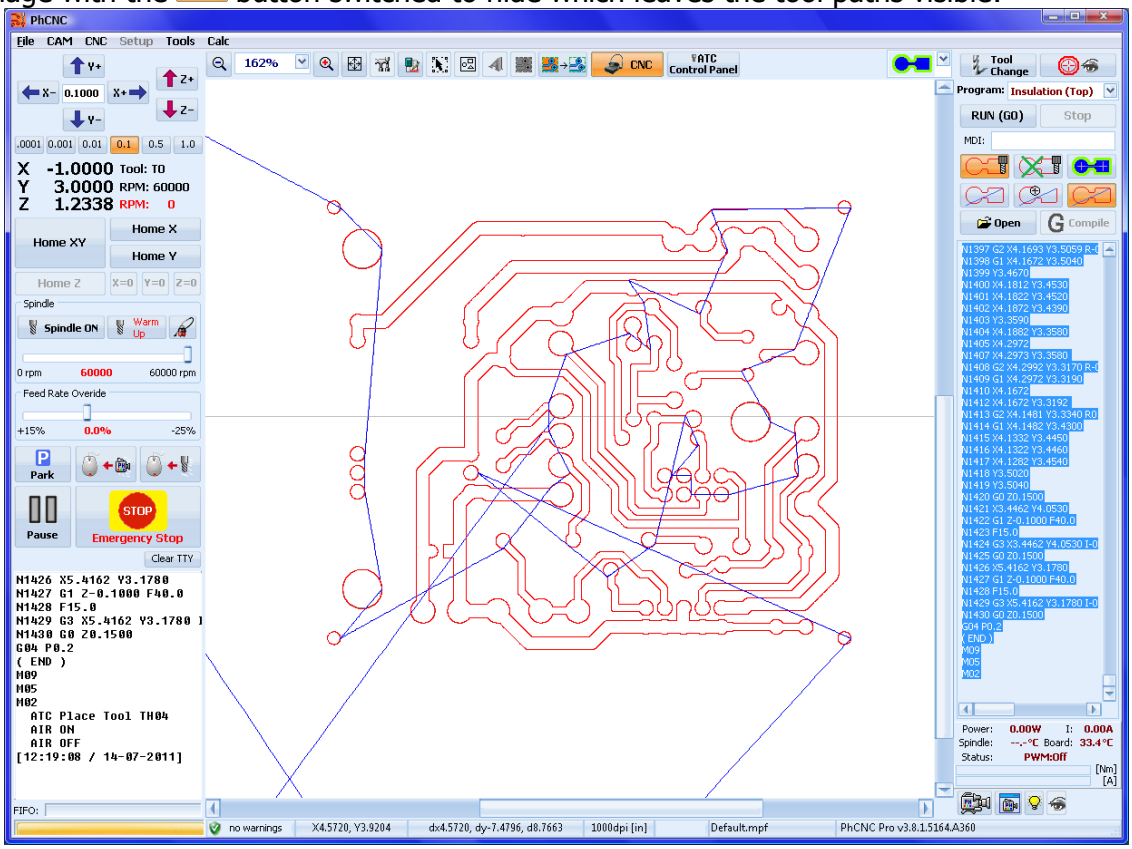
4.2.2.6

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.

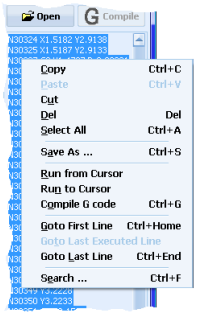
Shown below is a completed Top Layer insulation program the button is set to show allowing the Gerber information to be visible:



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



4.2.3 G-Code editor menu

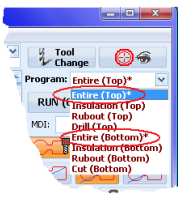


This menu is available by right clicking inside the text window to the right. You can perform the following operations from this menu:

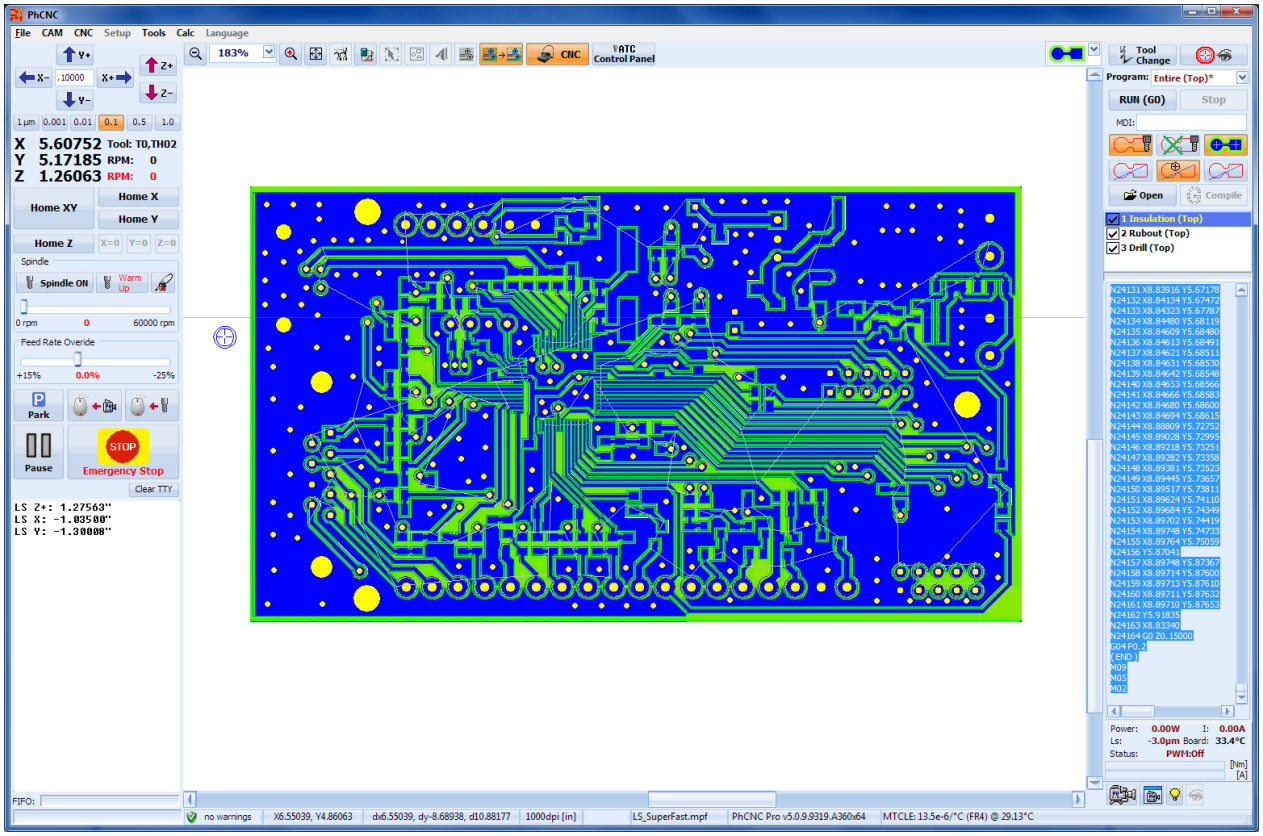
- **Copy:** this is the standard copy function used in most text editors and windows programs.
- **Paste:** this is the standard paste function used in most text editors and windows programs.
- **Cut:** this is the standard cut function used in most text editors and windows programs.
- **Del:** this 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 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.

4.2.4 Programs and operations execution using a ATC system

We will call each G-Code program generated by PhCNC in CNC mode "program". There are different programs in a single project that are separated by their type of operation. Drilling, Insulating, Copper rubout, Cutting. The selection and the execution of the programs using an ATC machine is the same as in our previous models. Select the program and run it. The main differences now are two additional programs generated by PhCNC. They are "Entire (Top)*", and "Entire (Bottom)*". These programs combine together all the programs related with the board side that is to be machined, and thus allow completing the entire machining process without any user action required.



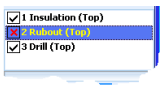
The "Entire (Top)*", and "Entire (Bottom)*" are generated only when there is more than 1 program for the particular board side (top or bottom). When you select any "Entire ( )*" program it shows a list of the programs contained in it.



Shown above "Entire (Top)*" program is selected and this one includes:

- Insulation (Top)
- Rubout (Top)
- Drill (Top)

These are shown in the program list. The order of execution of these programs may be changed simply by using a "drag & drop". Each program comes enabled but you can disable any if needed. Simply left click on the box to the left of the program you want to disable, and the check mark will change to a red X. In the sample below we have disabled the "Rubout (Top)" program.



Once each of the programs included in the pack is completed it gets a green check-mark to the far right. Shown below the "Insulation (Top)" program is complete and has the green check mark next to it circled in red, also circled in red is the familiar windows progress indicator that shows the global execution progress for the entire pack.

All the control buttons discussed in the previous chapters work the same with "Entire ( )*" programs as with the individual programs such as the button.

"Run From Cursor" in "Entire ( )*" programs

The "Run From Cursor" option in the popup menu (described in paragraph **4.2.3 G-Code editor menu**) has completely the same function as with single machining operations, the only difference is that it will continue with the rest of the programs in the pack (if any).

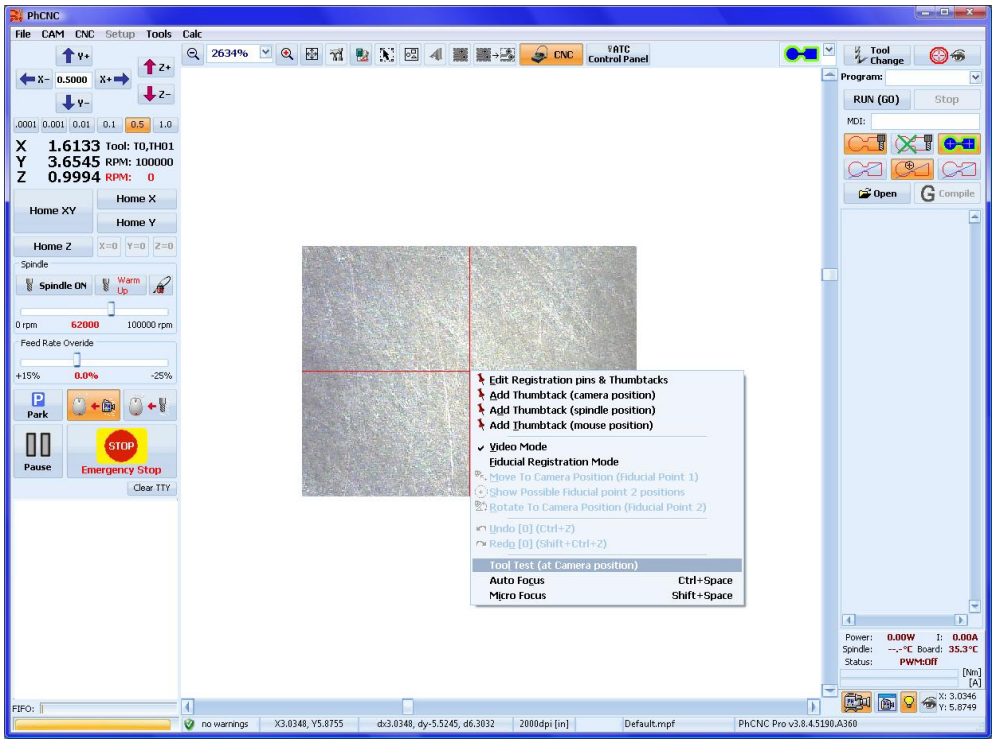
"Run To Cursor" in "Entire ( )*" programs

The "Run To Cursor" option in the popup menu (described in paragraph **4.2.3 G-Code editor menu**) has completely the same meaning as with single machining operations, the only difference is that it will execute the lines from the beginning of the particular machining program to the cursor position.

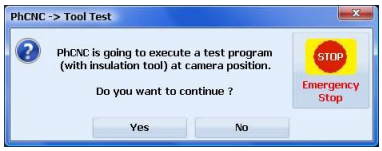
4.3 Tool Test (Pro version only)

PhCNC Pro (v 1.11.3+) has a new function that allows you to make a fast and reliable test of your insulation tool settings. This is very important when you use the most common V type tools. This test should be done before running the Insulation (Top), or Insulation (Bottom) programs. It requires a video camera/microscope option installed and calibrated, and will provide you with real measurements and visual information of the tool performance. The first step is to home your Z axis to your existing work piece (if this is not already done). See 4.1 for Z home procedure: button "Home Z". This test will provide you with the best settings of the micrometer head, controlling the depth limiter. If your project uses expensive RF materials you may test your tool with FR4 material with the same thickness and (most important) same copper thickness. The test cut pattern has a total size less than 0.1 X 0.1 inches. It resembles a cross with the bars 10 mils wide and 40 mils long. The center of this test cut pattern will be the current camera center when you confirm the process execution. The test takes a few seconds and after it is done the camera goes back to show you the results. In order to execute this test you have to:

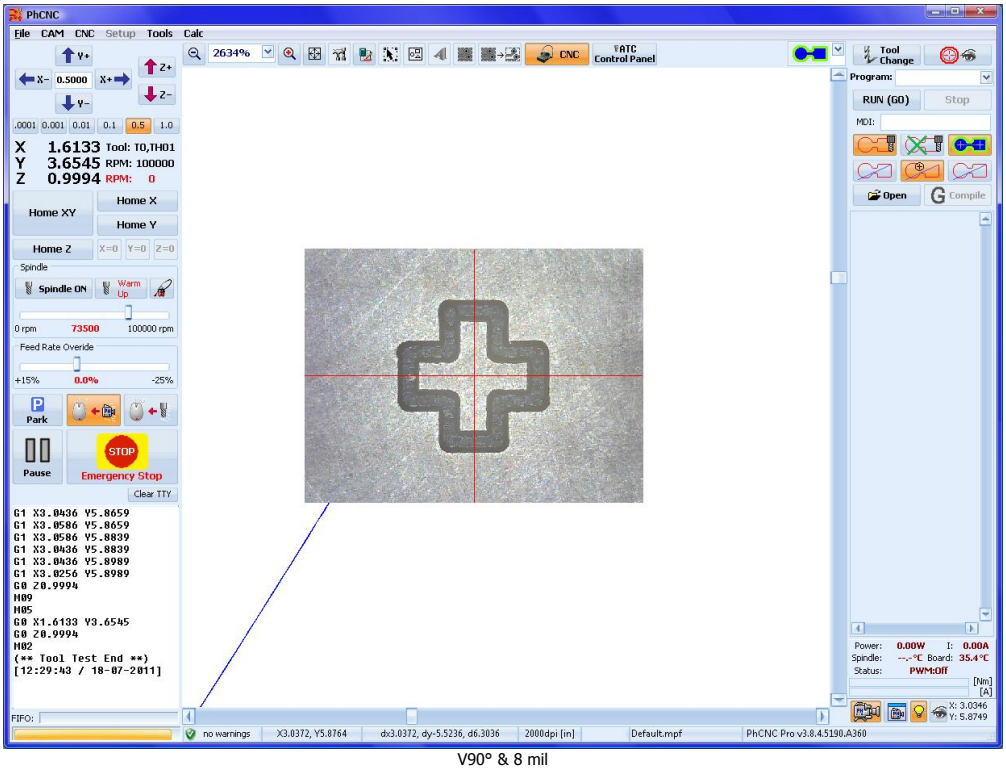
- Activate the camera/microscope and position it at the desired place for the test. Your test pattern will be entirely in the position (window) that you currently are able to see.
- Make a right click with the mouse inside the window (clicks outside the window will not activate this option for safety reasons).
- Select "Tool test (at Camera position)"



The actual execution requires conformation.



The test program includes a tool change procedure that can be used for all needed adjustments. After the execution, the camera will go back to the center of the test pattern and allow you to inspect and measure it.



V90° & 8 mil

If your settings are right the bars of the cross should be 10 mils wide. If you have a different actual measurement and need to achieve better results check your tool settings and repeat the test using a new spot.

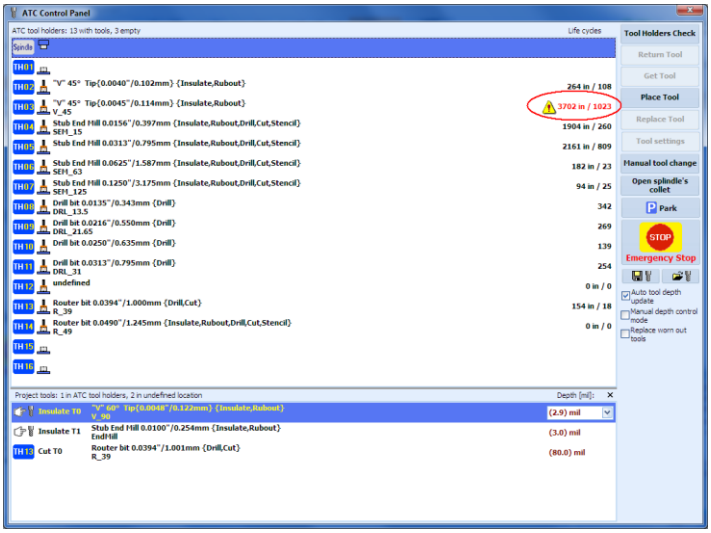
Notes:

- The tool test can give you time saving information if the tool condition is poor when you are using a used tool.
- The tool test procedure can be used in the camera calibration process described in Appendix A.

4.4 ATC Control Panel

When an ATC machine is in use and the PhCNC is in CNC mode there is a new button "ATC Control Panel". It will open a window that is designed to edit the machine magazines tools content.

ATC Control Panel:



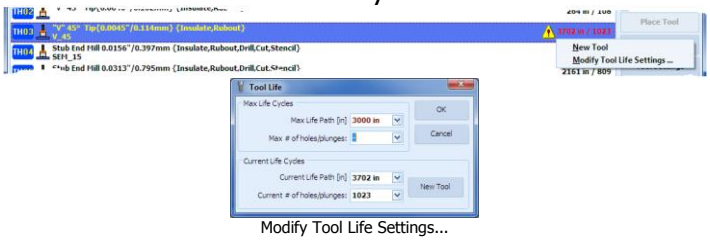
This window is split in two parts:

- The upper part contains information about the tools that are currently in the magazine and the spindle collet. All (12, 16 or 20 tool holders, depends on the machine model) are listed with the tools descriptions or as an empty holder symbol. The spindle collet is treated as an additional tool holder.
- The lower part contains the "Project tools" information discussed above.

In the list of tool holders we have an information for the current usage and if the usage exceeds the usage limits it generates a warning sign ⚠, as it is shown above (3702in > 3000in).

Using the right click over the tool usage values will call an additional popup menu (shown below) that allows you:

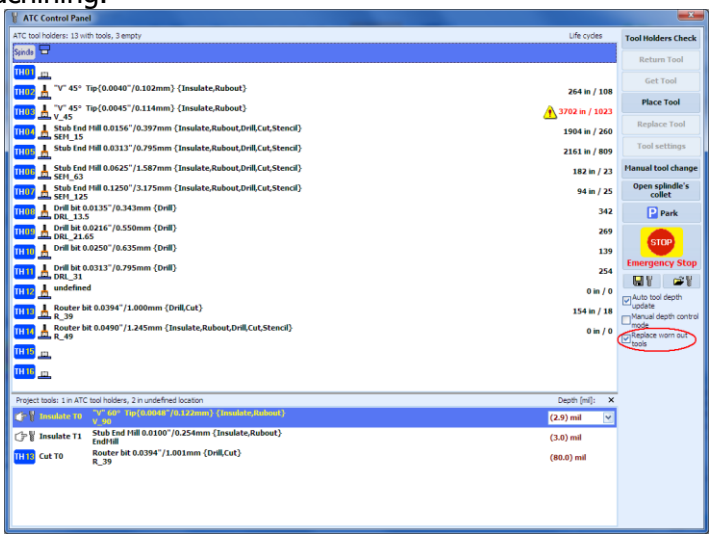
- to clear the current statistic of this tool, "New Tool"
- to edit settings, "Modify Tool Life Settings..." and limits "Max Life Cycles" also to edit the current statistics of any particular tool "Current Life Cycles"



Modify Tool Life Settings...

The usage of worn-out (dull) tools may create significant burrs and as a result problems during the "insulation" and "copper rubout" procedures. It is recommended to use the tools usage records in order to replace promptly the tool that need replacement.
The "Machining Window" option allows the operator to repeat the machining of a problematic areas due to an overused tool.

The option "Replace worn out tools", allows PhCNC software to request wornout tool(s) change during the process of machining.



Buttons in the "ATC Control Panel" window:

4.4.1 "Tool Holders Check"

This button activates a scan procedure (using infrared sensors) that checks all tool holders in the machine magazine and spindle collet. It will determine the status of each holder (occupied/empty) and will assign the tool description if it exists in the records. Holders that are empty on the record and appear occupied after the scan will have "unknown" tool description. This procedure will also check for eventual plastic rings left in the empty holders (optional, but enabled by default), a rare but possible situation. The scan will also detect a missing tool holder itself and report it to the user. In every new

start of PhCNC, when you click for the first time on "ATC Control Panel" it will offer the check described above.

Important:
If the holders scan above has never been activated it will be done automatically before first automatic tool change. **If the air pressure supplied to the machine** is below the required level a warning message will appear and all tool change operations will be terminated.

4.4.2 "Return Tool"

This button is enabled only when the spindle collet is selected as a tool holder. It will return the tool to the holder that is on the record to supply the tool to the spindle collet. If this holder for some reason is occupied the operation will be terminated and an error message will be displayed.

4.4.3 "Get Tool"

This button is enabled only when an occupied holder (including spindle collet) is selected.
If a tool holder is selected and the spindle collet is not empty PhCNC will ask you to remove the tool that exists in the spindle collet. If the spindle collet by itself is selected the procedure will be the same as "Manual Tool Change".

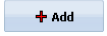
4.4.4 "Place Tool"

This button is enabled only when an empty tool holder/spindle collet is selected. When this button is activated it will place the tool that is in the spindle collet in to the empty holder. If the spindle collet is also empty the PhCNC will go to manual tool change position and ask for the tool to be inserted in to the spindle collet. In the next step it will check for the actual existence of the tool in the collet and then will place it in the empty holder selected. In the case of physical controversy between the content of the collet or holder (checked by the infrared sensors) an error message will appear.

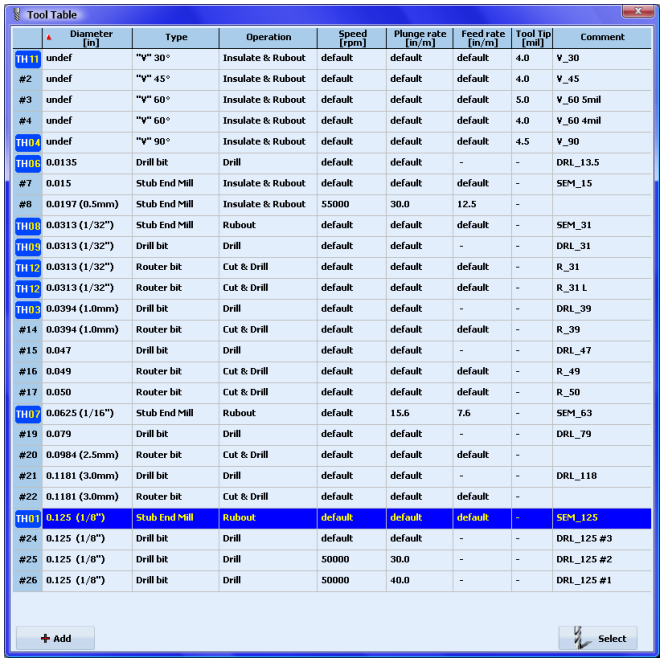
4.4.5 "Replace Tool"

This button is enabled only when occupied holder/collet is selected. It will create a procedure to change the tool that exists in a holder or the tool that is located in the spindle collet. If you are replacing the tool with a new one with the same parameters, this process will be completed after the change. If you replacing the tool with a different one you have to change the tool description. In this case use the process described below.

4.4.6 "Tool Settings"

This button is enabled only when an occupied tool holder/spindle collet is selected. A double click on the selected item will activate the same function as this button. This action will recall the tool table and you can select a tool from it. If your tool does not exist in the table you can add it by using this button 

. Be sure to enter the entire description of the tool it will make things much more convenient in the future.



4.4.7 "Manual Tool Change"

This button will bring the spindle to the manual tool change position. Once you are at this position you can use the target button on the calibration pad to open/close the spindle collet. Using the "Manual tool change tool" you can do this easily. The "Manual tool change tool" is included with the machine.

4.4.8 "Open Spindle's collet"

This button is for emergency situations only. It will open the spindle collet at any time including when the machine is in the "EmStop" condition. It should not be used for other purposes. This button is disabled when the spindle is running.

4.4.9 "Park"

This button is the same as the "Park" button in the main CNC mode screen of PhCNC. It 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.

4.4.10 "EmStop"

This button is the same as the "EmStop" in the main CNC screen. It will put the machine in emergency stop conditions and will close the "ATC Control Panel" window.

Important:
If you enter the "ATC Control Panel" during the "EmStop" condition, the only button that will be active is "Open Spindle's collet". All other functions/buttons are disabled in this mode.

4.4.11 "Check tool's ring"

This is an option that is active by default. It makes an additional check of the empty holders to provide a detection of any plastic tool rings left behind in the holders. This is a rare, but possible situation. Due to the relatively short time needed for this test we recommend you keep it active.

4.4.12 "Auto Tool Depth Update"

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. For example if you change the penetration of one tool in

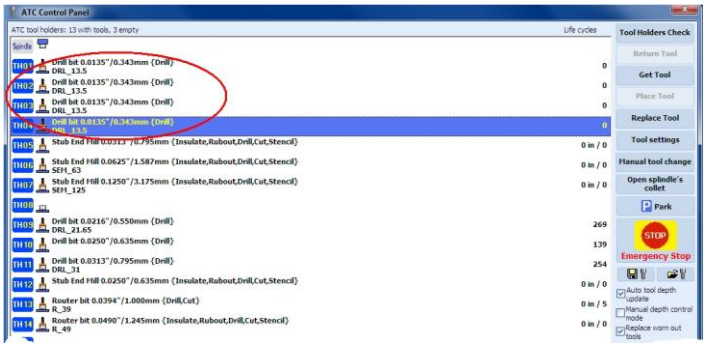
the "copper rubout" operation, the change will spread to the rest of the tools involved in this operation. If you like to have individual tool penetration depth for each tool this option should not be checked.

4.4.13 "Manual Depth Control Mode"

This option is not active by default. It allows a manual tool penetration setting after every tool change. The basic purpose for it is to allow the execution of G-code programs generated outside the PhCNC control software. In some cases they can be in the wrong format or using improper values for PhCNC.
If you activate a manual tool change during the execution of any machining program this will also allow you to set the proper penetration for the new tool.

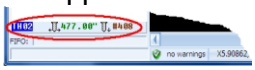
4.4.14 Automatic change of worn out tools

When the option "Replace worn out tools" is selected and the tool holders contain more than one tool with the same description PhCNC will return the worn out tool in it's original tool holder and take next one with the same description. In case that there is not a tool with the same description in the tool holders the software will request a manual tool change and wait until it is done. The life of the tools can be set when the option "Replace the worn out tools" is selected.
For the drill bits it is the number of holes and for all cutting tools (routers and end mills) it is the length of the cutting.
In the example below the 13.5 drill bit will be changed automatically when reach the maximum number of holes allowed.



This option is very convenient for projects that exceed the life of a certain tool for the project completion and allows the replacement to be automated.

Note:
If an ATC machine is in use, PhCNC is showing the selected tool life in real time. As the end of the tool life approaches the color of the announcement becomes more and more reddish.



4.5 ATC tool magazine

Depending on the model the tool magazine will contain 12, 15, 16 or 23 tool holders. In order to have fully automatic execution, it is recommended that all tools involved in your project are placed in the magazine. Keep in mind that all holes larger than the diameter of your cutting tool can be done with it (it is usually a 39 to 50 mil router bit). A highly complicated project can easily use 10 or more tools so this feature can be real handy. Placing the tools into the tool holders can be done two different ways.

- Using the "ATC Control Panel" (recommended)
- Manual placement, with the milling head in the "Park" position.

4.5.1 Tool placement using the "ATC Control Panel"

1. Click on the "Tool holders check" button. This will update the status of each holder in the magazine.
2. Select an empty holder by left clicking on it.

3. Click on the "Place Tool" button (it will be enabled only if the selected holder is empty). The system will move the milling head to manual tool change position.
4. Press and hold the Target button on the Calibration pad.
5. Place the tool in the spindle collet applying light "upward force" in order to have the tool ring in contact with the retracting collet.
6. Release the Target button on the Calibration pad. The tool should now be in the spindle.
7. Click "OK" to continue. The system will check the tool existence and proper position and then will place it in the selected holder.

If the system finds that the tool is not in the spindle collet or has a position outside the allowed limits it will issue an error message and will request to change/replace the tool. After the tool is placed in the holder you need to enter its parameters (type, geometry etc.) The easiest way to do this is to make a double click on the selected holder, or use the "Tool Settings" button. Both will bring up the main tool table. Select a tool from the table and close it, the parameters will be copied to the ATC Control Panel. If your tool is not in the table, use the "+ADD" button to enter it and proceed the same way. You have to repeat the operations above for every tool placed in the machine magazine.

4.5.2 Manual tool placement

Manual tool placement and removal is not recommended by us due to the fact that it creates a possibility to damage the Machine, Tool holders, Tools, and can cause errors in the tool description. The machine is the best tool for the job of loading and unloading the tools into the holders and should be the only method used.

Important:
If you Move tools or change them, PhCNC will automatically regenerate the machining programs to reflect the new tool position for every tool moved. If the spindle collet is occupied by a tool it will be returned to the holder after the program is completed. If the tool is an extra (Not belonging to a empty holder) PhCNC will go to manual tool change position and request it to be removed, in order to continue with the program. If your program contains tools that are not in the magazine PhCNC will request a tool change. Each program will end with placing the tool back in the corresponding holder. When you start your first program you may have a request to "warm up" the spindle if it has not been in use for the preset time. Always be sure there is a tool in the spindle before executing the spindle warm up. Also if you did not scan the tool holders PhCNC will do it automatically before the first ATC.

4.6. Machining Window (Pro version only)

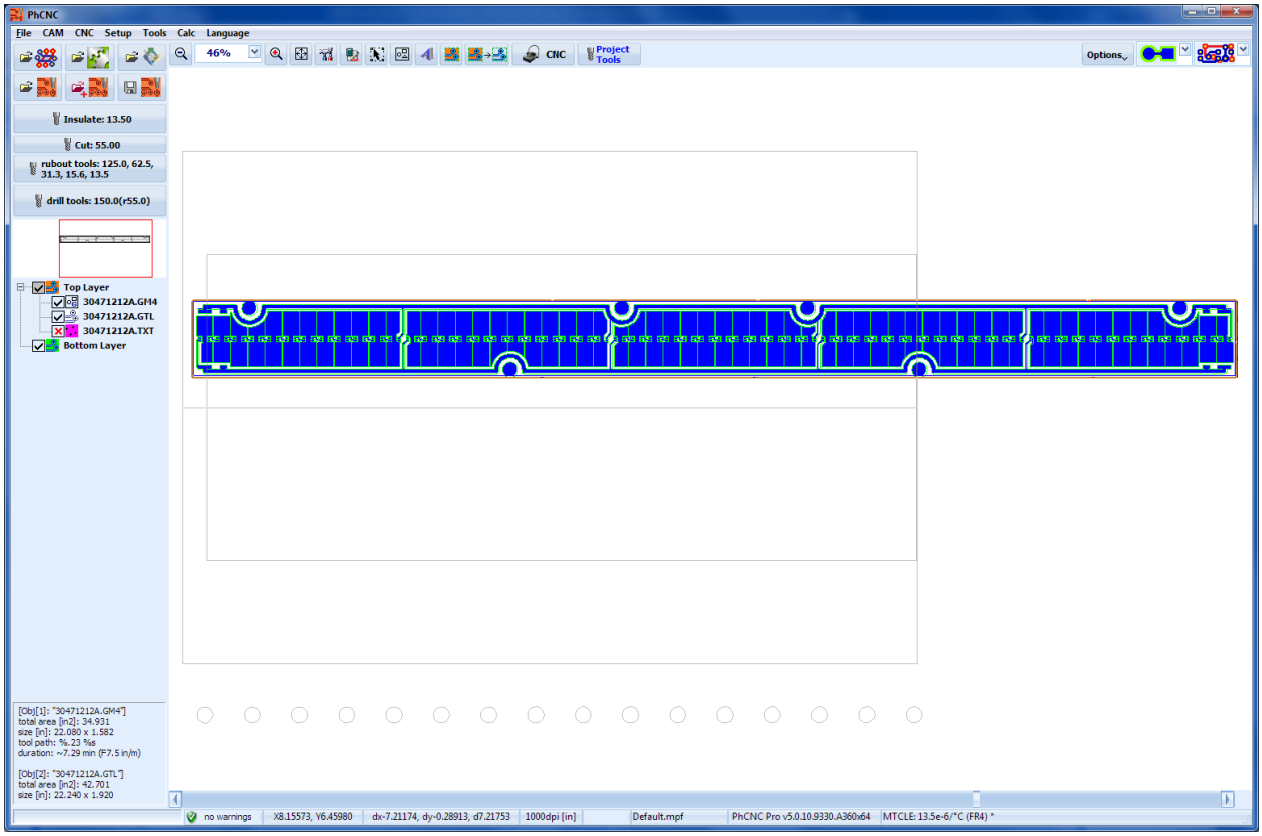
PhCNC Pro (v3.16.6 and up) has an option that we call "Machining Window". This option allows you to determine a square region of the board that will be machined in CNC mode. Any parts of the project outside selected region will not be machined.

4.6.1 Applications of the "Machining Window" option

- Machining projects that have one dimension larger than the machine X axis. Machines manufactured before August 2012 also can use this new feature but have more limits in Y axis dimensions due to the position of the calibration pad. Beginning August 1st 2012 we have new position of the pad that allows you to use more space on Y direction.
  The maximum size of the machining object can be up to 65 inch/1651 mm on X axis.
- Repeat the machining of a selected region of the project.

Important:
"Machining Window" option will be activated automatically in CNC mode if there are objects of the project that are out of the XY envelope of the machine (called further work or machining envelope).

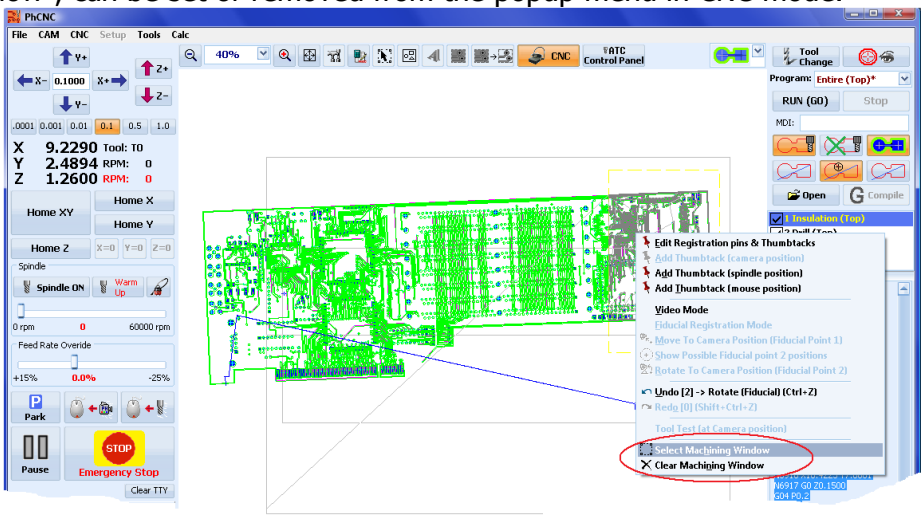
Example:



The project shown above can't be executed due to the fact that it exceeds the work envelope. Using the "Machining Window" option it is possible to be executed.
As shown below, PhCNC has limited the machining of the project inside the work envelope.
All machining programs generated will cover only the part of the project that is inside the envelope. The execution of the programs is identical to the projects that initially fit inside the envelope.
At this point we can move our board to the new position that the non machined part is inside the machine envelope. Next step is to use the camera/microscope to align the project data to the new board position (described in "6. Fiducial Registration"). The project shown below is quite tilted in order to emphasize that the actual mounting is not critical. Defining the "Machining Window" for the non machined part of the project is the next step. All machining programs generated will cover only the selected "Machining Window".

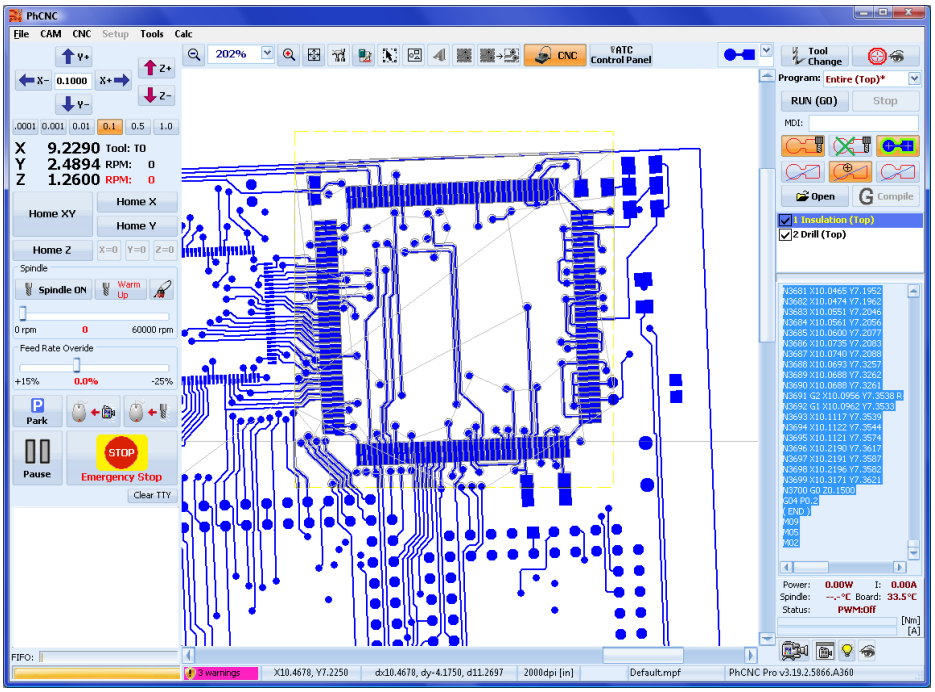
4.6.2 Selection of "Machining Window" region

"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.

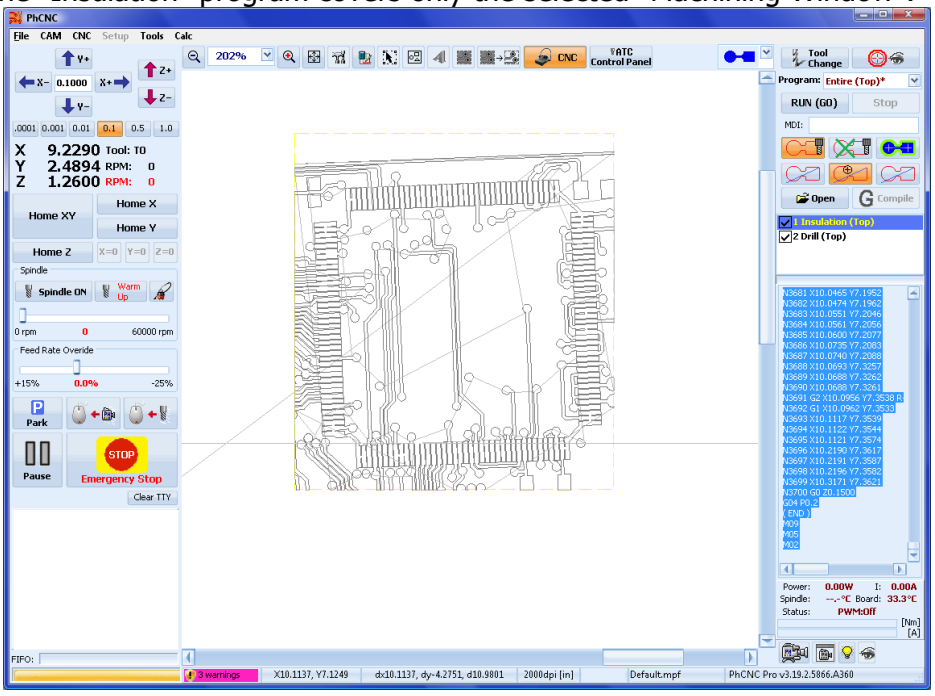


To select the "Machining Window" place the mouse cursor on the point that you like to be the lower left corner of the future "Machining Window" and make a right click. Then click on "Select Machining Window" line and move the mouse cursor to the desired right upper corner of the future window. Make a left mouse click that will complete the selection. Selected region must remain entirely in the work envelope.
After selecting a new "Machining Window", PhCNC will automatically recalculate all the machining programs in order to machine only the selected region.

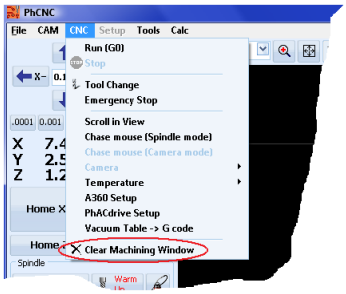
Example:
Shown below is a "Machining Window" that covers small part of the project.



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



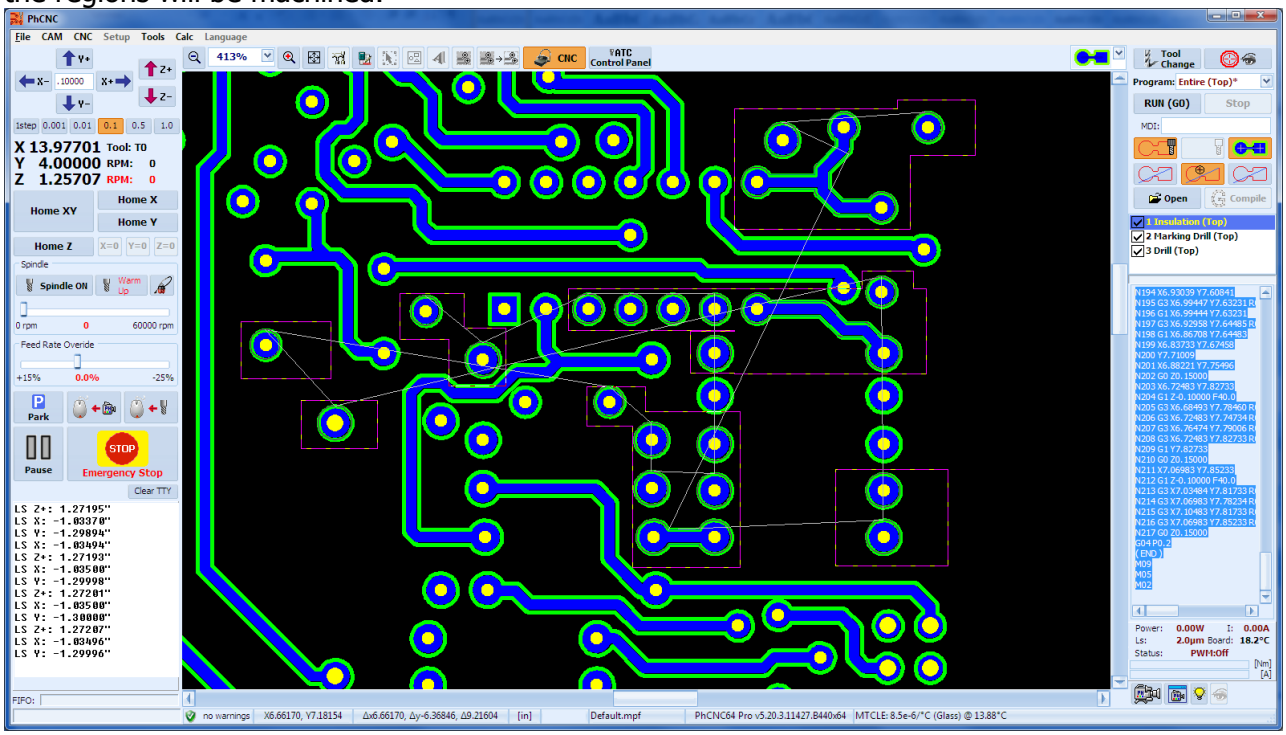
If you need to remove the "Machining Window" select "Clear Machining Window" from the popup menu. This function is also available in main menu CNC -> "Clear Machining Window" and in CAM mode of PhCNC.



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

4.7 Regions Selection

PhCNC 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.

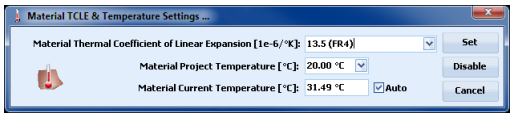
Clearing of the selected regions can be done using "Clear All Regions" of popup menu in CNC mode or in "CNC->Clear All Regions" (available in CAM mode too!)

4.8 Material Thermal Coefficient of Linear Expansion (MTCLE)

In the real world every material has a MTCLE. This leads to changes in its dimensions with changes of it's temperature. In many cases MTCLE can be ignored, but when the accuracy of the operations reaches the micron level it becomes the main source of errors.
Our PhCNC control software offers a thermal compensation of the machined material in order to achieve maximum accuracy of the project.
This option is available to all systems with calibrated screws A42x and A62x and also to all servo models A43x & A63x.

In order to achieve maximum accuracy the user must enter the right values of the following parameters.

- MTCLE of the machined material, or to select it from the list. By default it is FR4 glass epoxy printed circuit material.
- Material Project Temperature, this is the ideal temperature that the finished product will be exposed to and have maximum dimensional accuracy. For instance if the finished product is a microwave antenna that will be used in an office environment of 72 °F this is what you would set this parameter to. By default it is 68 °F (20 °C).
- Material Current Temperature. This is set automatically, PhCNC knows the approximate table temperature this is done mathematically from the internal temperature sensor, but requires the material to be placed on the table at least 30 minutes in advance in order to equalize its temperature with the table. If you wish to have better accuracy you may measure the temperature of the material and uncheck the auto box and enter it here.



Any change in MTCLE requires machine homming in order to take effect. For maximum accuracy it is recommended to have the system powered at least 30 minutes before usage.

Notes:
MTCLE is dependant on the temperature scale in use (°C or °F). For example, standard FR4 material has 13.5e-6/°C, or 7.5e-6/°F. The selection of the temperature scale can be done at menu CNC/temperature.
MTCLE is often called CTE (Coefficient of Thermal Expansion)

5. Video mode of PhCNC

All machines, are designed for factory or field installation of a video camera/microscope (VC). The video camera has an integrated mode in PhCNC control software. Having this feature will allow you to:

- Perform quality control of produced PCB's or other parts
- Conduct measurements with accuracy of 0.25 mil
- Perform fiducial registration of PCBs from the top layer to the bottom layer for unbeatable accuracy (see 6. Fiducial Registration).

Due to the precision Z axis positioning of our machines, VC has the capability to auto focus.

Basic buttons for activation and control of VC are located at lower right corner of CNC screen.

Button turns on/off PhCNC video mode.

Button shows/hides the popup window of "PhCNC Video". Hiding the "PhCNC Video" window does not turn off the video mode! This button is active only if video mode is on.

Button turns on/off VC lighting. VC lights come on/off automatically when video mode is turned on/off.

Button is to turn "Scroll in View" on and off. "Scroll in View" keeps the VC window inside the main window. This button is available only in active video mode.

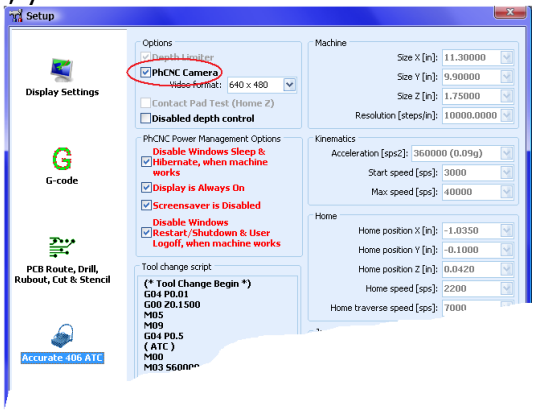
5.1 Video mode activation

Button is used to activate the video mode in PhCNC. Video mode is available in CNC mode only.

PhCNC shows live video in two windows:

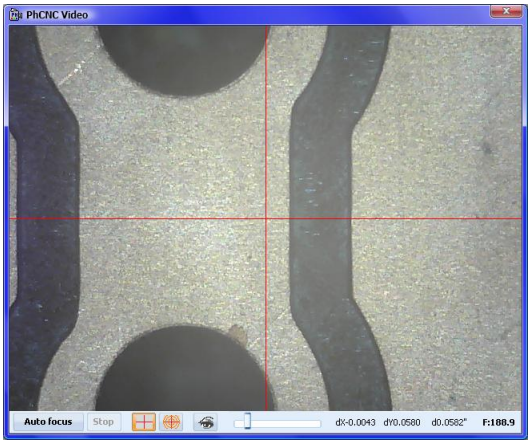
- The "PhCNC Video" popup window shows the VC image and is not always directly related to your project, and conforms to functions on the window such as Zoom IN/Out, positioning with mouse etc.
- The "CNC Video" is integrated in the CNC window of PhCNC and shows the image placed over your tool paths and conforms to functions on the project screen such as Zoom IN/Out, positioning with mouse etc.

Important:
It is easier to position the camera by clicking on the project in the CNC window using "Chase Camera Mode" and the "Hole Finder Photo Tool" than using the jog arrow buttons. The same arrow buttons however are the perfect tool for measurement in "PhCNC Video" mode. In order to use the VC and to be able to activate PhCNC video, you have to check the PhCNC Camera box in the Setup menu.



5.2 Popup window of "PhCNC Video"

"PhCNC Video" shows real time video in a separate window as shown below.



When you activate video mode the "PhCNC Video" window comes up. First you have to focus the VC. To do so, simply click on the "Auto focus" button.

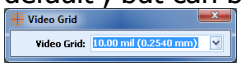
Note:
"Auto focus" procedure may not work if your Z axis is not homed to the work piece surface.

5.2.1 "PhCNC Video" control buttons

Button "Auto focus" – starts the automatic camera focus procedure.

Important:
PhCNC is searching for the best focus in the range of Z+0.1500 to Z+0.5000, therefore your Z axis should be homed. See **Section 4.1** for Z home procedure. If the VC cannot reach its focus distance, you will have some error in your measurements. Also all VC adjustments and calibration (described in APPENDIX-A), should be performed after the "Auto focus" procedure. "Auto focus" takes 5-7 seconds and can be canceled with the "Stop" button right next to the "Auto focus" button. Manual focusing and/or focus corrections are available using the mouse wheel or the Z axis jog buttons when video window is open.

Buttons and shows/hides the target grid. The diameters of the circles are 20/40/60/80 mil by default , but can be set to different values by pressing on the button :

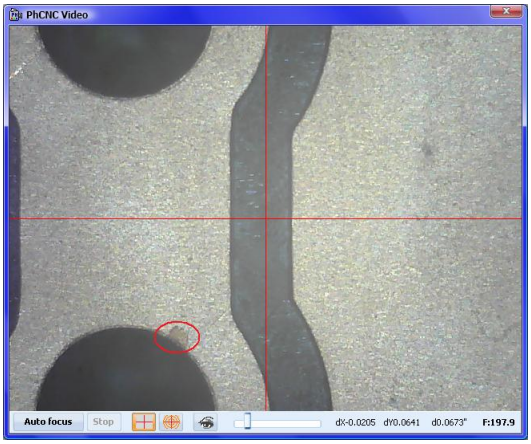


Button turns on "Scroll in View" of the "CNC Video" window, and keeps the integrated video window inside the main window (when possible).

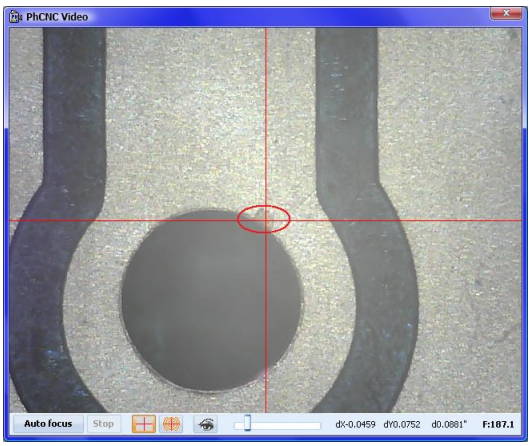
5.2.2 Other functions of "PhCNC Video" window

A right mouse click moves the VC center to the point of click. In the example bellow the red circle shows the right click point.

Note:
All revisions later than v1.11.2 require the "Control" button on your keyboard to be held down during the execution of this command.



The VC moves, till the selected point is in the center of the cross hairs.



If the selected object is not positioned in the center of the cross hairs, your VC will need to be calibrated. VC calibration is described in APPENDIX-A.

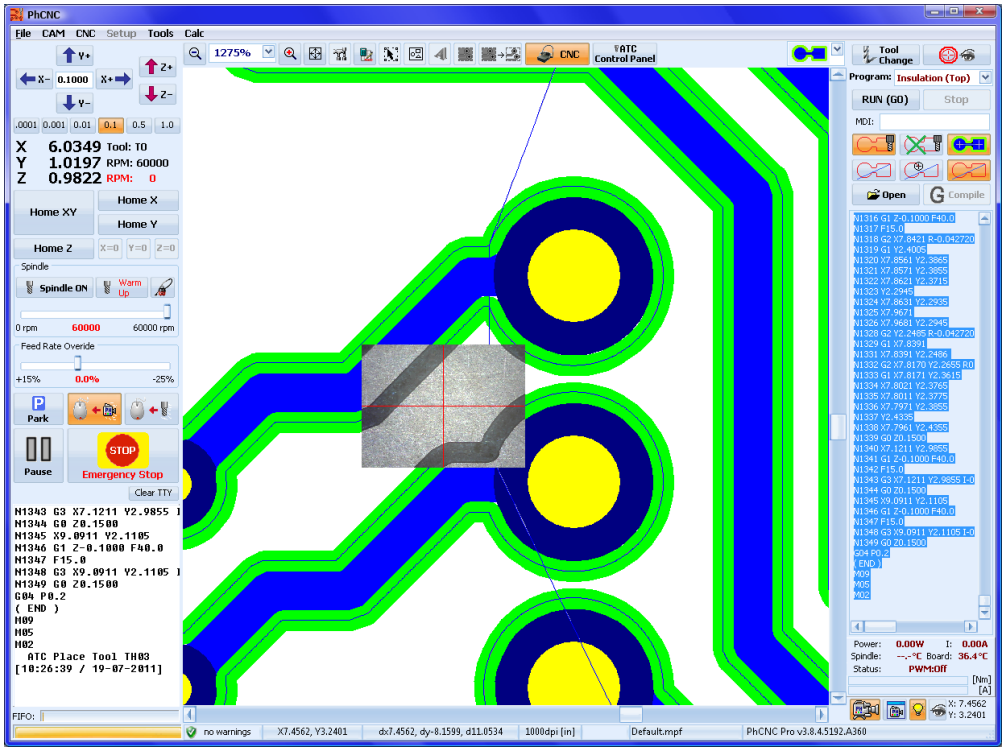
Left mouse clicking will set dX, dY and d counters to 0, they will show the relative position between the click point and current position of X and Y. These are very useful for obtaining a distance measurement.

Note:
You can measure distances outside of Video mode in CAM/CNC modes as described before.

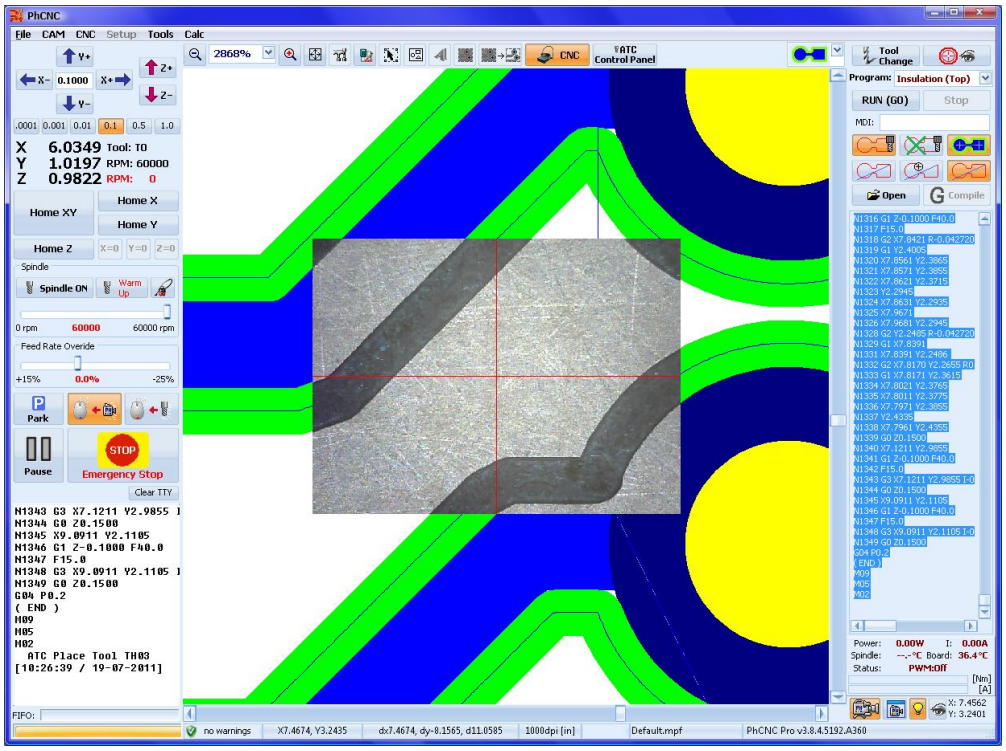
5.3 Integrated window "CNC Video"

"CNC Video" is actually a real time video projected over your tool paths in CNC mode. This makes it an extremely useful tool for inspection and measurement. It is scaled to the actual size and all operations such as Zoom in/Zoom out, positioning, and distance measurements are valid. When video mode is on this window is always active no matter what the status of "PhCNC Video" window.

Below you can see how it looks:



After Zoom in:



Button is to activate "Chase Mouse (camera mode)" It works the same way as "Chase Mouse (spindle mode)", instead of positioning the spindle it is positioning the center of VC image to the selected point. The usage is also the same. Left click on the "Chase Mouse (camera mode)" and right click on the desired point. The VC will position to this point.

Important:

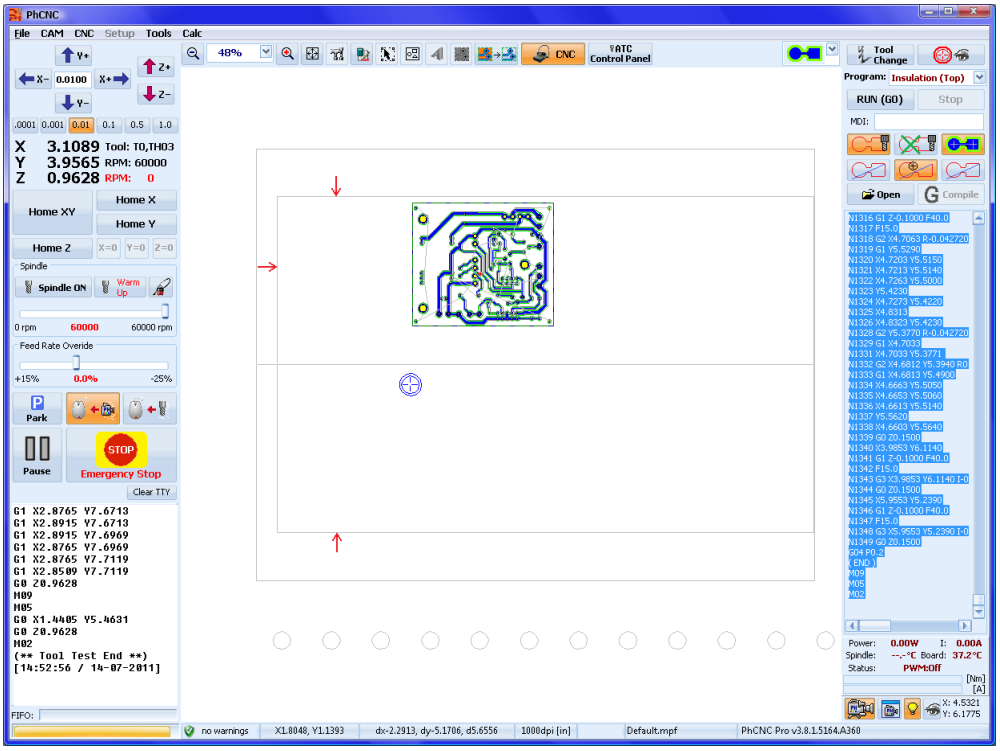
In order to improve operator safety (beginning with PhCNC v1.11+) "chase mouse" functions (spindle or camera mode) will require the control button on the keyboard to be activated (pushed) during the command. When you select "Chase Mouse (Camera mode)" and push the control key the cursor will look like this . Right clicking with the mouse will bring the camera to the position of the cursor.

Note:

Pressing the "Control" on your keyboard will suspend the "life video" in the embedded video window until the control button is hold down.

Note:
Do not click below the camera access frame (lower Y coordinates) it will cause the machine to hit the lower Y limit switch. (Upper Y coordinates) are not a physical limit and can be crossed if needed.
they are shown only to know where your selection of camera related objects can appear after you flip the board.

The VC image center and spindle center are shifted at a constant distance. Due to this offset not all the work surface is available for VC inspection and measurements. Shown on the screen below are the limitations that exist (including flip over for double sided boards). If you try to position outside these borders (pointed with red arrows) you will hit a limit switch.



Note:
All machines A36x/A56x manufactured after November 1, 2009 have the new controller, PhCNC360 R3. These machines have the capability to control the intensity of camera illumination. With these machines and newer models you will have the slider in your video window.

6. Fiducial Registration (Pro version only)

The procedure of matching the top and bottom layers of your project is named Fiducial Registration (FR). Typically it uses the holes drilled as the last operation of machining the top layer. In some cases the operator can add additional holes outside the project that can be used too. Depend of the system model, a calibrated camera is needed for this procedure. All ATC models has the camera as a standard, factory mounted unit. MTC (manual tool change) models can be ordered with this option or it can be added later by the customer.
PhCNC allows Fiducial Registration to be done in a few different ways:

- Auto Fiducial Registration (AFR): Fully automated process.
- Assisted Auto Fiducial Registration: Semi-automatic process. The operator is using the jog arrows (or mouse) to guide the camera to holes. Needed for large boards with small number of holes.
- Fiducial Registration: Manual procedure that is using two holes.

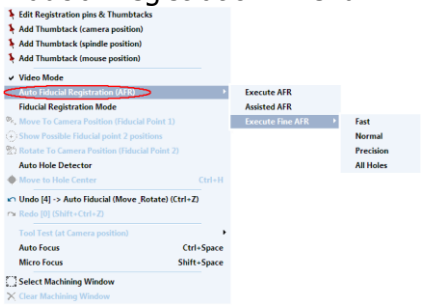
Supporting procedures that can be used as stand alone:

- Fine Auto Fiducial Registration: Fully automatic process of final precision positioning of the project. It is activated automatically by Auto Fiducial Registration procedures.
- Auto Hole Detector (AHD): This procedure can find the center coordinates and diameter of any hole in the camera window. Used in all Auto Fiducial Registration procedures.

6.1 Auto Fiducial Registration

PhCNC (v5.8.x+) allows Full Auto Fiducial Registration of the project, using the holes in the project drill file without any need of separate special fiducial holes.
PhCNC, can use any hole in the project with a size 10...90 mil (0.25mm ... 2.3mm), to do the registration. Hole diameter, from drill program is also used for identification using acceptable rounding.

All options are available from "Auto Fiducial Registration" menu:



Important:
The menu above is active only if a project loaded, contains at least 2 holes with the sizes in the range of 10...90 mil (0.25mm ... 2.3mm)!

As we mentioned above, PhCNC offers 2 modes of Auto Fiducial Registration (AFR):

- Execute AFR
- Assisted AFR

In both cases the process is automatic. The difference is the camera movement. In "Execute AFR" all camera movements are controlled by PhCNC. In "Assisted AFR" the camera movement is guided by the operator. This mode is needed for boards that have large surface and small number of holes. The assisted mode is then shortening the time needed to find holes for registration.

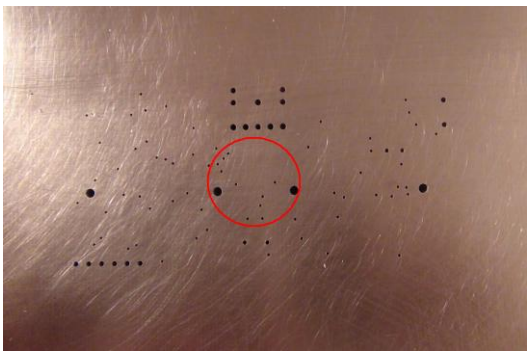
6.1.1 Execute AFR

If "Execute AFR" is selected, the process of Fiducial Registration is completely automated and the holes search movements of the camera are under PhCNC control.

Before activating "Execute AFR", the following steps must be done:

- select machining program for Bottom or Top layer, that will be a subject to Fiducial Registration.
- turn on video camera (all PhCNC AFR functions are disabled without it).
- place the camera aproximately in the center of the project (camera lights are a good guidance), execute "Auto Focus".

In the sample project below is shown the recommended zone (red circle).



"Execute AFR", can be canceled at any time using the Stop button!

Once the initial registration is completed, the system will continue with the precision part, "Fine Auto Fiducial Registration" (FAFR) that is fully automated process.

In the video clip below, the AFR is demonstrated:
http://youtu.be/JPwkqxDNjLY

6.1.2 Assisted AFR

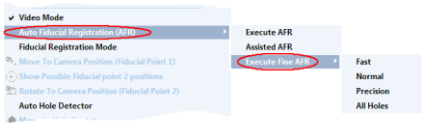
The option "Assisted AFR", allows to execute AFR, with camera movements under the operator control. It is designed for large surface boards that have small amount of holes. Operator's movement of the camera will short the time of finding the needed holes. Once the initial registration is completed, the system will continue with the precision part, "Fine Auto Fiducial Registration" (FAFR) that is fully automated process.

Note:
All changes in project position can be restored to original position using Undo (Ctrl+Z)!

In the video clip below, the Assisted AFR is demonstrated:
http://youtu.be/PAmpTHY4M48

6.1.3 Execute Fine AFR

This function require manual or automatic registration to be done in advance. In case that it is not done, this process will fail. AFR calls this procedure automatically, but it can be repeated with different grade of accuracy (if needed). See the options below.



- Fast: Uses not less than 18 holes of the project (default option)
- Normal: Uses not less than 32 holes of the project
- Precision: Uses not less than 50 holes of the project
- All Holes: Uses all holes in the project, that meet diameter requirements - 10...90 mil.

Due to sophisticated statistical algorithm, higher number of inspected holes increases the actual accuracy of the registration.

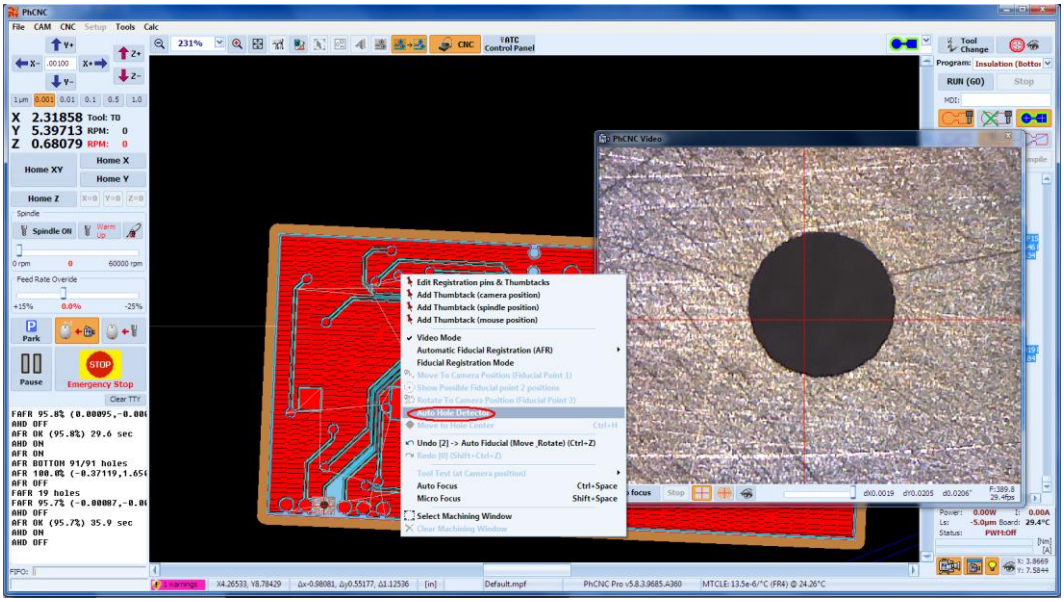
Note:
"Execute Fine AFR", can be canceled at any time using the Stop button!

All changes in project position can be restored to original position using Undo (Ctrl+Z)!

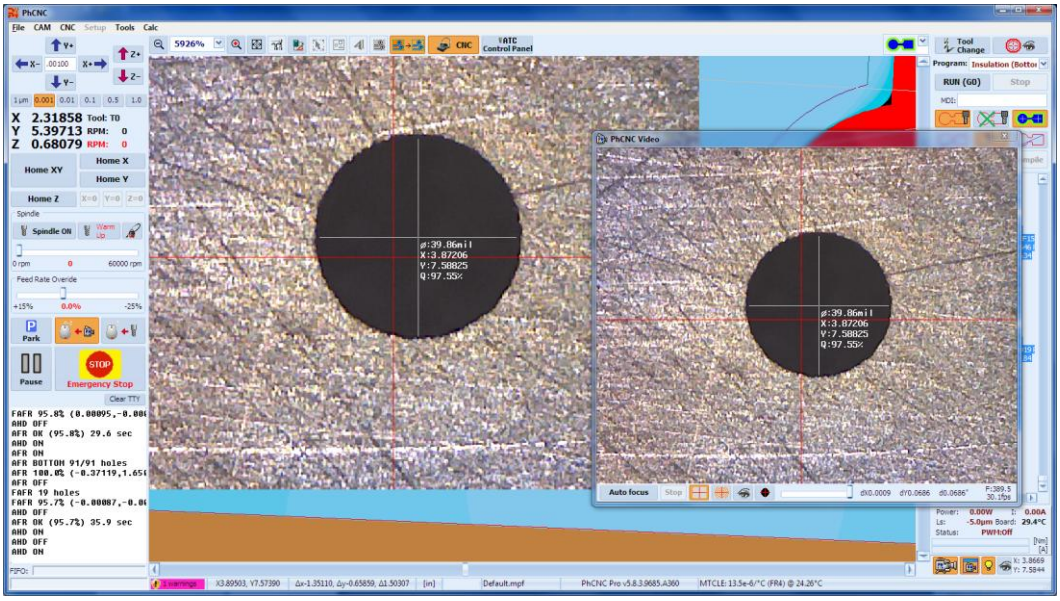
The video clip below shows FAFR:
http://youtu.be/jpFj4fkuLPE

6.2 Auto Hole Detector

PhCNC (v5.8.x+) offers the function "Auto Hole Detector" (AHD). This function allows detection of all holes in the video window to be recognized in real time. It is used in all AFR procedures, but can be activated as a stand alone function too.



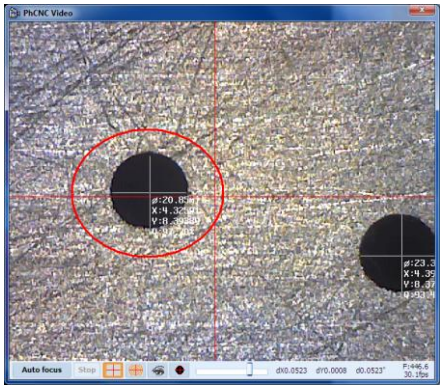
For any hole found you will have the following info: diameter, (X,Y) coordinates and quality of roundness in %.



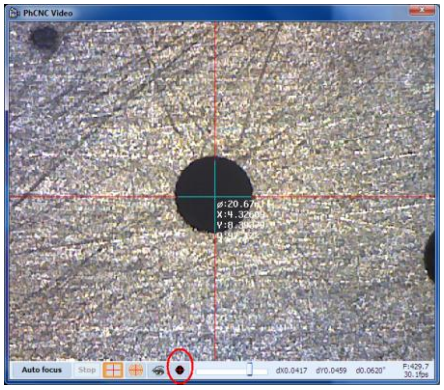
AHD function allows recognition of unlimited number of holes that fit in the size limitations of 10...90 mil.

6.2.1 "Move to Hole Center"

The option "Move to Hole Center", allows fast and precision camera positioning in the center of the hole on the video camera screen. Short cut "Ctrl+H". In case that more than one hole is on the screen, the closest to the center of the video window will be selected. In the sample below the one with red circle.



The button in the video window shown below in red, activates "Move to Hole Center". If there is no hole on the video window the button is deactivated.



Note:
Button is visible (active or not), only when AHD procedure is on. AHD is called automatically by the use of "Auto Fiducial Registration" (AFR).

The video clip below demonstrates the use of AHD:
http://youtu.be/EM5zQBNgSjo

6.3 Fiducial Registration (Manual mode)

Along with Auto Fiducial Registration (AFR), PhCNC offers also manual procedure for precision optical registration. It is the process of using two reference points to recalculate and realign your project to the current board position. Manual Fiducial Registration, can be used in projects where AFR is not possible (projects without holes, oversize on X axis projects etc.).
Here are some of the applications.

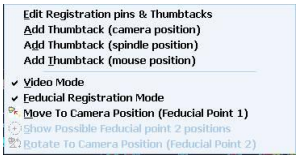
- Align your bottom side of the board to the top one (already machined and drilled), using two holes/points for reference
- Reinstall your board after plating through holes
- Any other precision alignments between your work piece and your project

You may use this new option with or without registration pins. If you use the existing pins and "Fiducial Registration" your accuracy will increase. If you prefer to use tape, tape your board to the backing material make sure that both, your work piece and the backing material are taped down securely on all four sides. Our experience shows that if you forget to tape even a single side, you may experience significant errors.

Below is the order of operations to produce a double sided board using "Fiducial Registration":

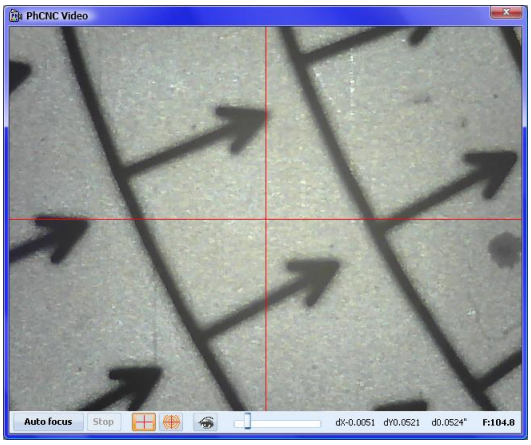
1. Mount your work piece on the backing material. You may use registration pins, or tape but it is not obligatory. Just make sure that the backing material and your board are firmly fixed to the machine table.
2. Execute all programs for top layer (drill, insulation, and rubout). Drill must be executed. If you do not have drilling in your project add two dedicated holes using the "Insert Function" from the popup menu (See 2.18). The best place to add them is outside your project and along the X axis. Using dedicated holes for fiducial registration is strongly recommended even if you have a drill program. A size of 40 to 65 mils is recommended. For maximum accuracy you may use stub end mils instead of drills. Also recommended is to reduce the plunge rate for this tool to 15-20 IPM (Inch Per Minute). The goal is to produce holes perfectly perpendicular to your work piece. This will transfer the top layer coordinates to the opposite side with maximum accuracy.
3. Select a program to execute for the bottom layer. In most cases it will be Insulate bottom.
4. Turn the camera mode on using the button or from the popup menu "Video Mode" If the camera mode is not on the Fiducial registration mode is not allowed.

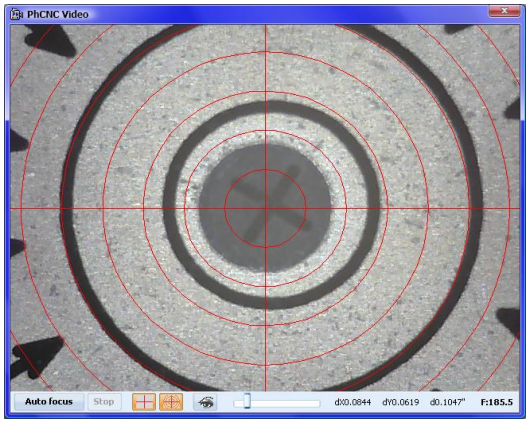
5. Focus your camera using "Auto focus" button on the video window. In order for the "Auto focus" to work properly the Z axis needs to be homed to your work piece. (See 4.1 for Z home procedure: button "Home Z")
6. Activate "Fiducial Registration mode"

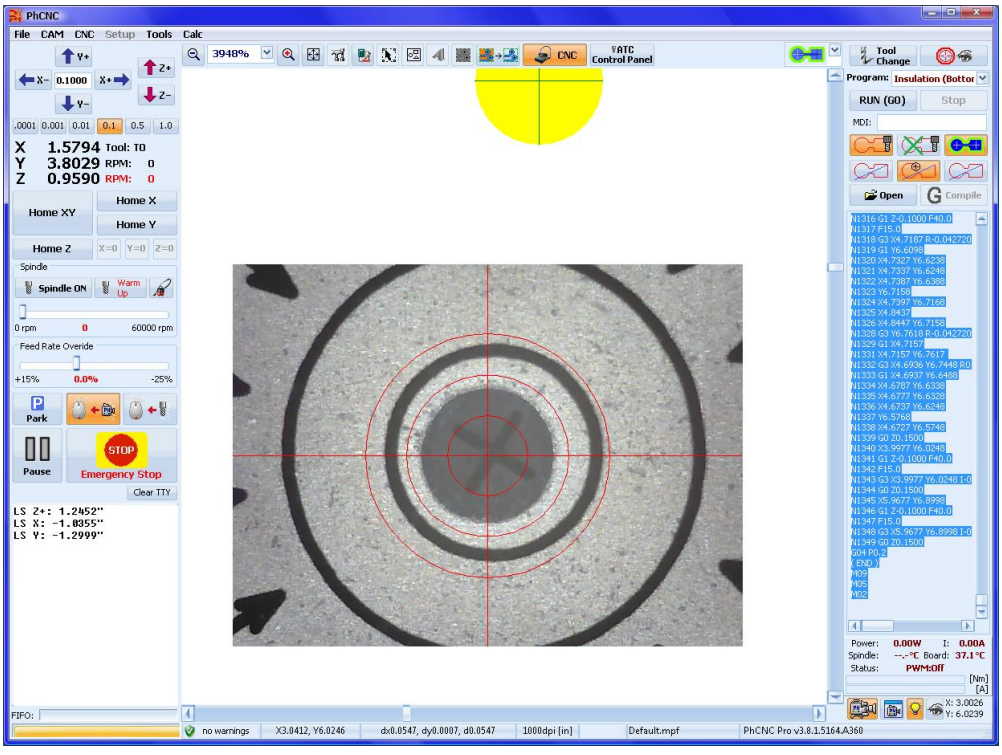


You will see the following changes in your screen:
- The centers of all flash apertures and drill holes are shown.
- Drill information is visible, no matter the layer shown.
- In the video (CNC Video window) the centers of the holes will be visible.

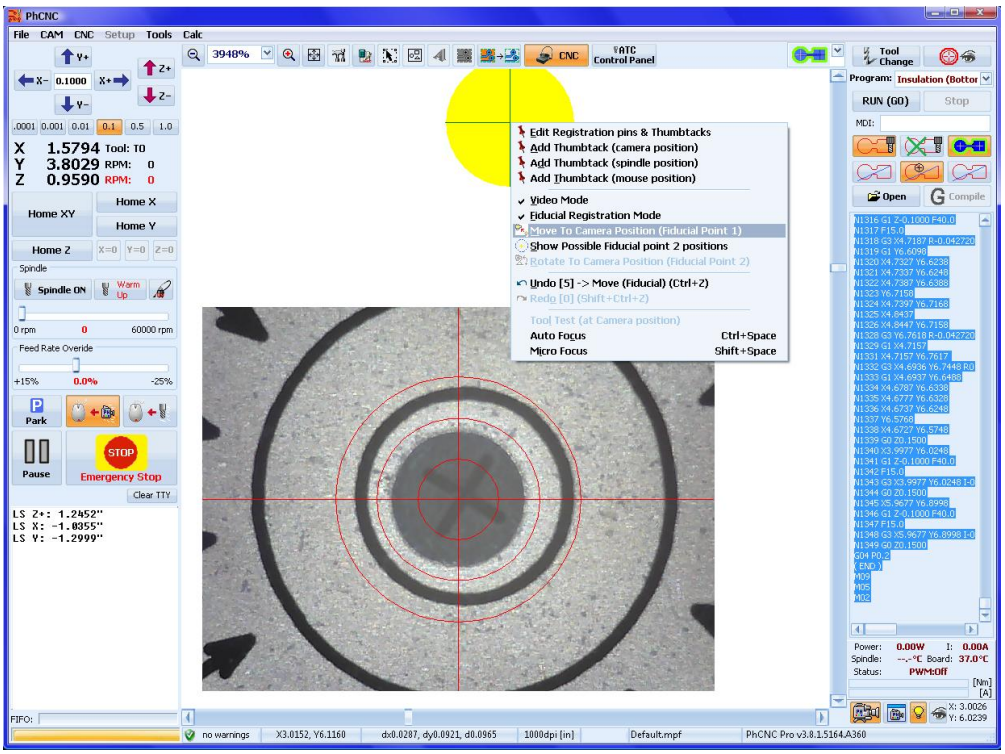
7. Find the first reference hole, or other reference point (Fiducial Point 1). Finding the hole can be difficult due to the necessary magnification to achieve accurate positioning. We offer a Hole Finder Photo Tool (included with the camera option) that will help you to find the reference hole quickly. Place the Hole Finder Photo Tool center at the center of your (reference) hole. It consists of a center cross hair and thousands of small arrows (clearly visible with the camera) pointing to the center. You may need to use a magnifying loupe to place it on the reference point. The placement is not critical, and you do not have to fix it to the work piece, it is just to find the reference hole. Then follow the arrows using the camera chase feature by right clicking + control key inside the camera window. Once you can see the hole on the camera, remove the photo tool and align it precisely to the center, using the target circles.



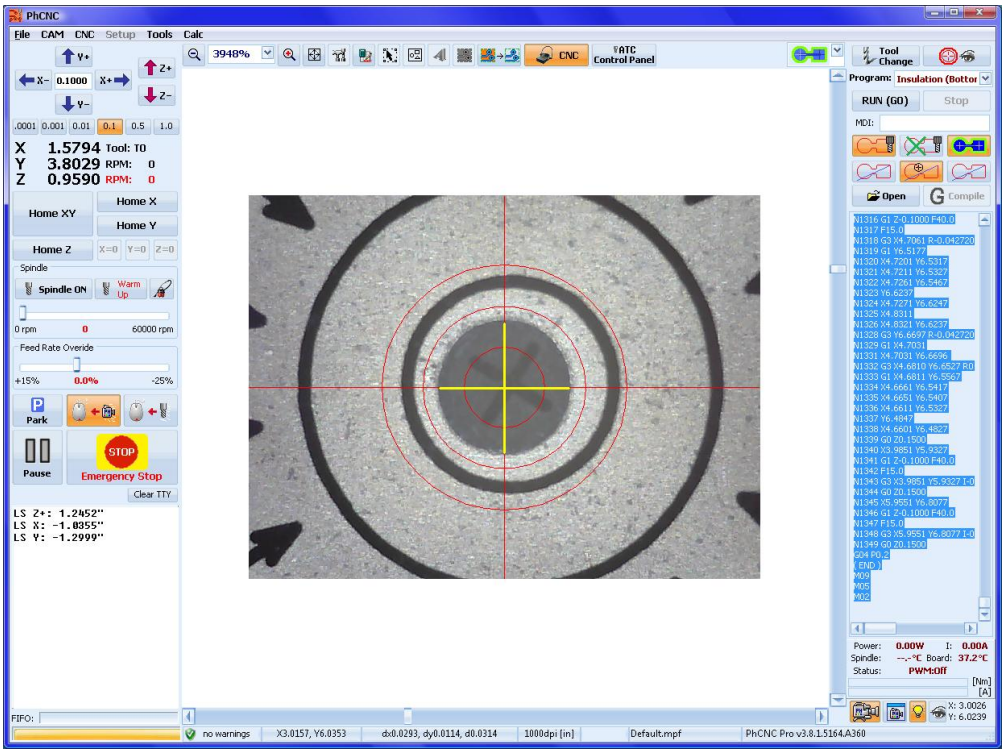




8. Declare Fiducial Point 1 with your camera on the center of the hole (reference). Close the "PhCNC video" window and find your "Integrated CNC video". (Use 5,000 to 20,000% zoom for maximum accuracy). Place the mouse pointer on the center and make a right click (CONTROL BUTTON must NOT be pushed). Select "Move To Camera Position (Fiducial Point 1)".

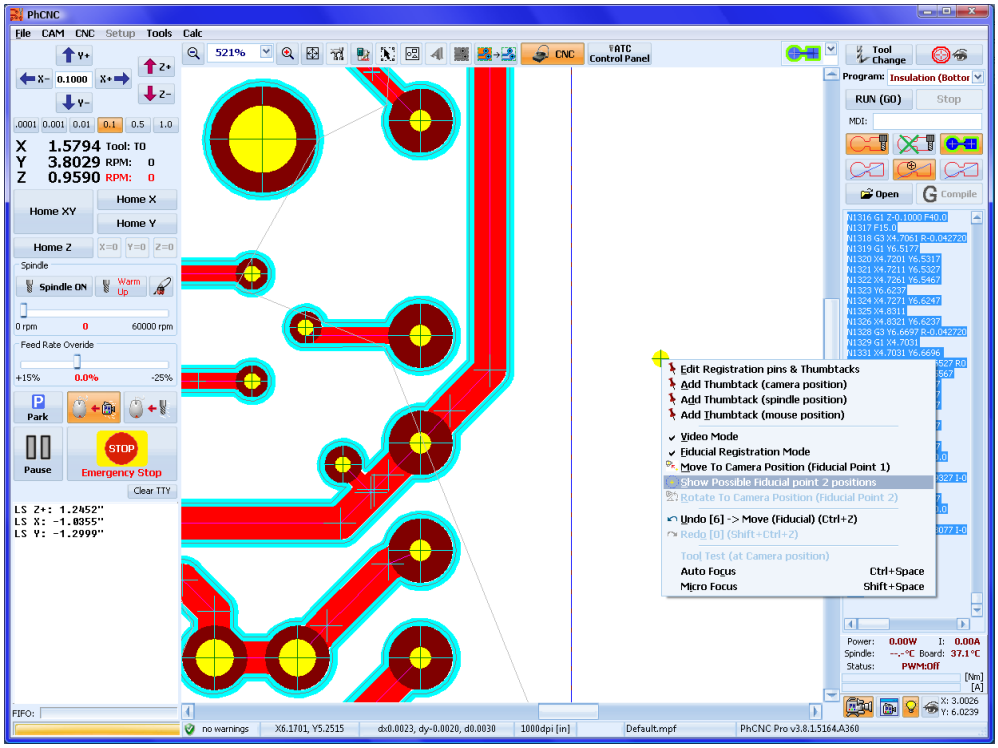


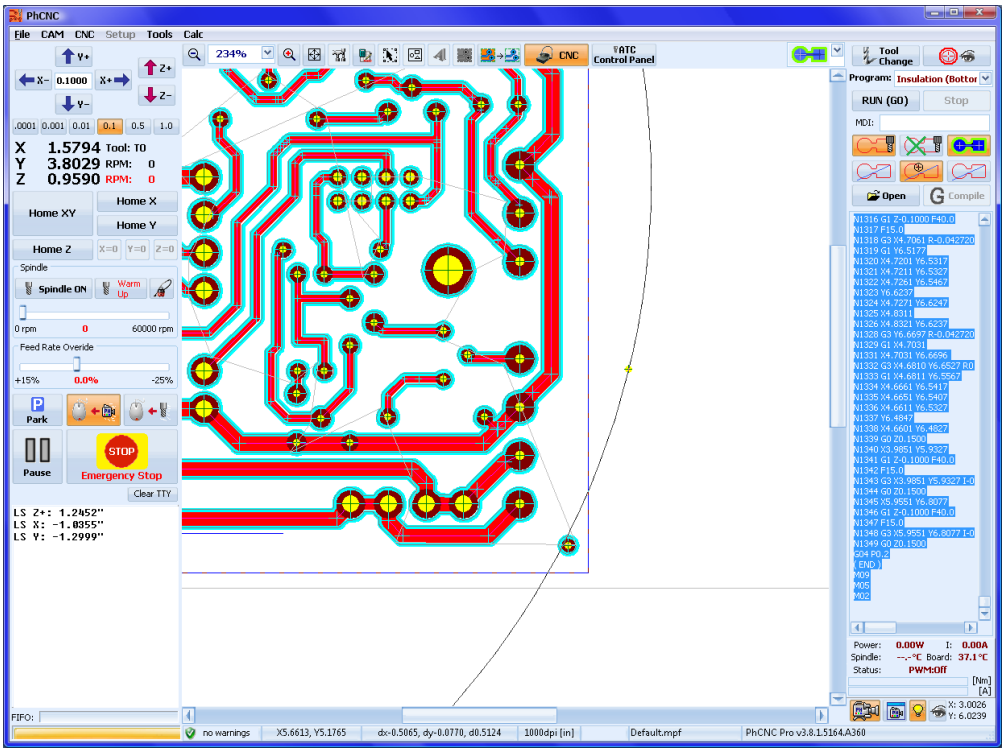
Now your project data is moved to the actual position of your hole (reference). The yellow cross hair shows the coordinates from your drill file. All G-Codes for your machining programs will be recalculated at this time.

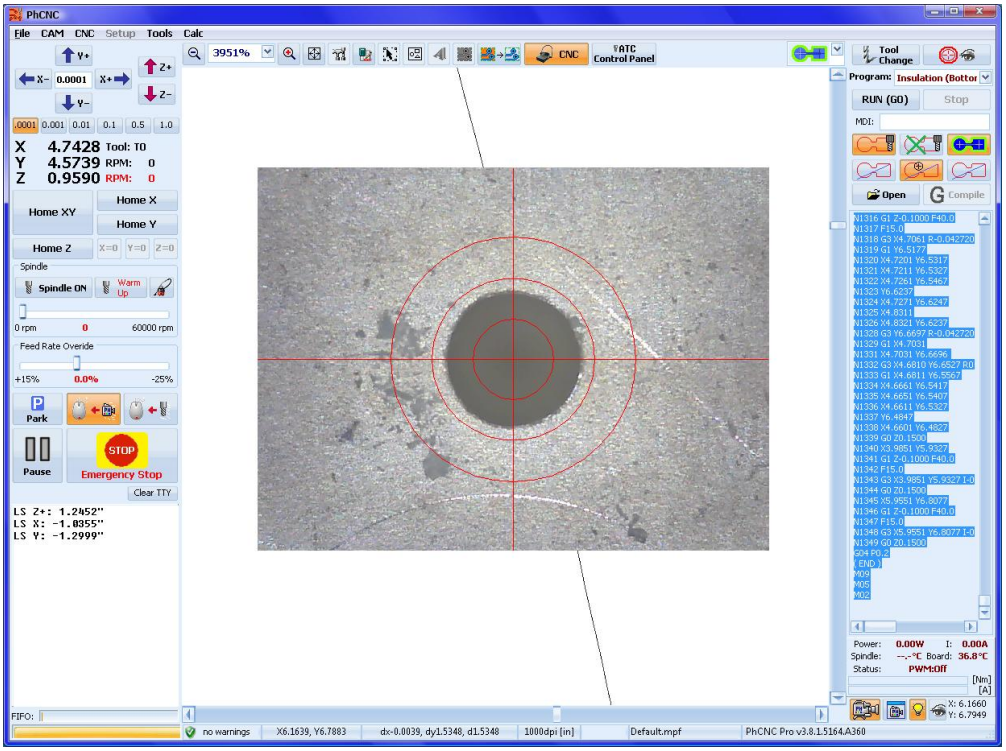


9. Finding the second point for registration (Fiducial Point 2): Once the first registration point is declared the project will be aligned to it. Now we need the second point to rotate the project in order to get a complete alignment. To find the second point, you may use the function "Show Possible Fiducial Point 2 positions" that will draw a circle of possible positions, centered at the first point and

having a radius equal to the distance between references, or you can use the photo tool described in the previous two steps. Again using a set of dedicated holes (references) will decrease the chance of errors and the wider apart they are the more accurate your alignment will be. See Insert function (2.18).

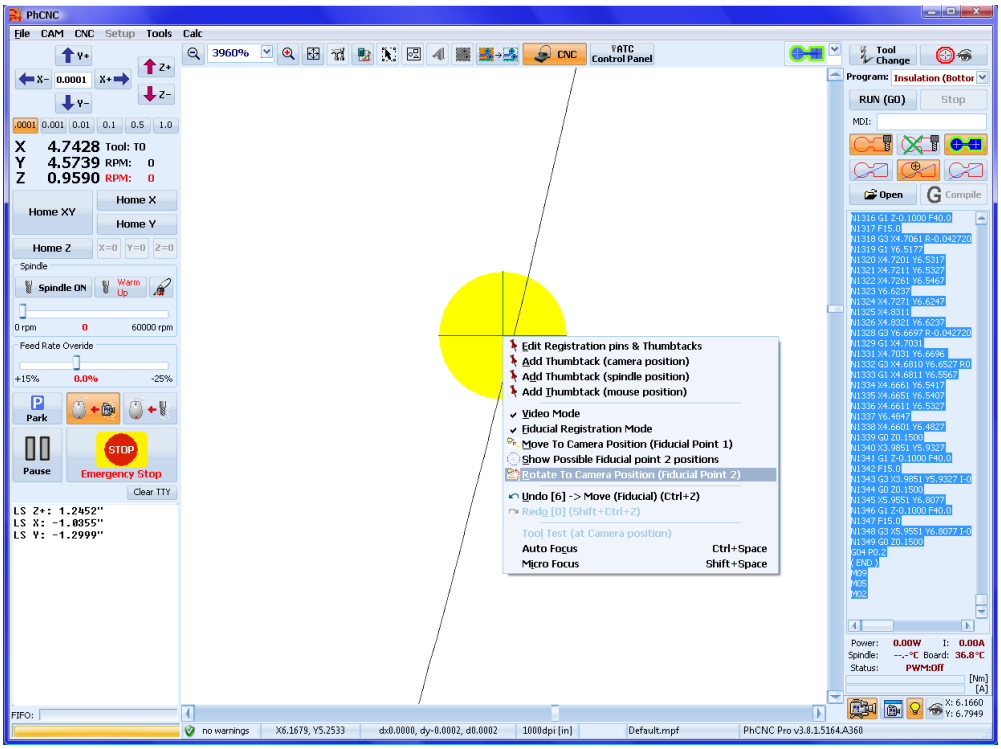




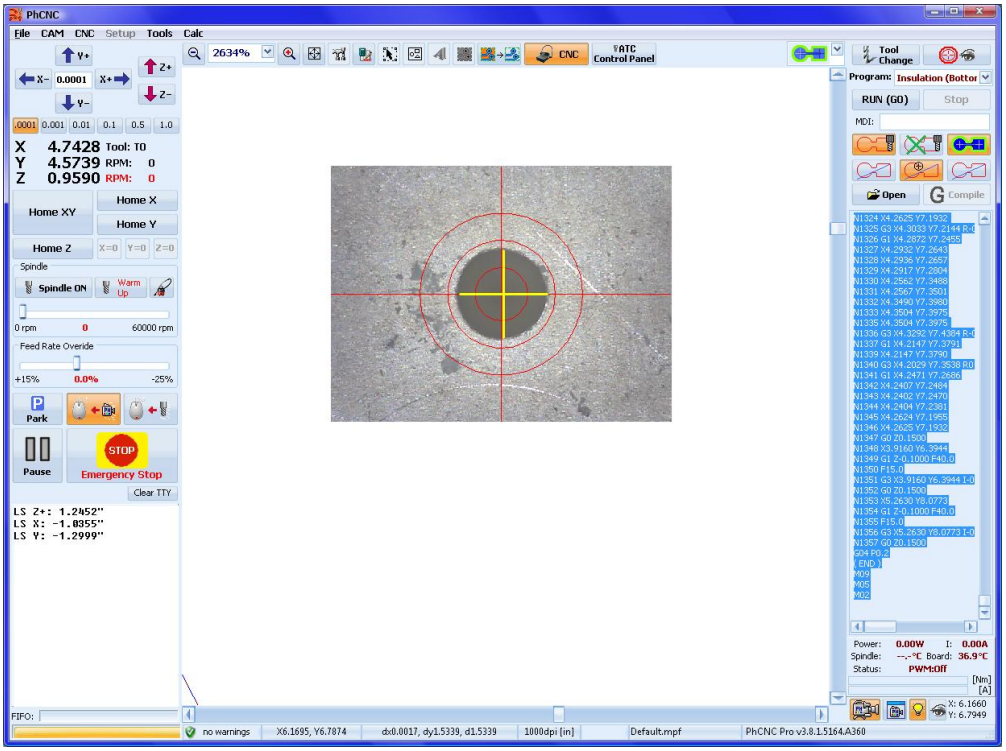


Note:
If your board is aligned to X axis to the best that human eye can do or you are using registration pins, the second fiducial point (most likely) will be in the camera window already.

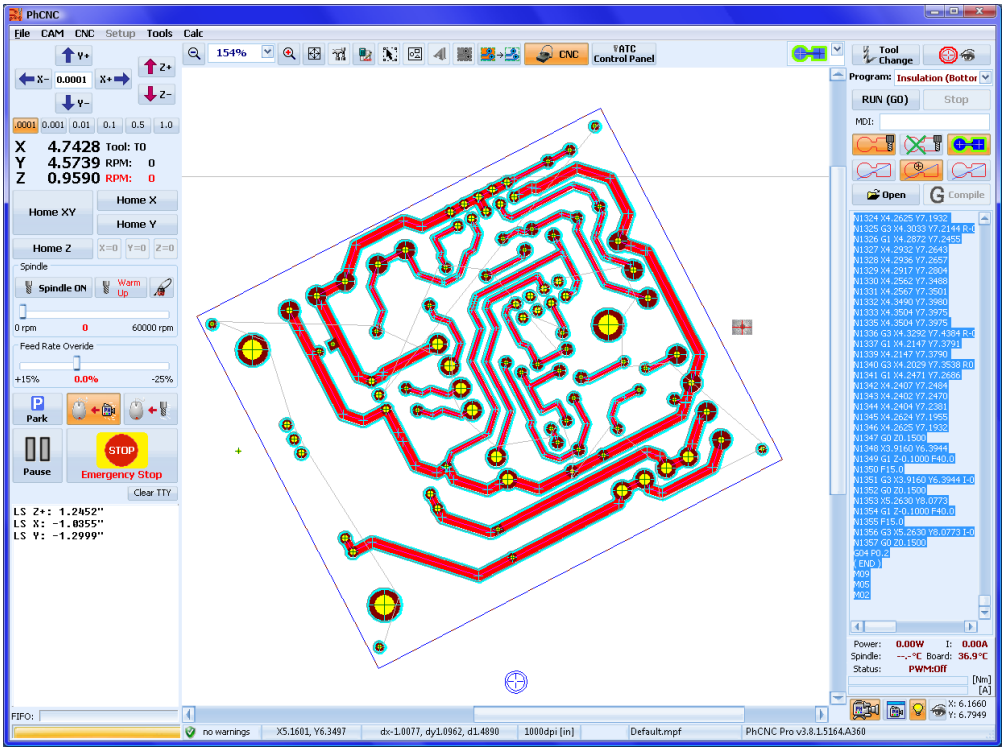
10. Declaring the second fiducial hole (reference): Once the second fiducial point is found, place the mouse pointer on the center (use 5,000 to 20,000% zoom for maximum accuracy) and make a right click (CONTROL BUTTON must NOT be pushed). Select "Rotate to Camera Position (Fiducial Point 2)"



After the rotation is finished the second fiducial registration point (of your project) will be centered in the camera's cross hairs:



Your project now is completely aligned to the work piece.



Notes:

- Unselecting the "Fiducial Registration Mode" (from the popup menu) will reset the registration (references) and will redraw the window to the normal mode.
- In our example the work piece is rotated considerably in regard to X axis. It was our intention to show you that any position is possible for the fiducial registration option. If you place your work piece in parallel to X axis you will reduce the chances of error selecting the desired references.

IMPORTANT RECOMMENDATIONS:

- When it is possible use dedicated holes for front to back registration.
- Place them along the X axis and utilize the maximum possible distance

- Create the dedicated holes using as rigid as possible tools such as drill bits larger than 40 mils or even stub end mils (the best accuracy). Tools larger than 60 mils will not fit in the camera window.
- Use a plunge rate between 10-20 IPM for the tool that drills your registration holes.
- Using existing (in the project) holes with a small diameter (20 mils or less) may invoke significant error due to the drill bit flexibility.

Important:

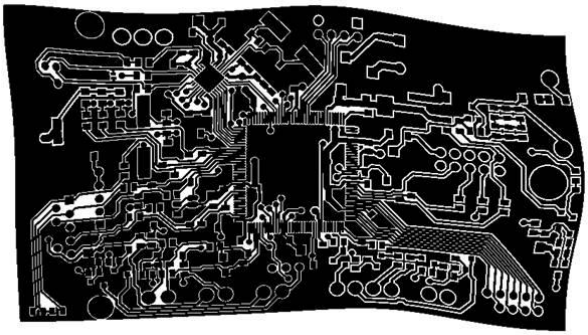
Press and hold "Shift" on your keyboard to hide the image transmitted from the camera. The camera target grid will remain on the screen, showing an accurate camera position. This function is designed to make precise positioning over the program features that are invisible under the camera image.

6.4 "2D Fiducial Transformation" Function

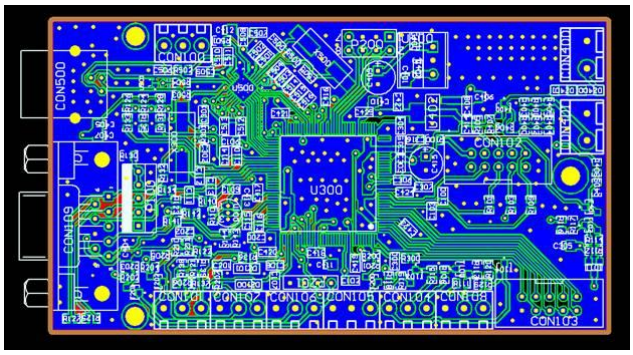
The "2D Fiducial Transformation" function of PhCNC allows Fiducial Registration of a project loaded in PhCNC and physical object, created with other machine and/or technology. Difference in dimensions, geometrical distortions etc. Graphical examples below are exaggerated in order of better understanding the function. Typically they are invisible with naked eye. Actually this function can have many applications. Here is a typical one.

A customer have to produce 100 boards with relatively low precision and density. At this point the customer decides that using chemical etching is far more economical compare to the tools that he/she will need to do the batch. Using pre-laminated (with photo resist) materials.
Prints a photo tool on its laser printer and then expose and etch the boards. As the accuracy of a printer is far less than needed at the time when the drilling comes up there are significant difference between the pads and the drilling program for the project.

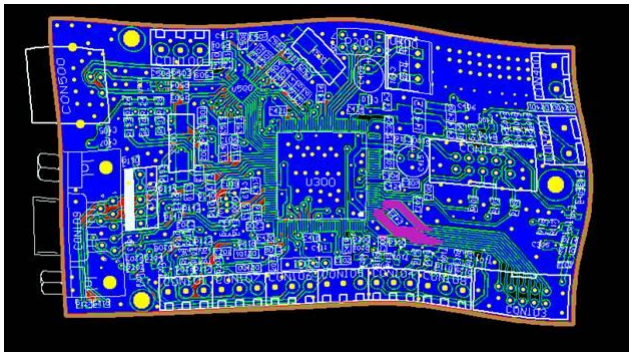
2D fiducial transformation allows to solve this situation and have the project be drilled cut and all other related operations with pristine accuracy.



As the project loaded in PhCNC looks like this:



Then with the help of 2D Fiducial Transformation (using our camera), the project can be transformed into:



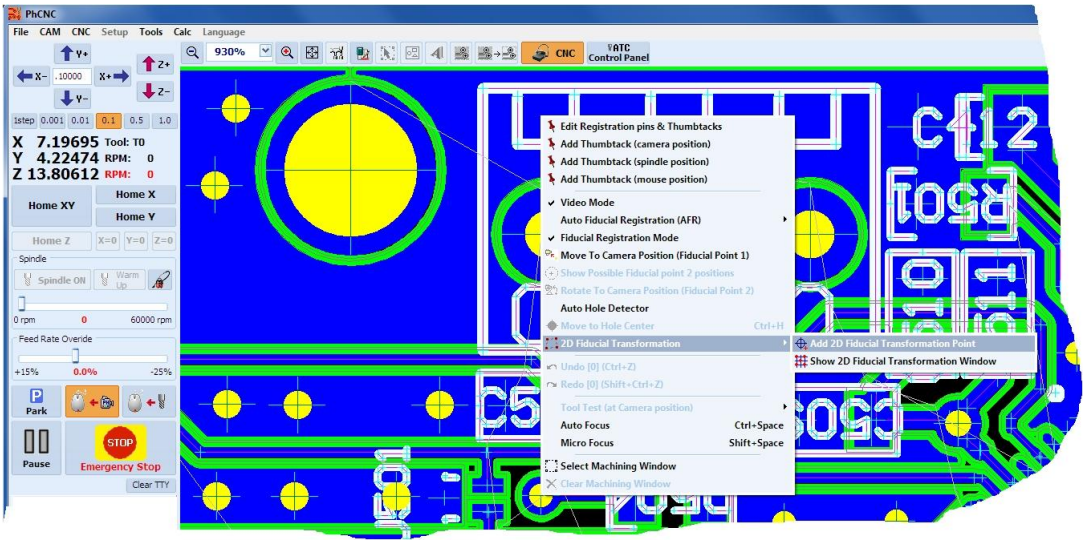
And this way ensuring perfect drilling, cutting, stencils, etc.

How it works:

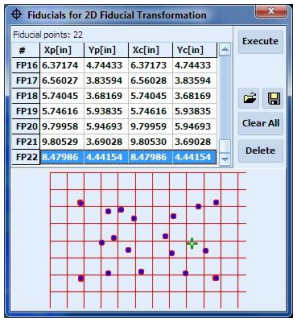
The options for "2D Fiducial Transformation" are available in popup menu in CNC mode shown below.

First you have to do "Manual Fiducial Registration" – later, this will help you to find easily the fiducial points.

As in "Manual Fiducial Registration", you have to:
a) find with the camera some specific point (pad center, corner, ..)
b) show with the mouse the corresponding point in the project and then with the right mouse click - select from the pop-up menu the "Add 2D Fiducial Transformation Point" option (shown below)



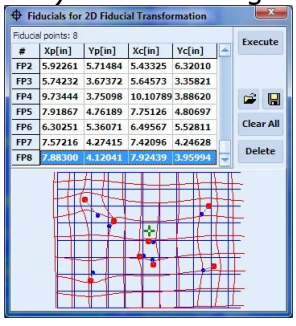
The fiducial points are added to the "2D Fiducial Transformation" window:



#	Xp[in]	Yp[in]	Xc[in]	Yc[in]
FP16	6.37174	4.74433	6.37173	4.74433
FP17	6.56027	3.83594	6.56028	3.83594
FP18	5.74045	3.68169	5.74045	3.68169
FP19	5.74616	5.93835	5.74616	5.93835
FP20	9.79958	5.94693	9.79959	5.94693
FP21	9.80529	3.69028	9.80530	3.69028
FP22	8.47986	4.44154	8.47986	4.44154

The grid below the table shows the locations of the fiducial points. With double mouse click on some row, you will move the camera to the fiducial point position (for example, to check after transformation).

In normal conditions (errors < 20-50 mils), the blue and the read grid must be overlapped. In case of huge mismatches (like the pictures at the beginning), the red grid will show the current 2D transformation (deformed coordinate system) and the blue grid show the project coordinate system.



#	Xp[in]	Yp[in]	Xc[in]	Yc[in]
FP2	5.92261	5.71484	5.43325	6.32010
FP3	5.74232	3.67372	5.64573	3.35821
FP4	9.73444	3.75098	10.10789	3.88620
FP5	7.91867	4.76189	7.75126	4.80697
FP6	6.30251	5.36071	6.49567	5.52811
FP7	7.57216	4.27415	7.42096	4.24628
FP8	7.88300	4.12041	7.92439	3.95994

There is no limit for the fiducial points, the rule is simple: more points - better match !
The minimum number of fiducial points is 4, but at least 8-10 are recommended !

The button "Execute" will do the transformation. The transformation is executed for every layer (Top, Bottom, Mech, Drill, Stencil, Top overlay, ...), so you can easily create stencils or print mask or overlay - which best matches the object on the table!

After the transformation, you can check the registration of the project and if you want you can add additional fiducial points and execute the 2D Fiducial Transformation again (many times!).

Before the "2D Fiducial Transformation", the project will be automatically saved (the name will include the timestamp) and added to the recent projects list (File menu!). The Undo is also available (in the CAM mode only!).

2D Fiducial Transformation video clip:
We recommend that you watch the video, presenting the "2D Fiducial Transformation".
https://youtu.be/3lf7SIG8D20

7. G code commands available in PhCNC

Below are listed all G codes maintained by PhCNC.
Non mandatory and modal parameters are listed in square brackets [].

Important:
The X, Y, Z coordinates are in inch or mm, according to your G20/G21 selection. The feed rate command **Fnnn** is in inch/min or mm/min according to your G20/G21 selection as well.

[Nnnnn]
Line number is a non mandatory item but helps you find the line(s) of G code you want when you want to use features like "Run from the cursor" or "Run to the cursor" (partial execution of G code programs).

G20
Inch Mode

G21
Metric Mode

G90
Absolute coordinate mode

G91
Incremental coordinate mode

G0 [Xd.d] [Yd.d] [Zd.d]
Rapid move (linear interpolation), to the coordinates.

G1 [Xd.d] [Yd.d] [Zd.d] [Fd.d]
Feed move (linear interpolation), to the coordinates with
speed (feed rate) Fd.d

Important:
G commands G1, G2, G3, G81, G82, G83, … require feed rate.
Without this parameter they will be ignored.

G2 [Xd.d] [Yd.d] Id.d Jd.d [Fd.d]
ARC CW
CW circular movement to the (X, Y), center shifted with (I, J) from the current coordinates (before command execution).

G2 [Xd.d] [Yd.d] Rd.d [Fd.d]
ARC CW
CW circular movement to the (X, Y) with entered radius (Rd.d). Starting point current coordinates (before command execution).

G3 Xd.d Yd.d Id.d Jd.d [Fd.d]
ARC CCW
CCW circular movement to the (X, Y), center shifted with (I, J) from the current coordinates (before command execution).

G3 Xd.d Yd.d Rd.d [Fd.d]
ARC CCW
CCW circular movement to the (X, Y) with entered radius (Rd.d). Starting point current coordinates (before command execution).

G4 Pd.d

Dwell seconds
Pause in seconds

G81 Rd.d Zd.d [Fd.d]
Drill Cycle
Z axis movement to the coordinates Zd.d. In the machines equipped with micrometric screw or with linear optical scale the movement stops after the set point is reached. Next the tool is pulled up to the coordinates Rd.d.

G82 Rd.d Zd.d Pd [Fd.d]
Chip-breaking Drilling Canned Cycle
Same as G81, with pause P in the lowest tool position.

G83 Rd.d Zd.d Qd.d [Fd.d]
Chip-breaking Drilling Canned Cycle (Peck drill)
multiple cycle drilling with increments of Q.

Tnn
Tool change (number)
Here **nn** is the number of the tool from the G code program. This number is NOT RELATED to the number of the tool holder that the tool resides in on the ATC models.
The execution of this command depends of the machine model.
In the comments line you may have information for the tool type, penetration depth, tool holder number (in ATC models)

Example for **Tnn** command

...
(Tool 2: Comment {Z-0.0117"/-0.297mm, TH08})
N3 T2 S60000
...

This command will be executed in an ATC machine in the following order. First the spindle will place the existing (in the collet) tool back into the tool holder that the tool was assigned to. If the tool is placed in the spindle manually the spindle will go to "Manual tool change" position and the system will ask the operator to remove it. In the next step the spindle will go to calibration pad in order to check that its collet is empty. In the next step it will pick up the tool from the tool holder TH08 in this example and go back to calibration pad in order to set tool penetration at 0.0117" (11.7 mil)

Parameters THnn and Z must be placed in curled brackets { }. The execution of the command on ATC machines without the parameters above will proceed as a manual tool change.

Sddddd
Set spindle speed (rpm)
For all machines with Alfed Jaeger or SycoTec spindles the limits are 5,000 to 60,000 or 5,000 to 100,000 depending on the model.

M02
Program end

M03
Spindle ON

M05
Spindle OFF

M06
Tool Change
This command is ignored. Our tool change command is Tnn only (see above).

M07
Vacuum ON

M09
Vacuum OFF

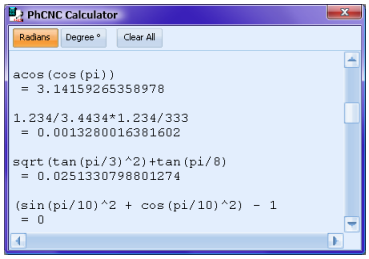
Sample of G code program:
N1 G90
N2 G20
({Z-0.0117"/-0.297mm, TH08})
N3 T0 S60000
N4 G0 X1.4685 Y5.0025 Z0.1500
N5 G1 Z-0.1000 F15.0
N6 X1.4685 Y4.9285
N7 Y4.7289
...
...
N14058 G0 Z0.1500
M09
M05
M02

Important

- First movement command G0, G1, MUST contain X, Y, Z coordinates.
- The First and last movement commands MUST set the Z coordinates at a safe position compared to the materials surface. Usually Z0.150 is enough be sure it clears your work piece and fixture if you are using one.
- The Home Z procedure must be performed initially and every time when the "stack thickness" is changed. This gives the system information where the material surface is in terms of Z coordinates. It is not related to the tool penetration, but is important to the Z position during the rapid traverse movement and the point that the plunge rate is changed from the maximum to the programmed for the current operation.
- In all machine models the G code for Z depth of cut should be set to a depth greater than needed. It will be limited to the desired value by the scale reading or micrometer screw setting or depth limiter on the pressure foot (depending on the technology used in the model). A typical value for printed circuit boards is Z-0.100

8. PhCNC Calculator

The PhCNC Calculator can be invoked by clicking the button. It pops up on the screen in a separate window. All calculations are executed after pressing the Return (Enter) key, and the result is then displayed on the next line. All calculations are made with double precision (16 digits after decimal point). The calculator can compute arithmetical expressions with unlimited complexity (up to 256 chars long).



The following operations and functions can be used:

- Arithmetical operations:
  +, -, *, /, ^ , ( )
- Arithmetic operators:
  ^, *, /, +, -
- Relation operators:
  <=, =<, <, >=, =>, ==, !=
- Bitwise operators:
  &, ~, |
- Logical operators:
  !, ||, &&
- Trigonometric functions:
  SIN(x), COS(x), TAN(x), COT(x), ACOS(x), ASIN(x), ATAN(x), ACOT(x), SINC(x), SEC(x), ASEC(x), CSC(x), ACSC(x)
- Hyperbolic functions:
  COSH(x), SINH(x), TANH(x), COTH(x), ASINH(x), ACOSH(x), ATANH(x), COTH(x), SECH(x), CSCH(x), ACSCH(x)
- Mathematical functions:
  LOG10(x), LOG(x), EXP(x), SQRT(x), INT(x), ABS(x), RND(x), FACT(x), !(x)
- Numeric Constants:
  "pi" = 3.14159265358979
  "e" = 2.71828182845905
  "in" = 25.4
  "mm" = 0.0393700787401575

Calculation Examples:

- 4495*1.18*73.592
  = 390339.3272
- log10(1.345*2.4567/(2.34556+23345.23))
  = -3.84916737941656
- fact(49)/(fact(6)*fact(49-6))

  = 13983816
- (FACT(90)/(FACT(15)*FACT(90-15)))/(FACT(37)/(FACT(15)*FACT(37-15)))

  = 4890505.87751033
- 12*sin(60)
  = 10.3923048454133
- 10*in
  = 254
- 25.4*mm
  = 1
- (20 > 10)
  = 1

APPENDIX-A: How to calibrate PhCNC video camera (VC)

If your machine is ordered with VC, it is factory calibrated. Some heavy shipping conditions may create the need of fine recalibration, but in most cases you may skip this procedure.

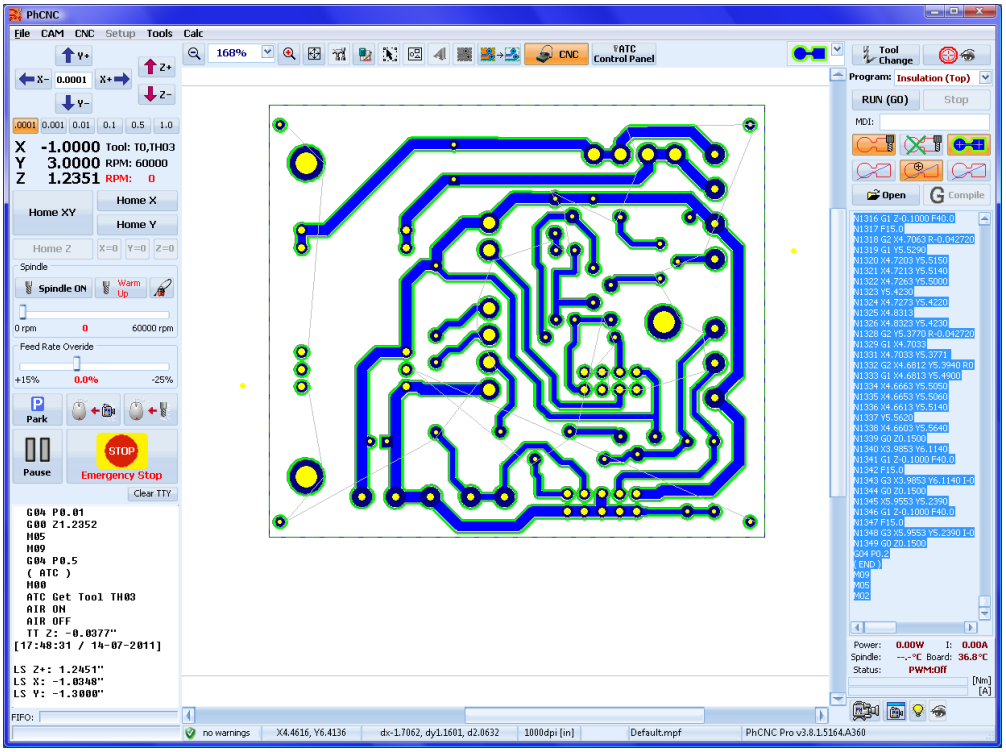
If you purchased the camera separately you will need to install it yourself. This will require you to perform the following steps for calibration listed below. This will also need to be done after any camera or spindle removal/replacement.

- Camera axes to machine axes zero angle calibration
- Camera magnification calibration
- Camera to spindle X offset calibration
- Camera to spindle Y offset calibration

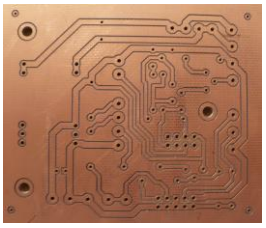
Before attempting any of the steps listed above the VC must be at focus distance. Once the Z axis is homed you may use the "Auto focus" button of "PhCNC Video".

The best and easiest way to calibrate the VC is to make a simple single sided test board. It may contain just two traces, one straight vertical and one straight horizontal.

For our example we will use a simple single sided PCB shown below:



Shown bellow is a picture of the finished board used in this case. It is a single sided PCB with 15.6 mil Insulation traces:



Important:
After the traces of insulation are made this board represents the actual machine axes, and it should not be removed from the machine table until all steps of VC calibration are completed!

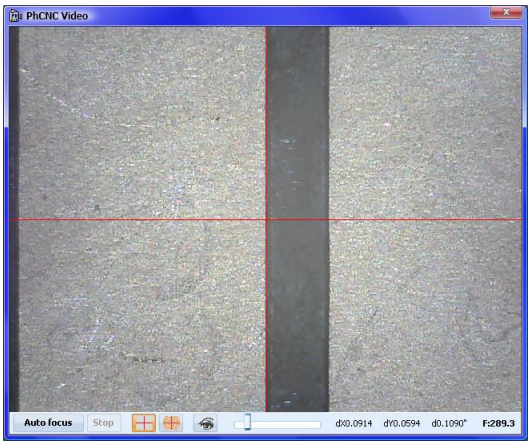
A.1 VC angle calibration

To perform an angle calibration you need to loosen the screw on left side of VC body:

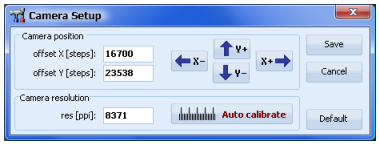
Shown below is a trace that should be perpendicular but is not:

After you loosen the screw you have to carefully turn the VC watching the video image. You may need to repeat the process because when you retighten the screw it may change the angle a bit.

Of course you can use either a horizontal or vertical trace for calibration. The goal is to align the machine axis to camera image. Shown below is a proper angle calibration:



The next 3 steps of calibration are using Camera setup menu (CNC/Camera/Setup)



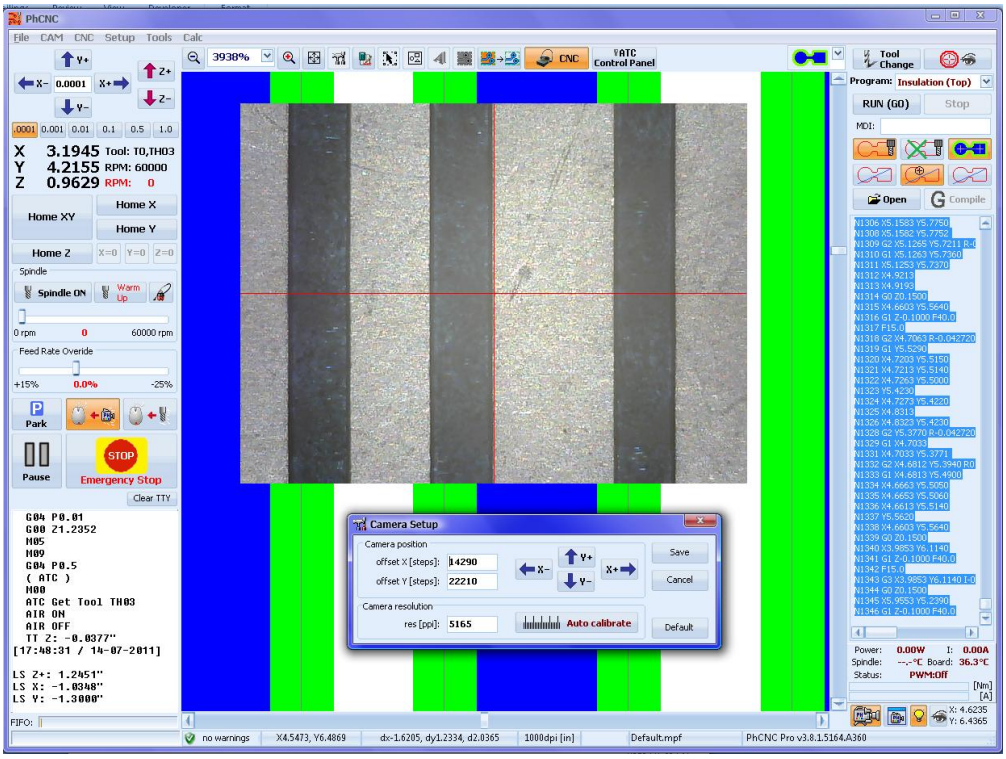
A.2 VC magnification calibration.

It is advisable to perform the magnification calibration next. The VC resolution is related to its magnification and it is measured in Pixels per Inch [PPI]. Magnification by itself is closely related to the distance between camera optics and board surface. This means the VC should be at focus distance during this calibration. The best way to perform this calibration is to click on the Auto calibrate button, it is a fully automatic process. If your automatic calibration fails you may do it manually in "CNC Video" mode. The calibration is correct when the image matches the size of your traces and tool paths in your design. In order to get best results use a stub end mill to make the insulation traces. It has a constant width independent of the tool penetration. DO NOT forget to declare the right insulation tool diameter in CAM mode.

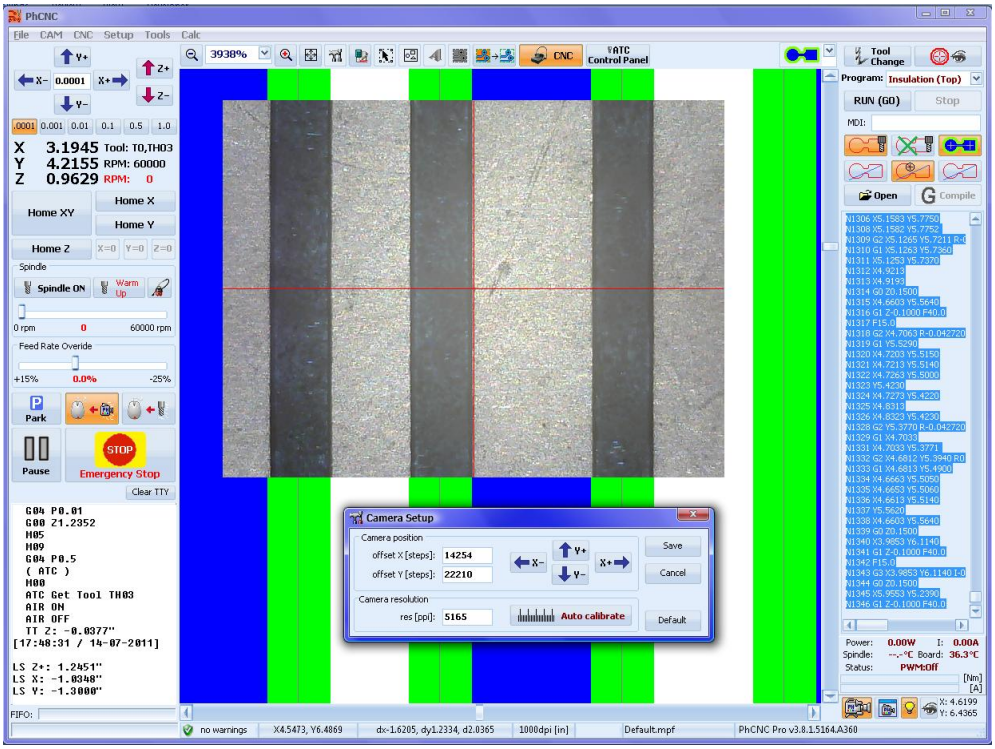
A.3 X offset calibration

Offset calibrations of both X and Y axes must be performed in the "CNC video" window. "PhCNC Video" window is not needed and should be closed.

To correct the X offset you will need to position the VC (using chase mouse camera mode) over a trace parallel to Y axis, as shown below.

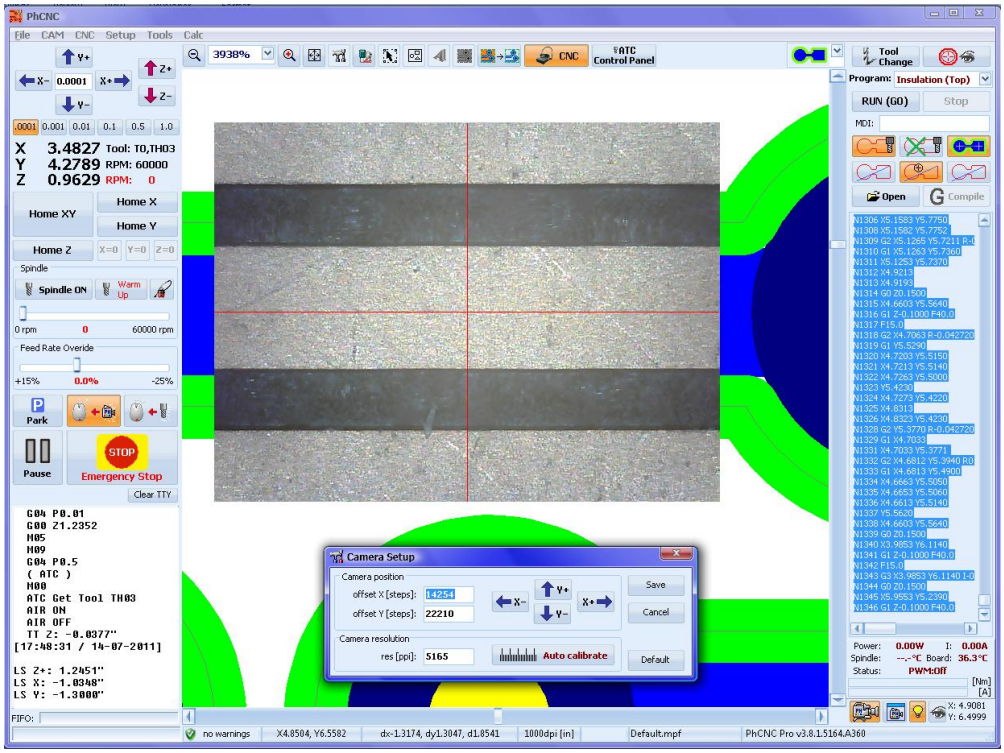


You will see the actual traces are shifted compare to your project data. Using the CNC/Camera/Setup and moving the image by X arrows you can calibrate it (or if the offset is too big change the figures in the offset boxes for coarse alignment). In the example above using the "X+" arrow button from "Camera Setup" menu would correct the camera position after moving the VC 3.4 mils (34 steps). Save the current offset before you move to the Y offset.

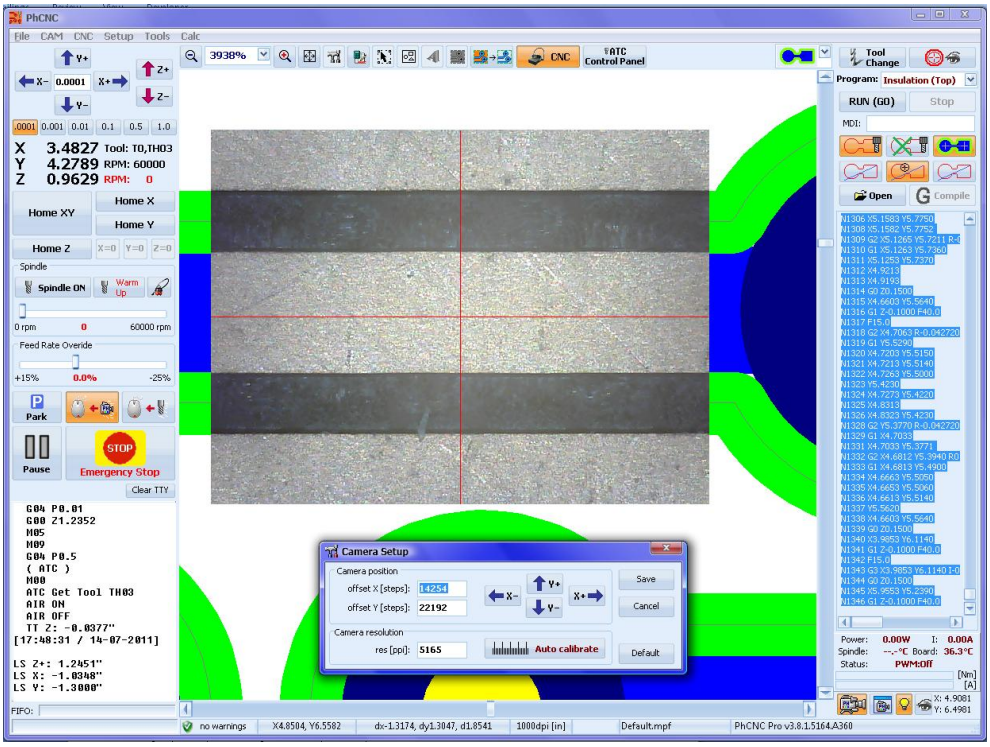


A.4 Y offset calibration

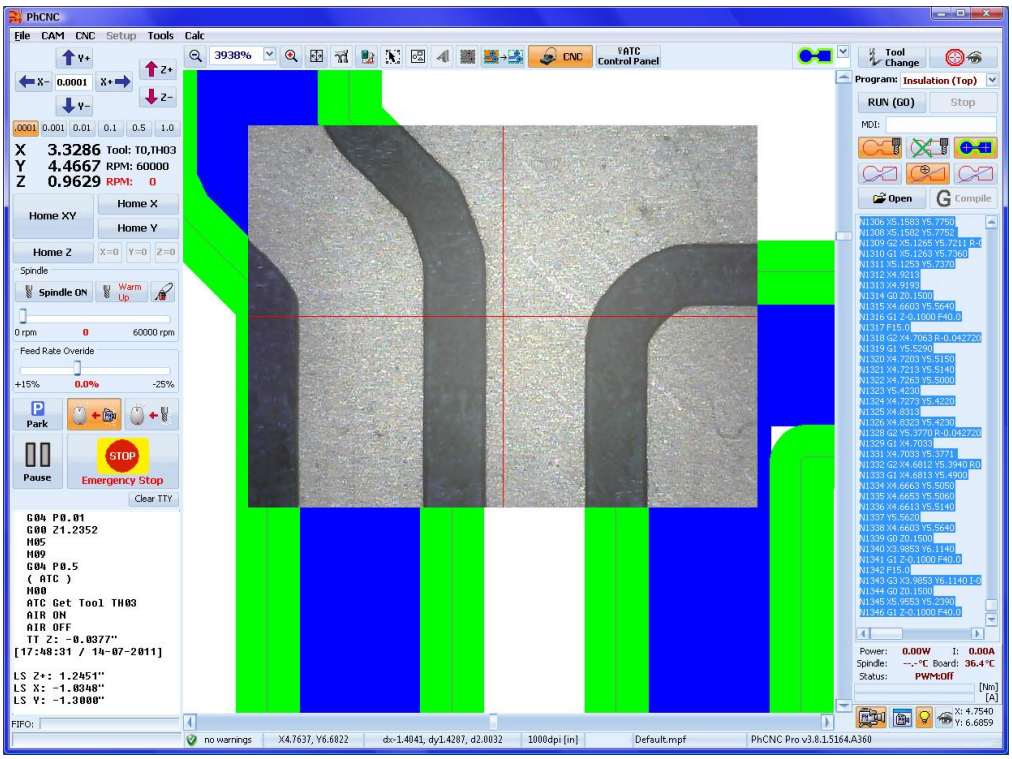
To calibrate the Y offset you need to move the VC over a trace parallel to X axis. Shown below is the current camera position.



After 3.6 mils (36 steps) correction using the "Y-" arrow button, Y calibration is completed. You may use the Y offset window to make coarse corrections and the arrows for fine ones.

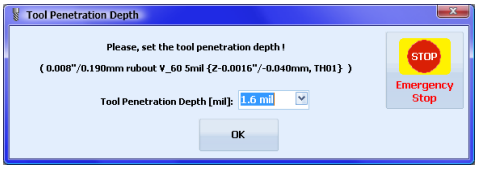


Again, save your settings before any other operation. Once the calibration is completed you may use the VC to observe any spot of your project. Using "Case Mouse (camera mode)" make a right click at the desired spot (first zoom out to find it and then zoom in to select it accurately).

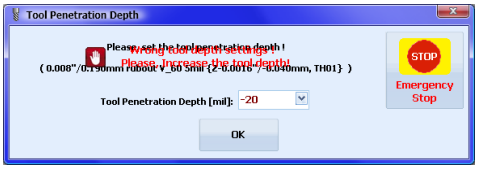


APPENDIX-D Tool Penetration Depth

All systems have the tool penetration depth controlled by the PhCNC control software. No manual action is required by the operator. The tool tip position compared to material surface is measured with an accuracy of 0.00004 inches (0.001 mm) and is maintained during the programs execution with an accuracy of 0.0001 inches (0.0025 mm).
Tool penetration can be set manually by entering the value in the corresponding line (if "Manual depth control mode" is active, described in 4.4.13).
If you use the "Tool Change" button the PhCNC will always ask you to confirm/change the depth setting.

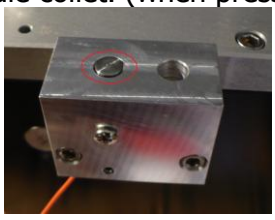


If you enter a negative value you will have a red blinking warning message but will be allowed to continue. Negative entries mean that the tool will be above the material surface. This can be used for training and maintenance purposes. The range of executable values is +375 mil (9.5 mm) to -50 mil (1.27mm)



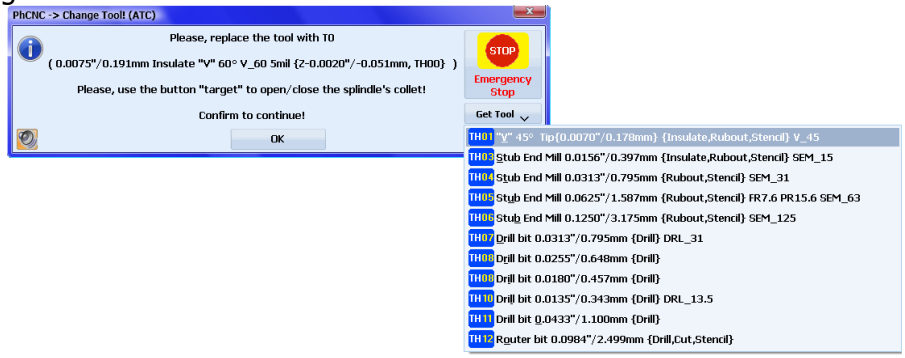
D.1 Tool change operation in ATC machines

For the CNC models equipped with direct ATC spindles, the tool changes and tool calibrations are completely automatic. All prearrangements of the tools should be done in the "ATC Control Panel" (described in the manual). Manual tool changes are also available in these models. The button on the calibration pad serves to control the spindle collet. (when pressed the collet opens)



Manual tool changer

As we mentioned in the main part of this manual our ATC machines have the capability to combine automatic tool change with manual tool change. If operator fails to define a tool in the magazine they will be asked to insert that tool during the machining process with the manual tool change request window. In this window the operator needs only to click on the Get Tool button and select the undefined tool from the magazine and click ok to continue.



D.2 Tool change operation in MTC machines (Manual tool change)

The tool change procedure is manual, but the tool penetration calibration and control is fully automatic and software controlled.

At the tool change position, push the knob (on top of the spindle) down until it clicks. Turn the knob CCW until the tool is released. Place the new tool in the collet using the "manual tool changer". Apply a small upward force and turn the spindle knob CW, gently tightening it. Pull the knob up until it clicks. The tool change is completed.

Manual tool changer

APPENDIX-E: Front panel LED indicators

There are two LEDs on the front panels, green and red.

- The red LED means a limit switch is activated when lit (including the contact pad).

- The green LED is blinking constantly once you have the powered up, and the blinking frequency shows the status of USB connection.

    - 0.5 Hz (2 second flashes) means there is no USB connection.
    - 2.0 Hz (flashes 2 times per second) means there is a USB connection with the PC, and the machine is recognized as a device, but PhCNC software is not running
    - 8.0 Hz (flashes 8 times per second) means the connection is good, and the machine is under PhCNC software control.

Note:
During the initial power up of the machine, both LEDs are on for ¼ of a second (250ms).

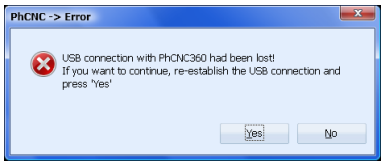
APPENDIX-F: PhCNC controllers

F.1 PhCNC360 USB controller (2008-2015)

The (A36x/A56x/A4xx/A6xx) use a USB 2.0 controller (PhCNC360) which is 10 – 15 times more productive than the original one used in the A350. Due to significant hardware differences PhCNC comes in two versions. PhCNC for A350 called (**PhCNC A350**), and PhCNC for (A36x/A56x/A4xx/A6xx) called (**PhCNC A360**). Newer revisions of PhCNC are coming for both versions. (Some of the future revisions are based on the new firmware of (A36x/A56x/A4xx/A6xx) and we will be unable to implement them in A350 machines)

Here are some of the new features of the (**PhCNC A360**) Hardware/Firmware.

If the controller had lost its connection with the software (let say you mistakenly disconnect the cable). It will go to "forced" PAUSE mode (This means it stops the execution of the program and spindle).
In this case you will see the window below. If you are able to restore the connection and answer the prompt with "YES" The machine will continue from the stop point without any loses.

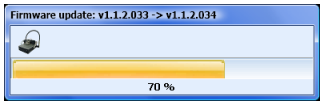


Notes:

- This option only works if power to the machine was not lost.
- (**PhCNC A360**) Also works with USB 1.1 ports, but productivity drops about 25%

F.2 Firmware updates

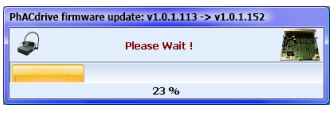
PhCNC control software reorganizes and updates, if needed, the CNC controller's firmware. During the update you will see the window shown below.



The process takes ~1.5 seconds and must not be interrupted. DO NOT turn the power off or disconnect the usb cable during this process. Replacing your current set of software/firmware with older revisions is impossible. In such a case PhCNC will prompt you to acquire a new revision. At the end of this process the controller restarts itself automatically.

F.3 PhACdrive firmware

The PhACdrive is a sensorless spindle drive utilizing our vector control technology (AC/PMSM/BLDC motor spindles). This drive offers a maximum energy efficiency and full protection (for the motor and itself) in the entire RPM range of (5,000 to 120,000 RPM) in all modes of operation.
Its firmware is automatically updatable, similar to the one used in the CNC controller (PhCNC360).

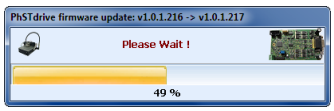


Important:
The process takes about 12 seconds and must not be interrupted. DO NOT turn the power off or disconnect the usb cable during this process. Interrupting the process above may lead to the destruction of your controller. When the firmware of the drive and the CNC controller needs updated, PhCNC will update first the CNC controller and then the drive.

F.4 PhSTdrive firmware

PhSTdrive is intelligent stepper motor drives based on DSP microcontrollers. It have the ability to use dual servo feedback from linear & rotary encoders and to store calibration data for screw mapping. Special real time processing allows them to drive the motors with extreme accuracy and smooth movements.

The firmware is automatically updatable (all system drives at the same time), similar to the one used in the CNC controller (PhCNC360/PhCNC440) and the spindle drive (PhACdrive).

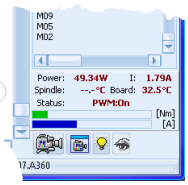


Notes:

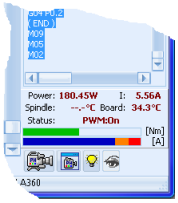
1. Due to the fact that we update the firmware of all (3) drives at the same time, the process is slightly longer (about 45 sec).
2. After the firmware update, the mashine must be homed. If the process of the firmware update is interupted (definitely NOT RECOMMENDED) it will be repeated in the next connection between the machine and PhCNC software.

F.5 Additional status information for the machines equipped with PhACdrive

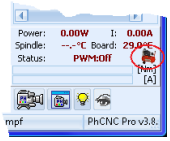
Additional status information for the spindle and drive motor is offered on the CNC mode screen for the models equipped with PhACdrive, as well as information on the air pressure for the ATC models.



Shown above is the spindle load during the cutting of 1/16 FR4 board material with a 2.5 mm router at 60000rpm.
Displayed in the lower left corner are: motor power consumption, line current (RMS), spindle temperature (optional), drive board temperature, drive board outputs status (PWM: On/Off). Graphic bars also show dynamic motor load (current [A] & torque [Nm]). During an overload a portion of the bar goes red. During a continuous overload (up to 6 s / 12 s) the overload protection will be activated to prevent the motor from overheating.



Low air pressure indicator (ATC models only)

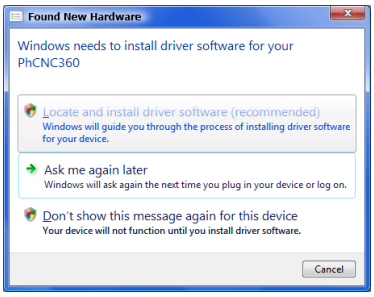


APPENDIX-G: USB drivers installation

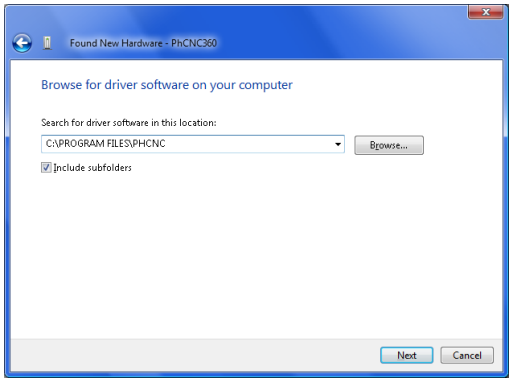
Drivers are compatible with Windows XP, Windows 2003, Vista/Win7/Win8/Win10 (32bit and 64bit)

During the first connection of the machine or dongle (PhCNCLSSE) to a USB port of your computer, Windows will ask you for USB drivers for this device. If you have the original CD-ROM with PhCNC software, you have to install them from the CD. Otherwise you have to point to the folder where the PhCNC is already installed (for example "C:\Program Files\PhCNC").

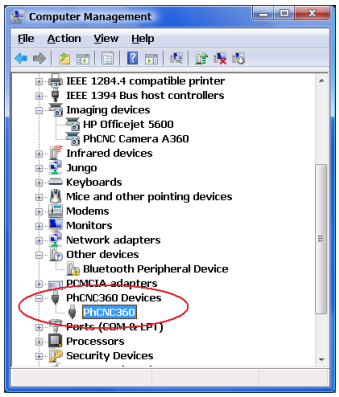
Installation process is similar to any other USB device.



If there is no CD, enter the path to the folder with PhCNC.



After a successful driver installation, the controller (PhCNC360) is listed in the "Device Manager" window:



APPENDIX-I: PhCNC control programs installation

PhCNC installations are identical, but they work only with the model that they are intended to. **PhCNC**

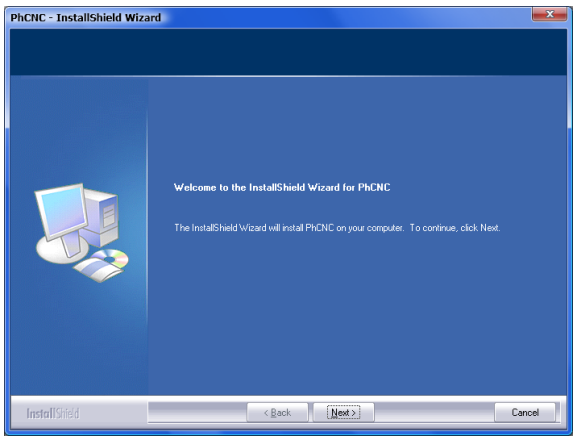
PhCNC64 is 64bit version of **PhCNC** for **x64** Windows (Vista/Win7/8/8.1) and machine models A36x/A56x/A4xx/A6xx.
PhCNC64 is completely visually and functionally identical to **PhCNC** (32bit), but allows addressing of more than 4GB RAM (an advantage when large and complicated projects are handled).
PhCNC64 installation is completely identical with **PhCNC** installation.
PhCNC64 and **PhCNC**, can be installed on the same computer but only one can be used at a time.

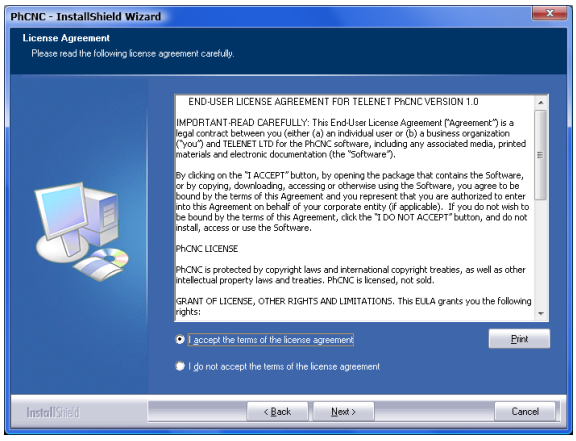
PhCNC installation from CD–ROM, start **Setup.exe**, **PhCNC_A360_Setup.exe** or **PhCNC64_A360_Setup.exe** (A36x/A56x/A4xx/A6xx only)

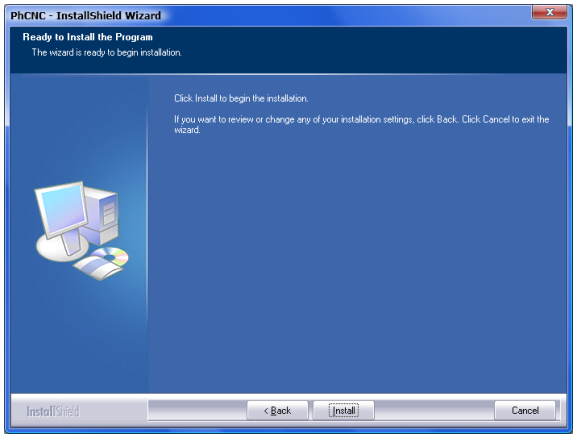
In case of download, start **PhCNC_A360_Setup.exe**, **PhCNC64_A360_Setup.exe** or **PhCNC_A350_Setup.exe** (A350 only!)

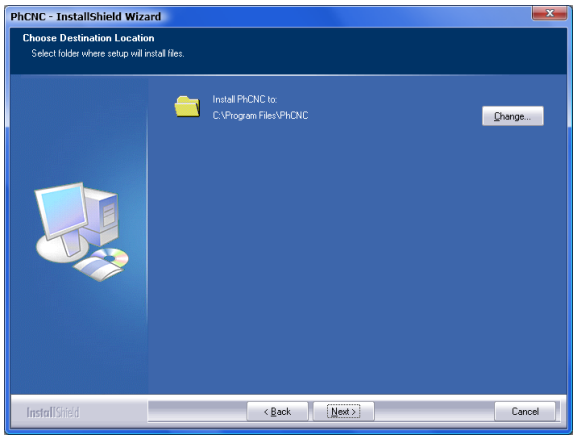
I.1 First time PhCNC installation

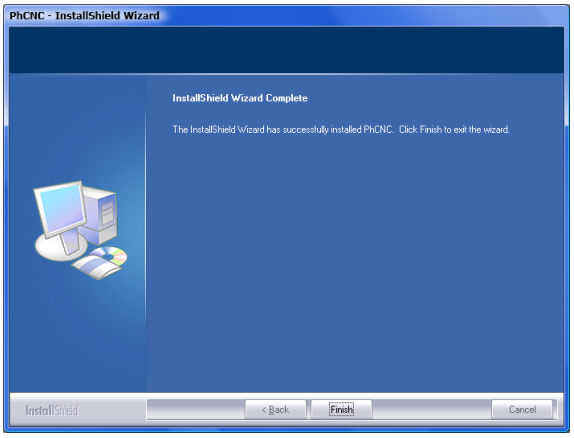
All steps of the first time installation are shown below.



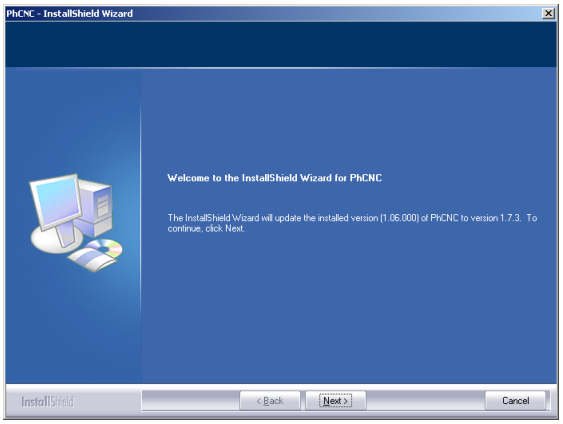


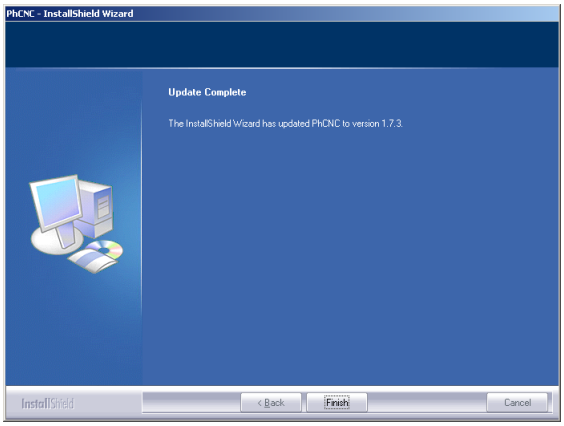






I.2 Upgrade of an existing installation





PhCNC creates a "PhCNC" folder in "Program files" folder, also a short-cut on desktop:

USB drives are located in the PhCNC installation folder (by "default" "C:\Program Files\PhCNC").

All upgrade installations preserve current settings of PhCNC.

The PhCNC uninstall procedure is the same as other software programs, "Control Panel"->"Add or Remove Programs" (Windows 2000/XP) or "Control Panel"->"Programs and Features" (Windows Vista/Win7/Win8/Win10).

PhCNC

For more information:

Accurate CNC Inc.

Email to: support@accuratecnc.com

Web: www.accuratecnc.com