

PCB Prototyping Software

IsoCAM3000



Version v5.74.4

User's Manual






Table of Contents

1. IsoCAM3000 Basics	4
1.1 First installation of IsoCAM3000.....	4
1.2 Upgrade of an existing IsoCAM3000 installation	5
1.3 Machine parameters' setup	6
2. CAM mode.....	6
2.1 Gerber/Drill Import	7
2.1.1 Window for Gerber/Drill files import.....	7
2.1.2 Gerber/Drill files import	8
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.....	19
2.1.7 Convert Gerber to Drill	22
2.1.8 Info layers import.....	23
2.1.9 Gerber editor	24
2.1.10 Single layer PCB import	25
2.2 IsoCAM3000 Project Files	27
2.3 Buttons and fields used in CAM mode.....	27
2.4 Tools	32
2.4.1 "Insulate" Tools	33
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.....	52
2.10 Delete objects	53
2.11 Rubout – excessive copper removal.....	53
2.12 Editor for the mechanical layer	56
2.12.1 Contour cutting selections	56
2.12.2 Contours selection in the mechanical layer.....	57
2.12.3 Contours generation with "IsoCAM3000 Printer"	57
2.12.4 Generation of contours with  ("Insert Text").....	60
2.13  Design Rules Violation (DRV) detector	60
2.14 Registration Pins and Thumbtacks.....	62



2.14.1 Thumbtacks	63
2.14.2 Registration Pins.....	63
2.15 Insulating copper rubout.	65
2.16 CAM350 import.....	68
2.17 DXF import	68
2.17.1 Selecting DXF layers for “Cut” operation.....	70
2.17.2 Selecting DXF layers for Drill & Countersink	70
2.17.3 Selecting DXF layers for milling (MILL).....	71
2.18 Holes and pads insertion.....	71
2.18.1 Holes insertion	71
2.18.2 Through-hole pads insertion.....	72
2.18.3 Insert Target	72
2.19 Vector Text insertion function.....	73
2.20 Breakout Tabs.....	75
2.20.1 Breakout tabs insertion.....	75
2.20.2 Breakout tabs removal.....	77
2.20.3 Breakout tab width setting	77
2.20.4 Breakout tabs complete removal.	77
2.21 Units of measurement [mm]/[in]	77
2.22 UNDO/REDO	78
2.23 Forced Insulation	79
2.24 Remove spikes (IPD)	80
2.25 Measure	83
2.26 G-Code / HPGL Export from IsoCAM3000	84
2.27 Gerber Export	86
2.28 Print.....	88
2.29 Panelization.....	89
2.30 Marking Drill	92
2.31 Holes Selection / Modifications / Removal	93
2.32 QR Code	95
2.33 DataMatrix.....	96
2.34 Pick & Place Helper	97
3. Setup 	100
3.1 CNC process control settings.....	100
3.2 G-Code post processor adjustments and selections	100
3.3 Display settings	101
3.4 Multithreading & IsoCAM300064	102
4. In case of further questions	103



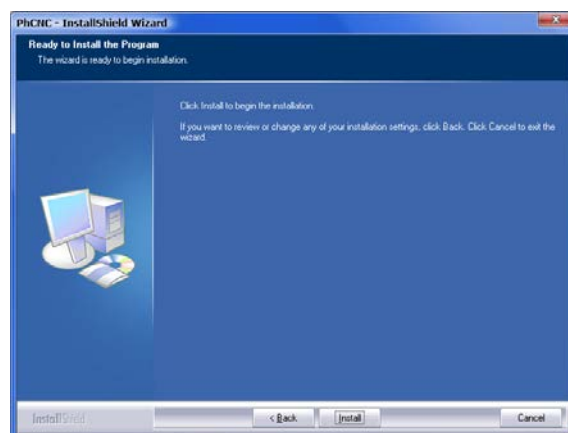
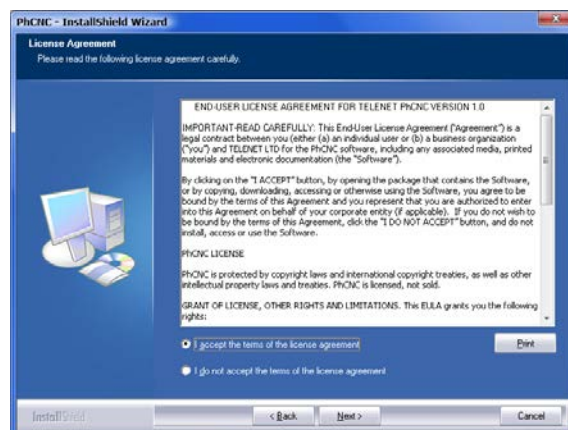
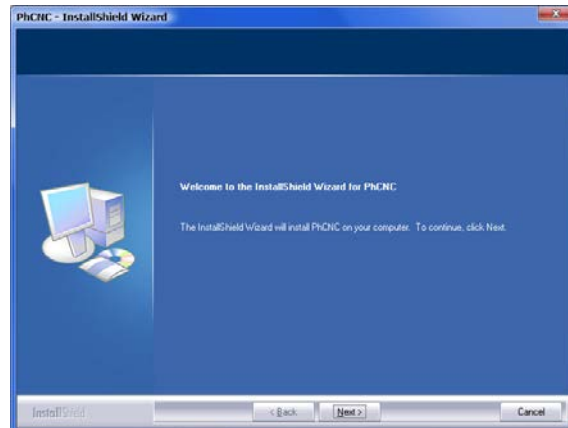
1. IsoCAM3000 Basics

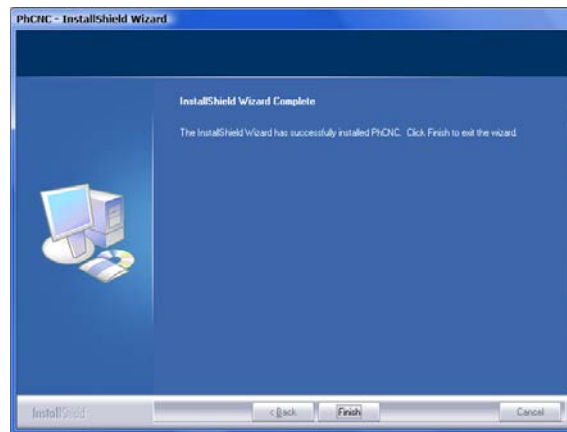
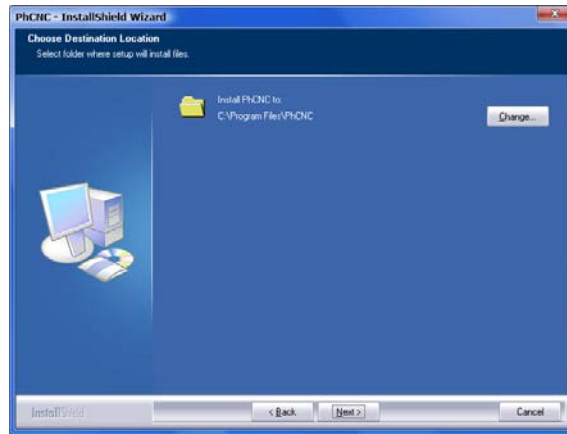
This manual aims to present how to use the different features of IsoCAM3000.

1.1 First installation of IsoCAM3000

Important remark: Before input of the software dongle in the computer's USB port, please, install the software first. If you have input the dongle before installing the software or if, for some reason, the dongle's drivers are not installed, read and follow the instructions in 1.3 USB drivers installation.

All steps of the first-time installation are shown below.



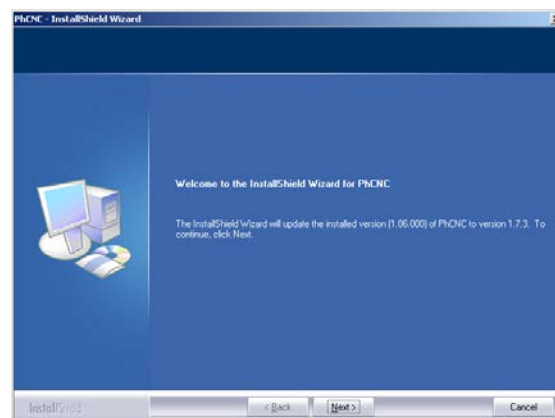


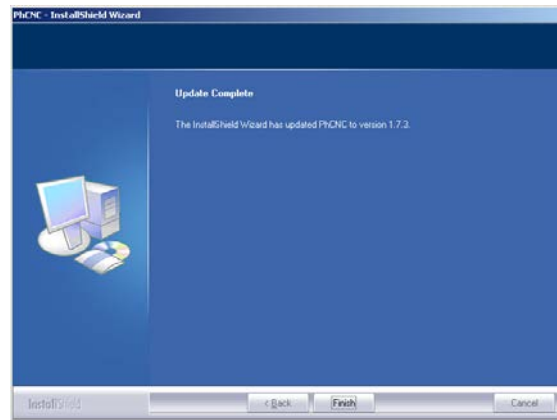
IsoCAM3000 creates a “IsoCAM3000” folder in “Program files” folder, also a short-cut on desktop:



USB drives are located in the IsoCAM3000 installation folder (by “default” “C:\Program Files\IsoCAM3000”).

1.2 Upgrade of an existing IsoCAM3000 installation



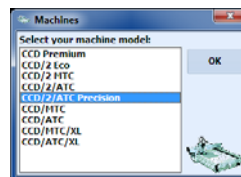


All upgrade installations preserve current settings of IsoCAM3000.

The IsoCAM3000 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).

1.3 Machine parameters' setup

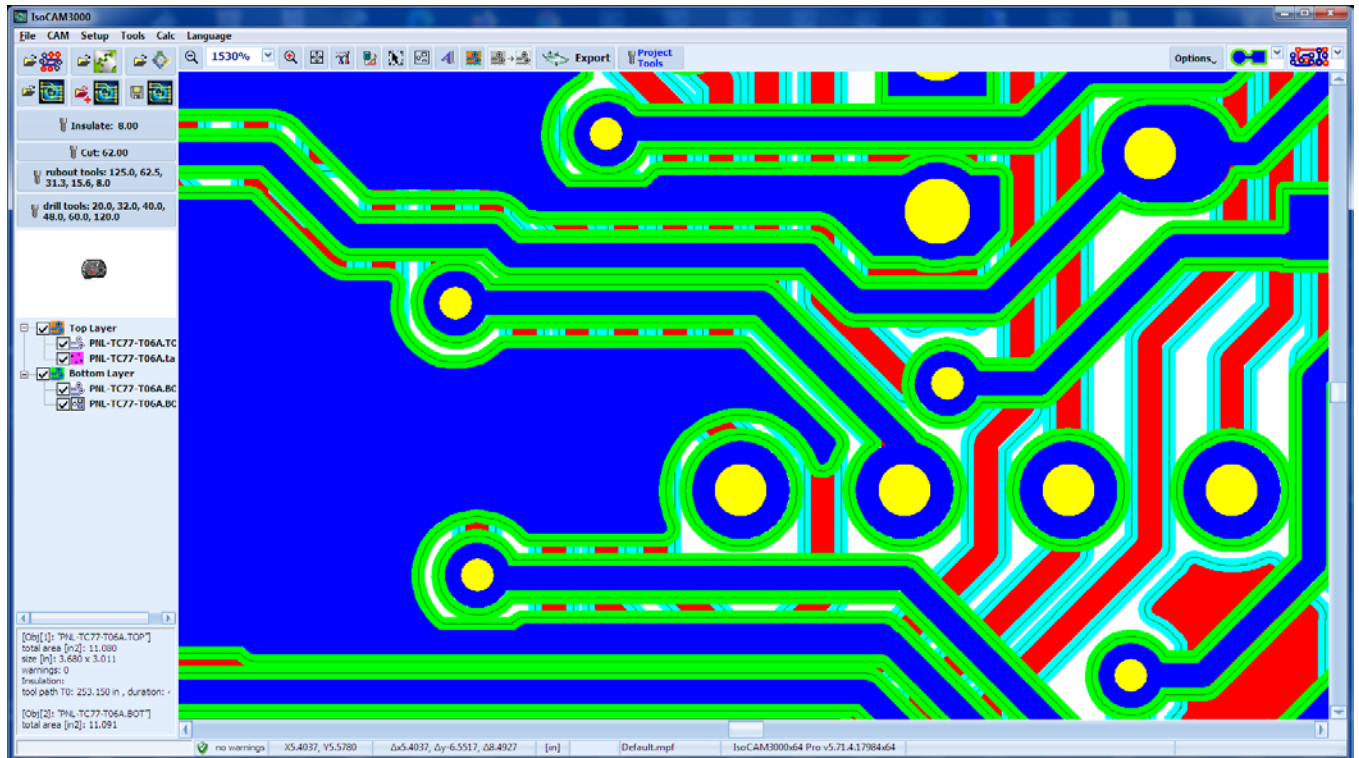
Before starting to use IsoCAM3000, it is highly recommended to input the dimensions of the working area of your machine. This can be done from the "Machines" window:



The above parameters are important mainly for the procedure of manufacturing of double-layer PCBs.

2. CAM mode


When you start IsoCAM3000 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:

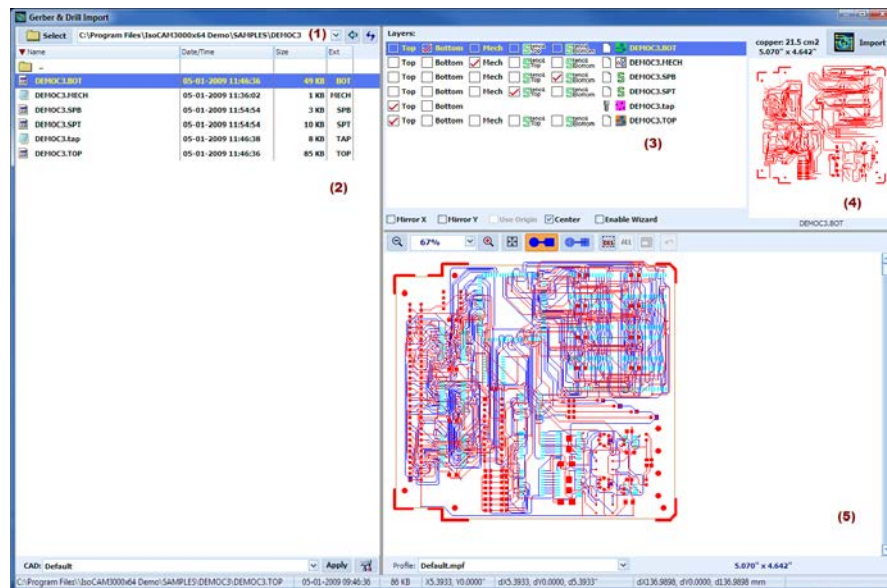
- Load one or more PCB designs
- Group more than one PCB in a single object
- Ungroup already grouped objects
- Clone (copy) objects or groups of objects
- Delete objects or groups of objects
- Select and deselect objects and groups
- Rotate selected objects or groups by an arbitrary angle
- Move selected 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 current revisions of IsoCAM3000 utilizes a visual approach to the process of Gerber/Drill files import. It is based on GerberView technology of IsoCAM3000 (button ).

2.1.1 Window for Gerber/Drill files import

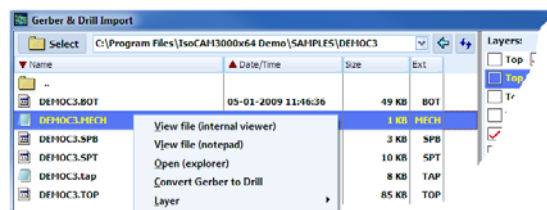
Shown below is the basic view of the Gerber/Drill file import window. Note the numbers in the red brackets. The meaning of each one is explained below.



- (1) – Currently selected folder
- (2) – List of current folder contents. It can be sorted out by any column in ascending or descending order (either numeric or alphabetical)
- (3) – List of files selected for import to IsoCAM3000 and the layer assigned to them.
- (4) – The graphics of currently selected Gerber/Drill file from the list (3)
- (5) – “Gerber View” window of all selected Gerber/Drill files from the list (3)

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

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



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

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

2.1.2 Gerber/Drill files import

To select a file simply click on it in the list (2). IsoCAM3000 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 IsoCAM3000.

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 IsoCAM3000.

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

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

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

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

Note:

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

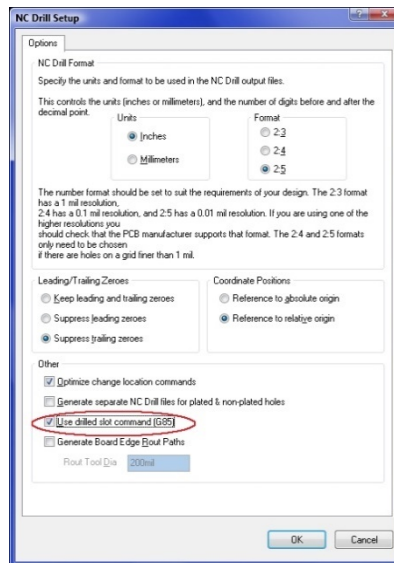


It is possible to import more than one drill/drill-slot files (up to 8) and up to 8 mechanical layers.

Note:

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

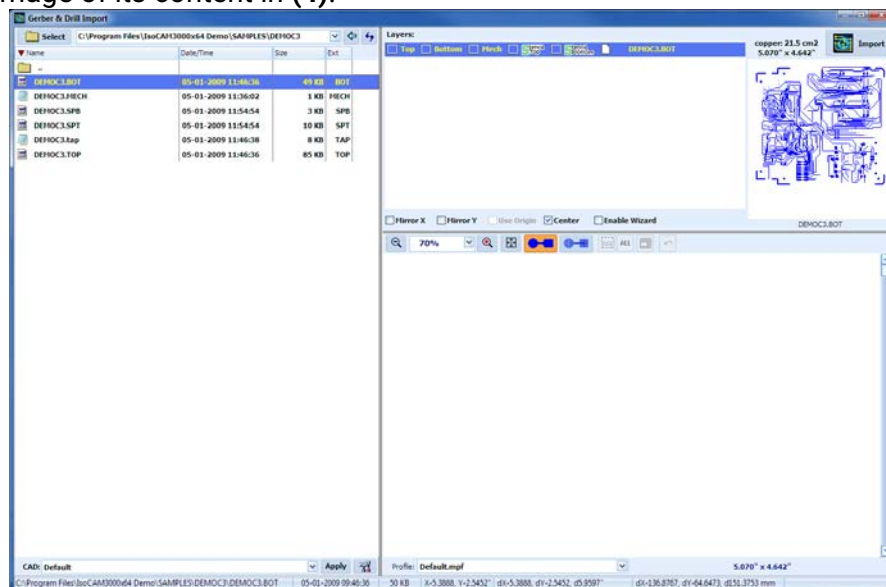
Below is an example of the export of drill-slot files from Altium Designer CAD system.

**Important:**

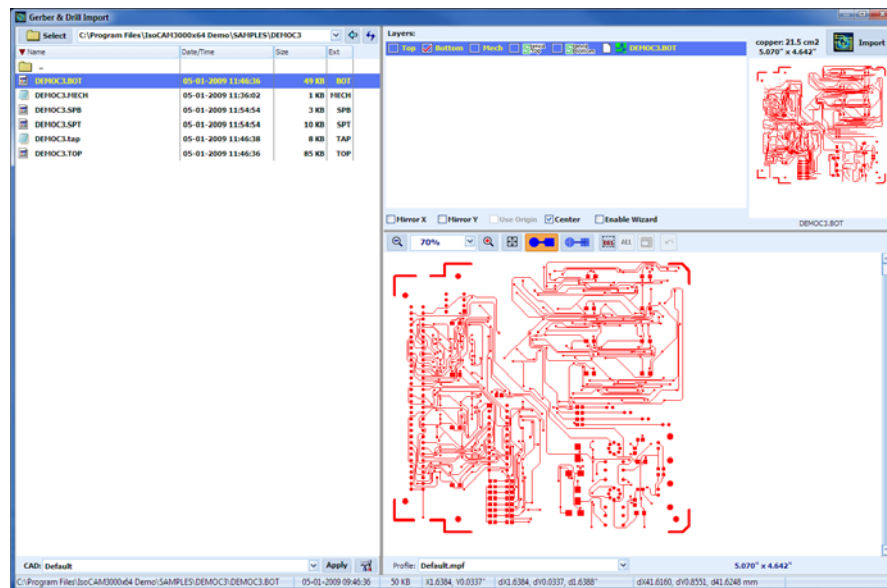
A router bit or drill bit designed for drill-slot operations will serve you better than traditional drill bits.

Here is an example of importing files:

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

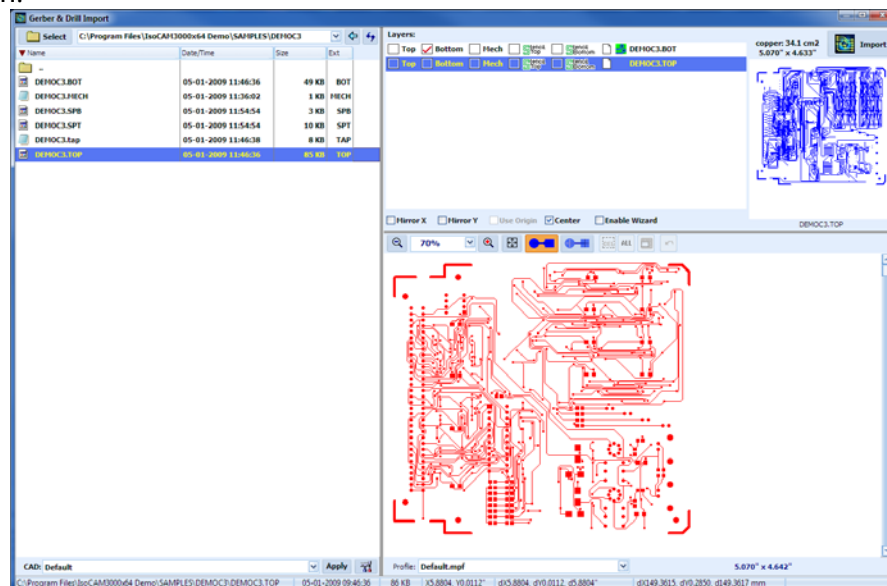


We will select "Bottom" as the status of this file.

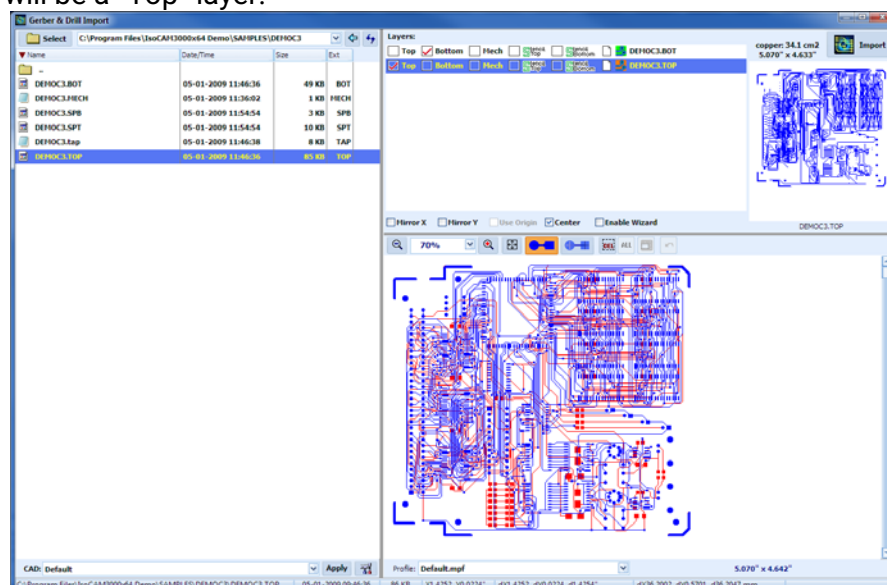


It will appear in the “Gerber View” window (5) with the color for bottom layer.

Next file selection:

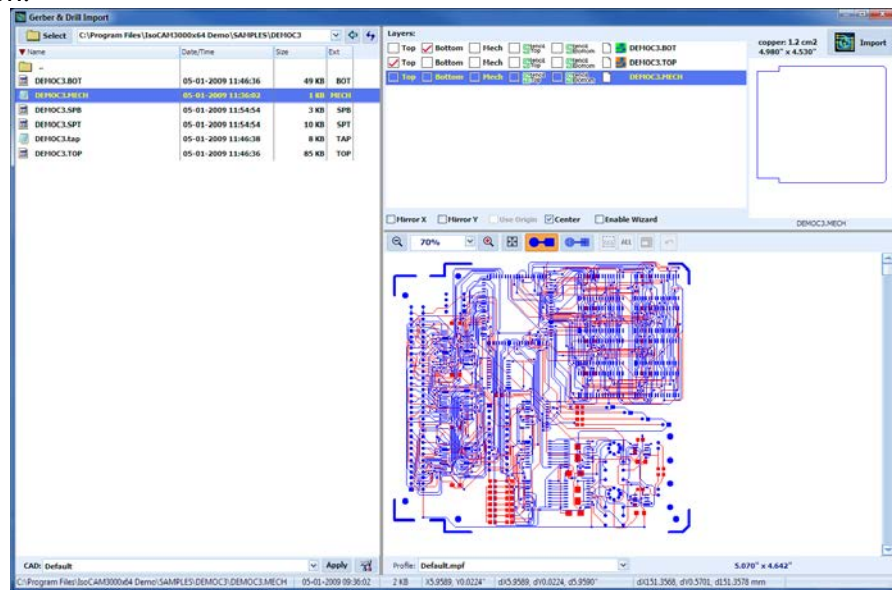


The selected file will be a “Top” layer:

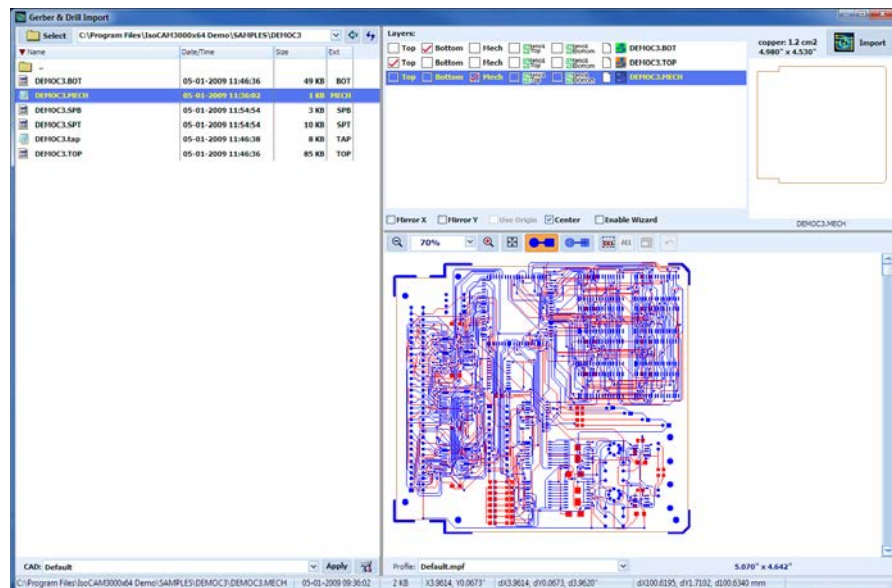





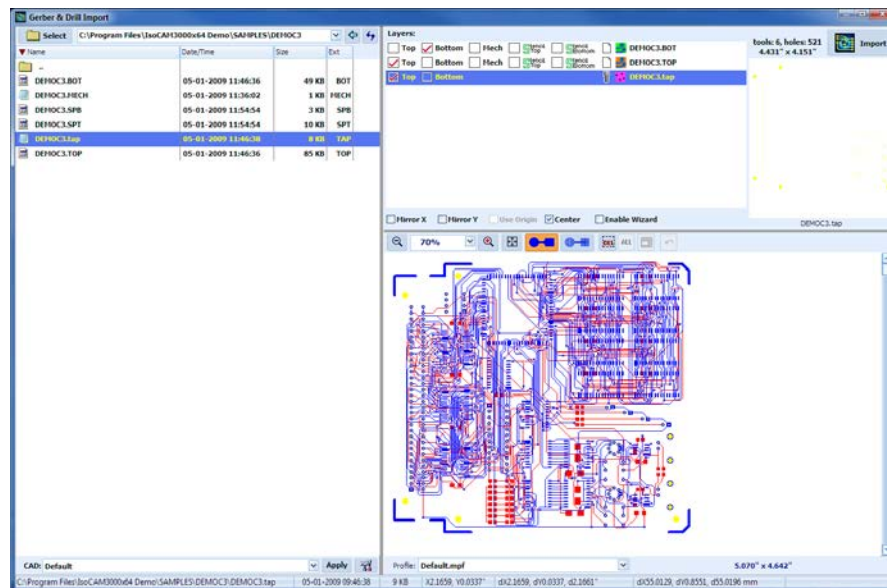
Third file selection:



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



The next selected file is recognized by IsoCAM3000 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, IsoCAM3000 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 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)

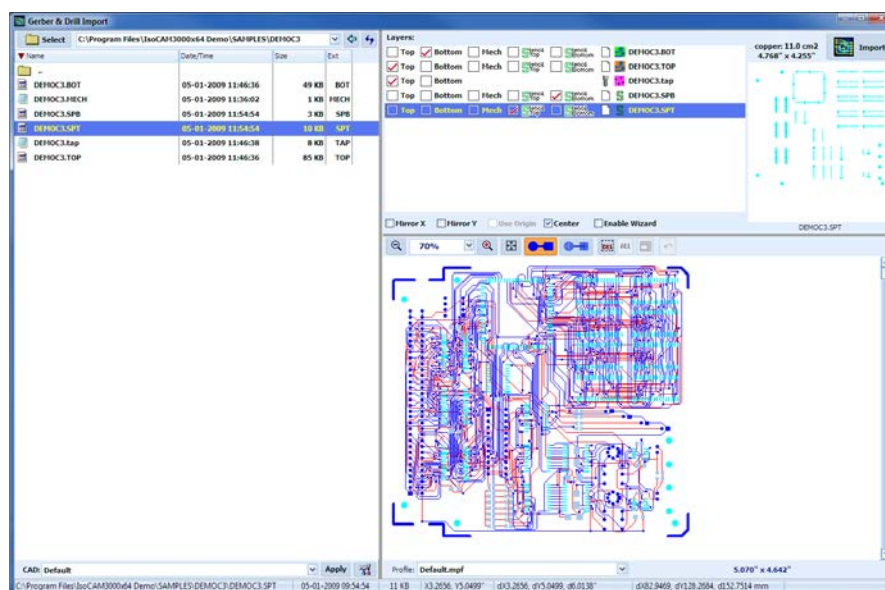
- Now you can select your drill file(s) as a separate object
- Make all transformations needed and align your drill files to the Gerbers
- Finally use high zoom for precise alignment (you will be able to match to 0.0001")

Important:

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

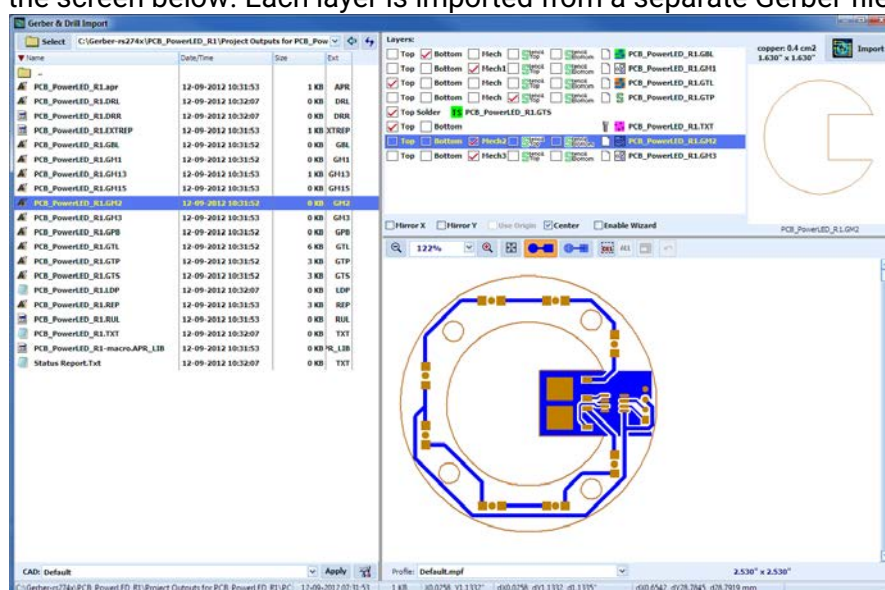
If for some reason the file format is not correct (like inch/metric, fractional digits, leading/trailing zeros suppression etc.) it will be very obvious. To correct the problem, use double click at file name and change the necessary settings.

In the example below all files selected are ready for import into IsoCAM3000.



Note:

IsoCAM3000 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.

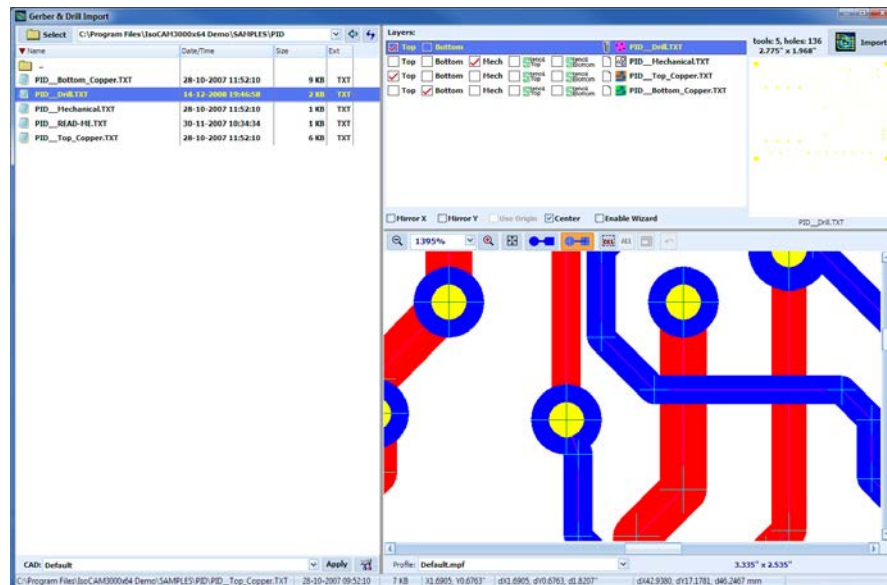


After the import of more than one mechanical layer, IsoCAM3000 generates a single file that contains all imported information. This file is named "Mechanical.GM"



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:

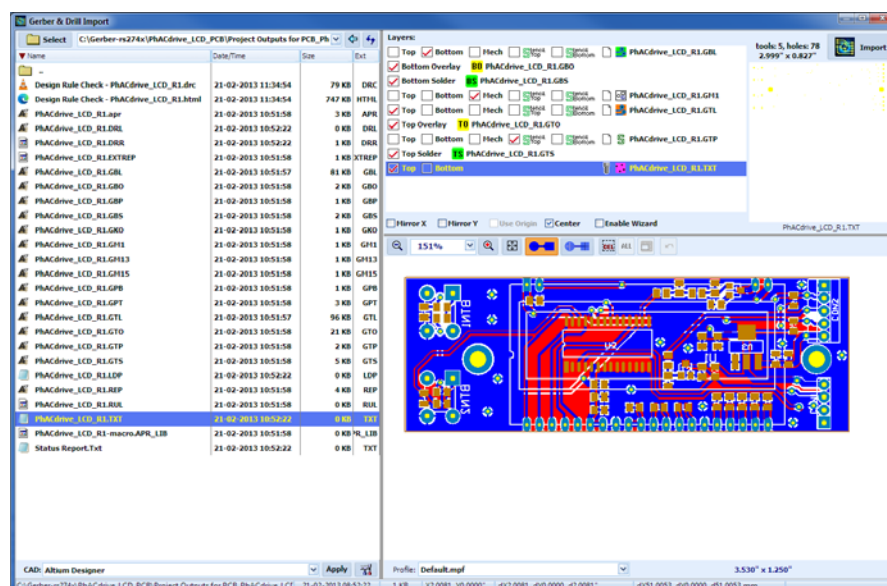
(2.1.3) <https://youtu.be/9BFilfKbBhs>

2.1.4 Profiles for automated Gerber & Excellon import

IsoCAM3000 has ability to create profiles for automated import to the generated from the PCB CAD systems Gerber & Excellon files for the layers.


The predefined profiles are shown in the list below. When the user select the desired PCB CAD profile and presses the button "Apply", IsoCAM3000 will select automatically the files corresponding to the PCB layers (according to the selected CAM profile).

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





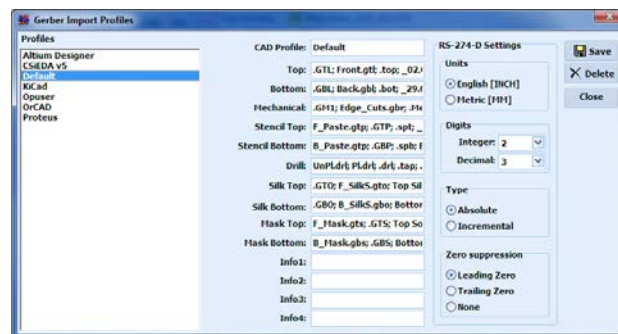
The predefined profile "Default" is a universal profile, and will work for most of the PCB CAD outputs (Gerber & Excellon).

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



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

IsoCAM3000 may support more than one extension for the file name for every layer. Typical example is the profile "Default":




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

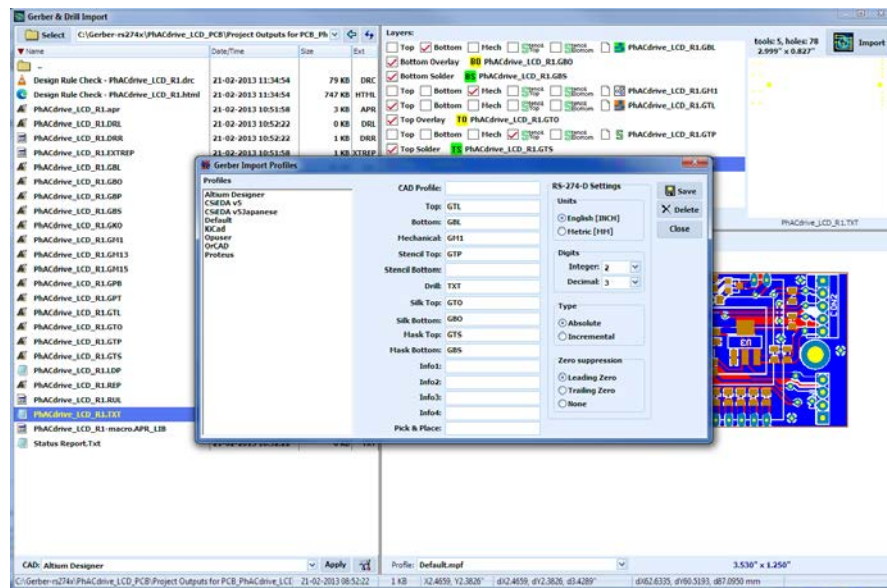
In the example, in the "Top" layer will be selected Gerber file which name ends with some of the next strings:

"GTL"
 "Front.gtl"
 ".top"
 "_02.GBR"
 "top.gbr"
 "Top Copper.TXT"
 "Top Pattern.gbr"

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

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

- Select the Gerber & Excellon files for the layers (ready for import into IsoCAM3000)
- Press the button 



Now IsoCAM3000 will fill the unique file names extensions (as is shown above) using the names of already selected files.

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

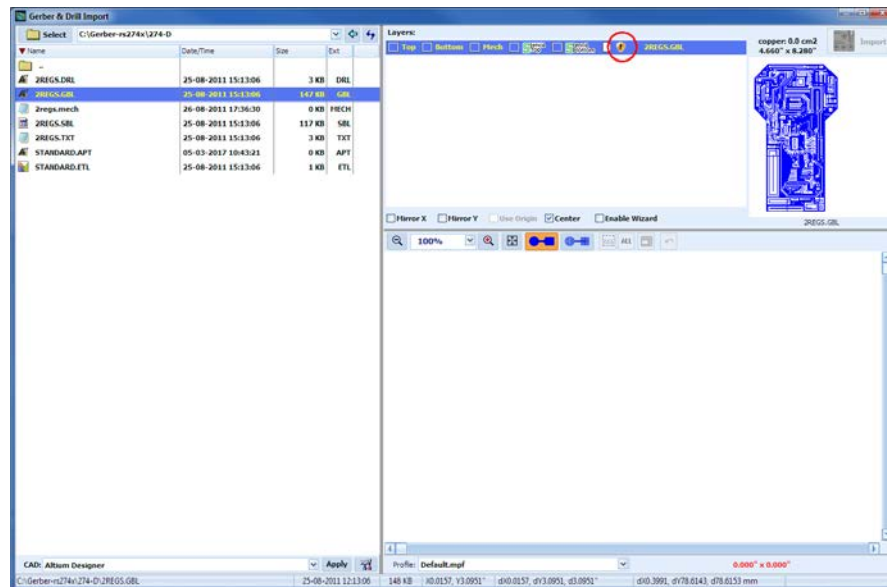
Video:

(2.1.4) <https://youtu.be/1535xRx2Vdw>



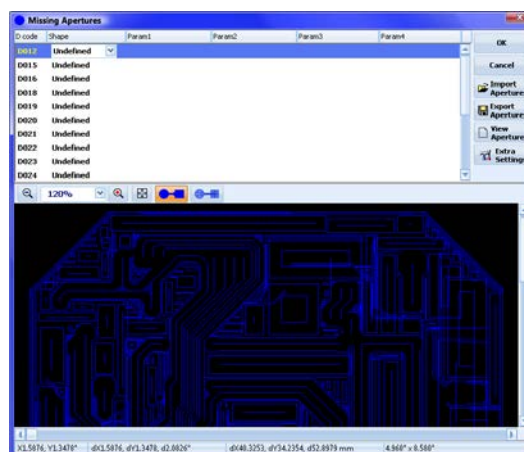
2.1.5 Gerber RS 274-D files import

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



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

2.1.5.1 Manual aperture entry



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

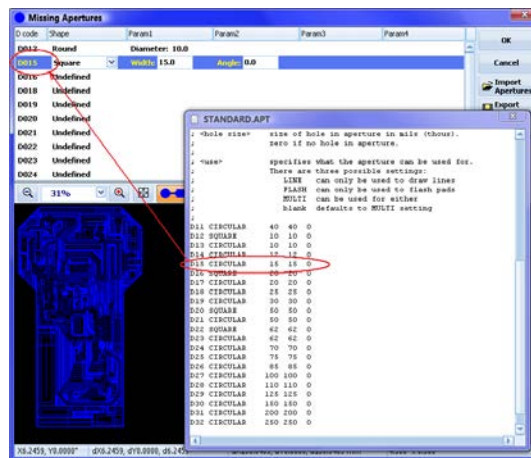


After the apertures and their parameters are entered, it is recommended to save them in a *.APT file for future use (another layer or another project) using the button "Export Apertures".



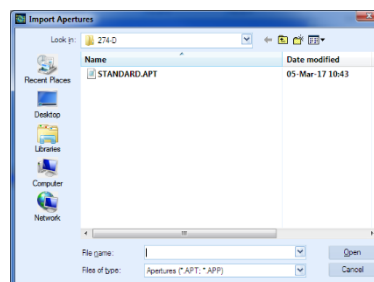
Important:

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

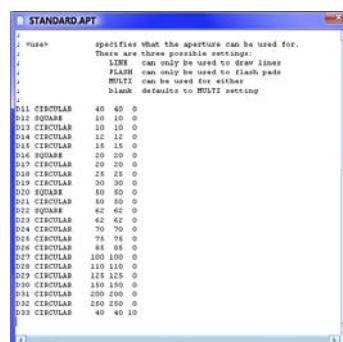


2.1.5.2 Aperture definitions import in IsoCAM3000 (file *.APT)

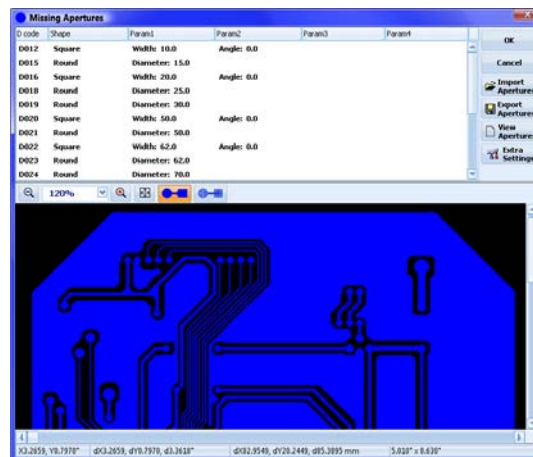
You can import apertures definitions in IsoCAM3000 when you use a RS 274-D Gerber format. Once the aperture file is generated (or supplied) you can enter it using “Import Apertures”



Shown below is the format of *.APT file that you can enter in IsoCAM3000.



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, IsoCAM3000 will redraw/display the changes in the layer.

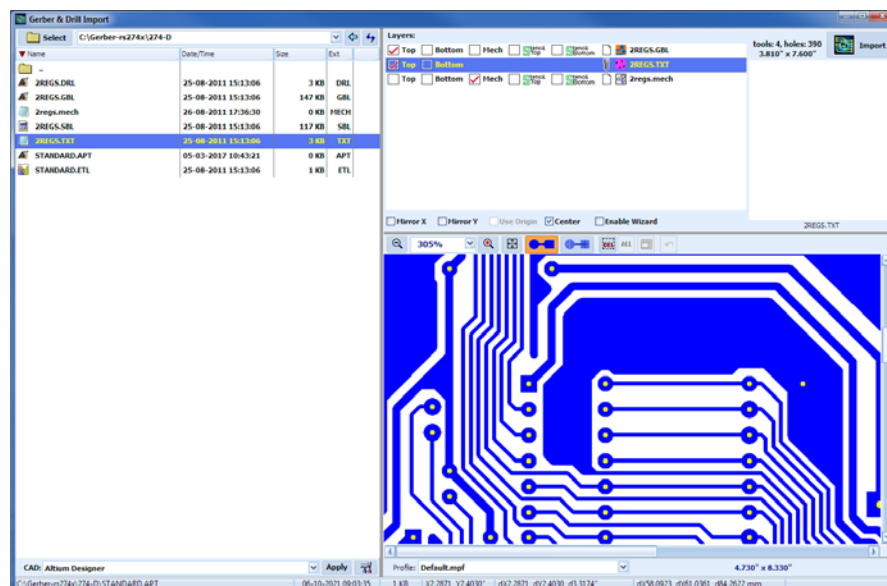


2.1.5.3 Extra Settings

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 IsoCAM3000 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.

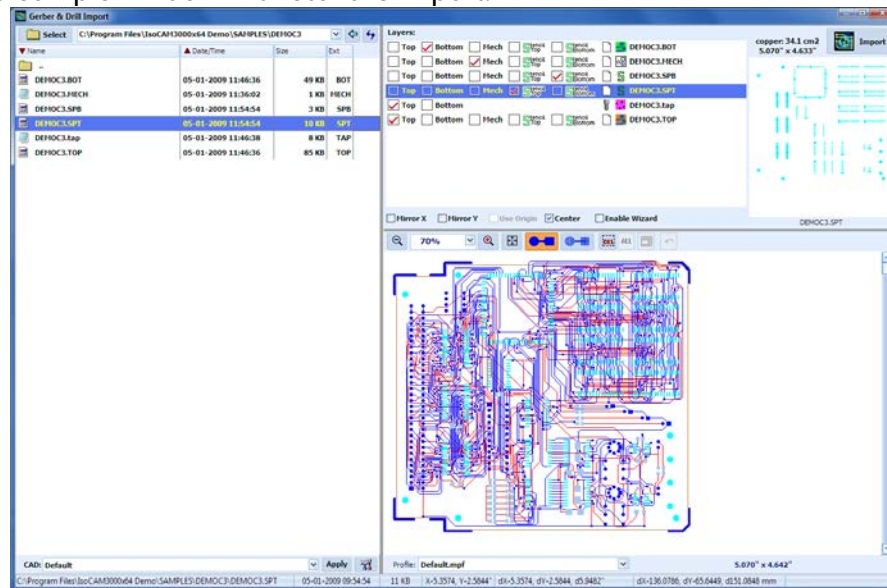
2.1.6 Gerber files for Stencils Import

IsoCAM3000 can import Gerber data layers designed to machine stencils for solder paste placement. Stencil layers can be imported together with the files for board(s) prototyping (recommended when cloning and reorientation of the board in IsoCAM3000 is needed) or separately. This import is similar to

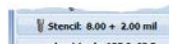


top and bottom Gerber files for boards. You may import top and bottom Stencil files ("Stencil bottom" is needed only for boards that are populated with components on both sides). "Stencil Top" and "Stencil Bottom" are virtual names, that reflect the stencil for board top side and the stencil for board bottom side, each made on separate work piece.

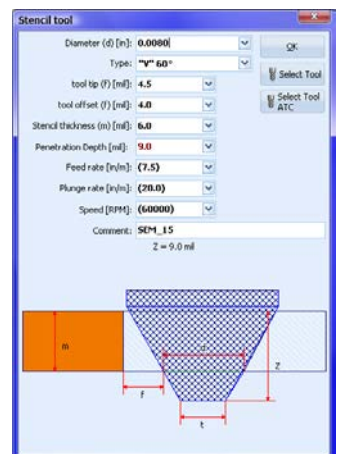
Shown below is a sample window with stencils import:



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 radii 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.

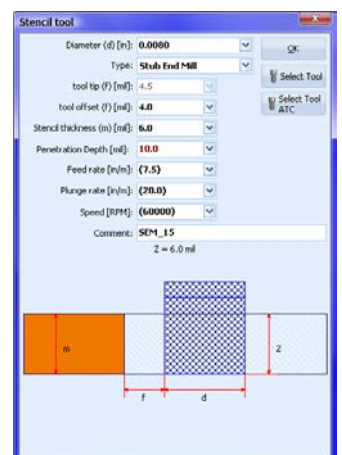


Where:

- '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 IsoCAM3000 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

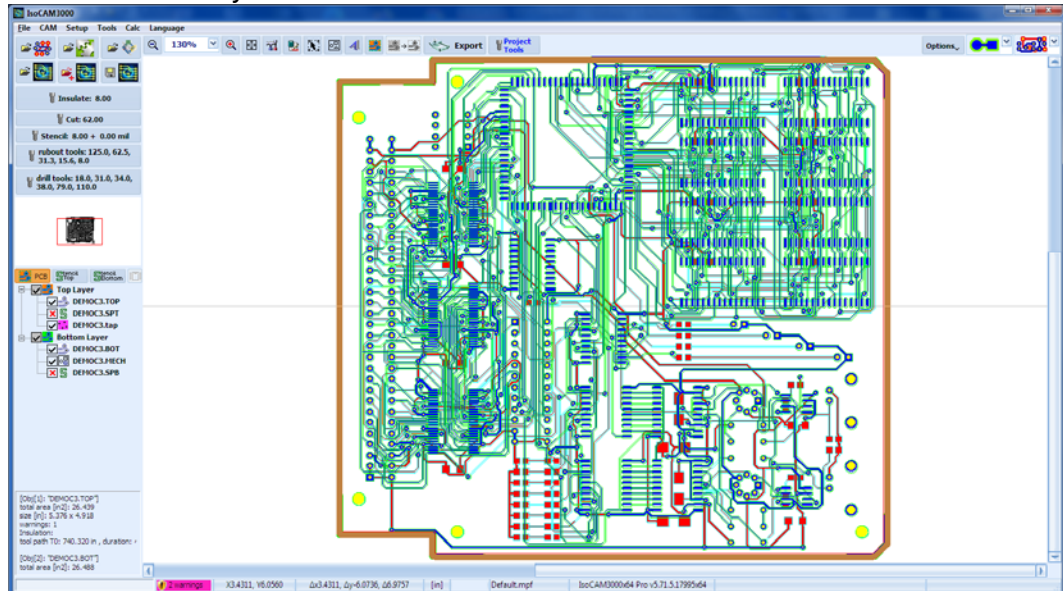
If a project containing stencil layer(s) has been imported, additional buttons that control the visualization of all layers will appear.



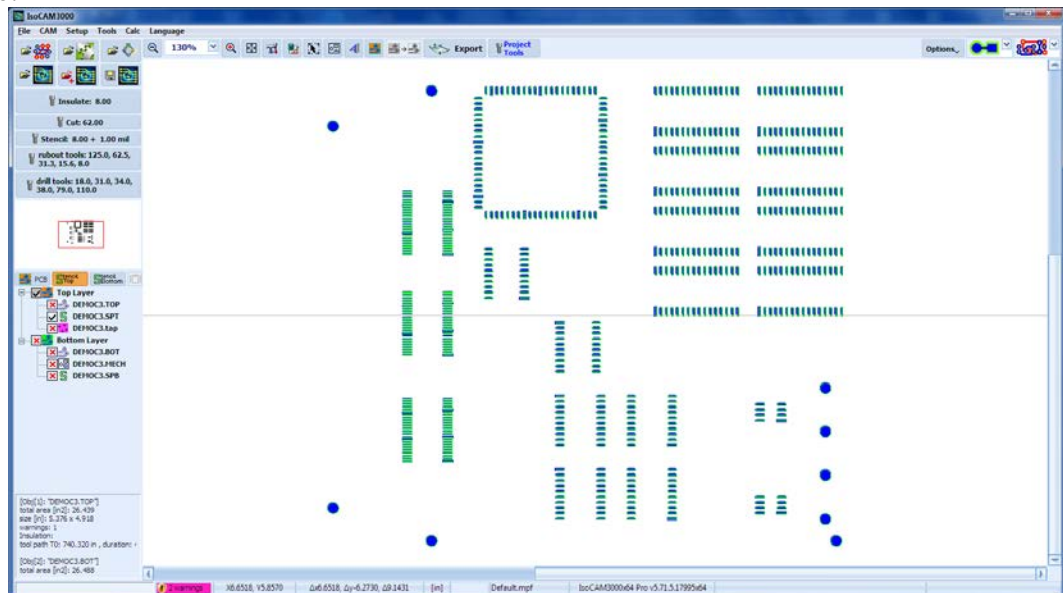
The "PCB" button, shows all files except the stencils

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

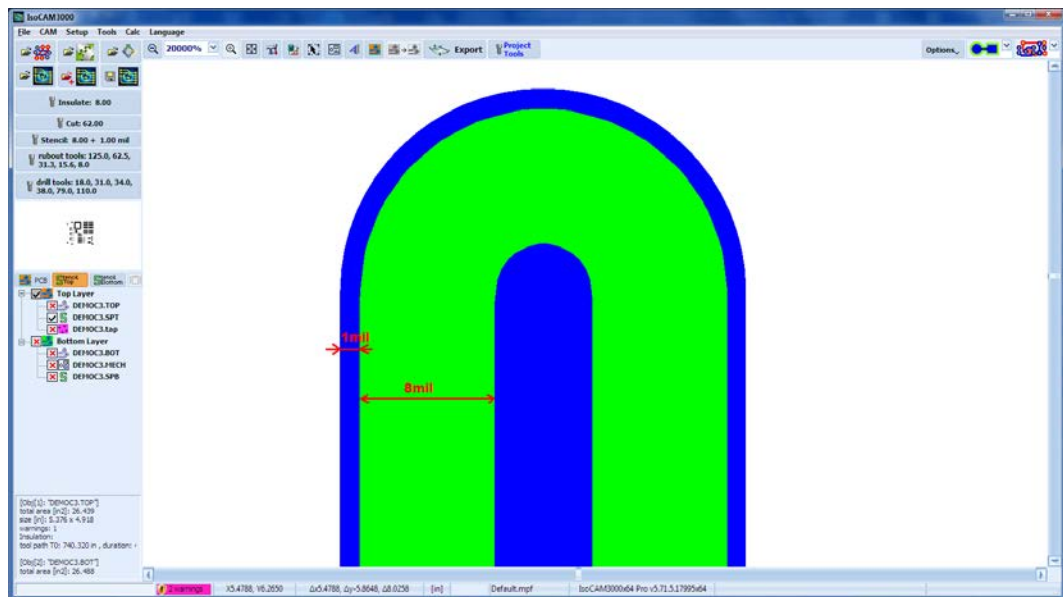
Shown below "PCB" files only:



Stencil Top:



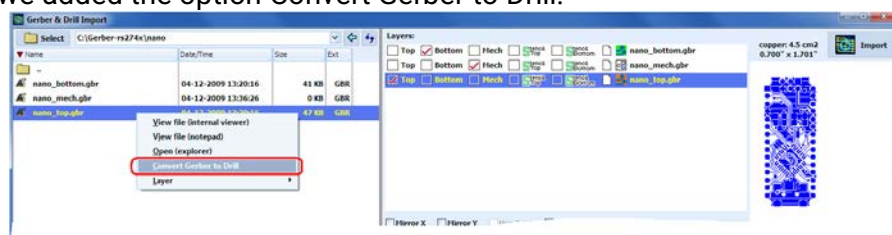
Below is an example of tool path dimensions and "safety zone" compensation

**Important:**

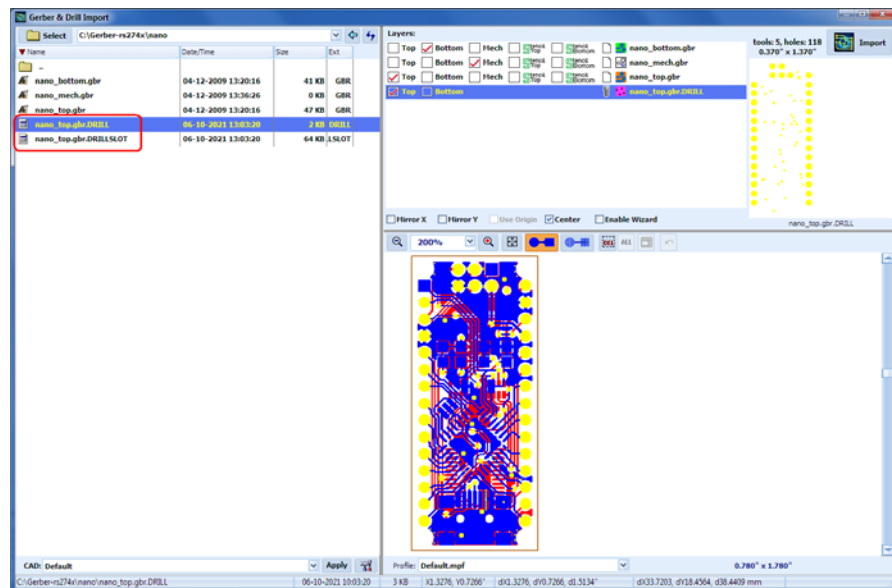
Due to significantly different properties of the materials used for stencils, there are separate settings for the machining in the "Setup" menu:

**2.1.7 Convert Gerber to Drill**

IsoCAM3000 allows drill files import in Excellon/Excellon2 formats. In rare cases it is possible that CAD system generates Gerber format (using round apertures for each hole). In order to be able to use these type of drill files we added the option Convert Gerber to Drill:



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



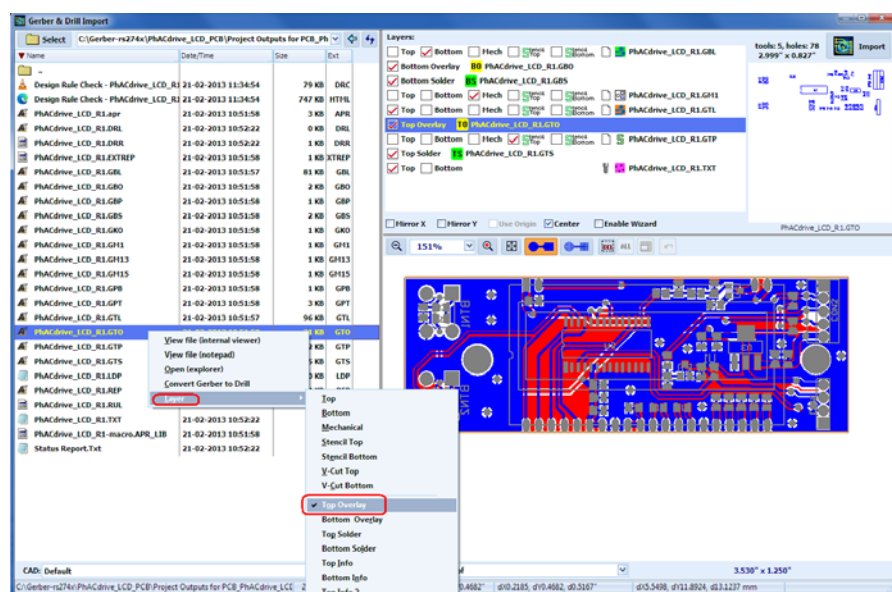
IMPORTANT

In order "Convert Gerber to Drill" to function properly the folder should allow to write files.
If a drill file already exist "Convert Gerber to Drill" option is disabled.
"Convert Gerber to Drill" option is also disabled if the file selected is not a Gerber file.
Please make sure that you are using full screen video to be able to see all details.

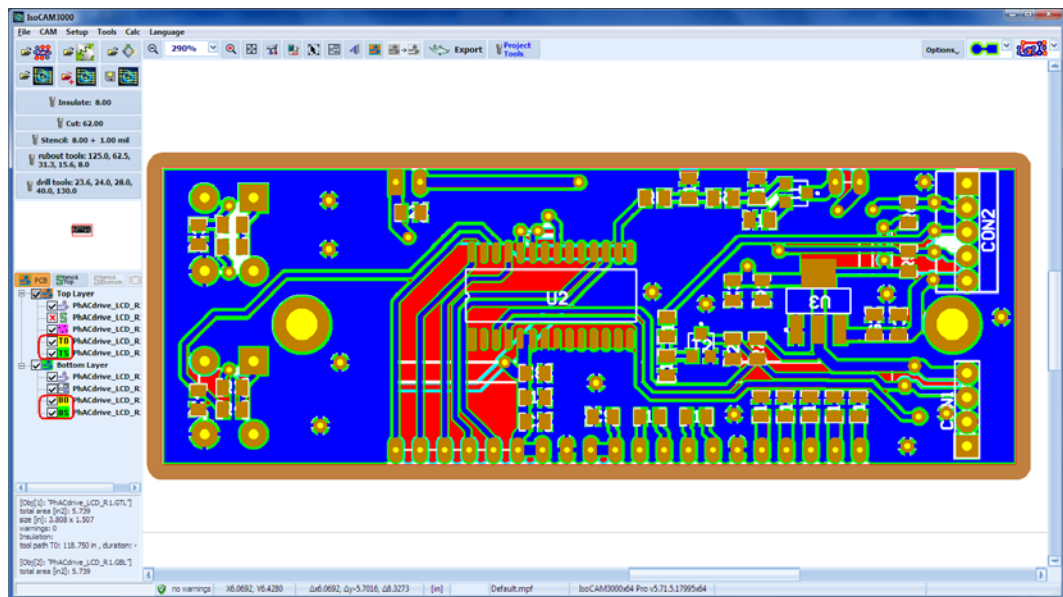
Convert Gerber to Drill - video:
(2.1.7) <https://youtu.be/29-tDWZ8ISE>

2.1.8 Info layers import

IsoCAM3000 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.

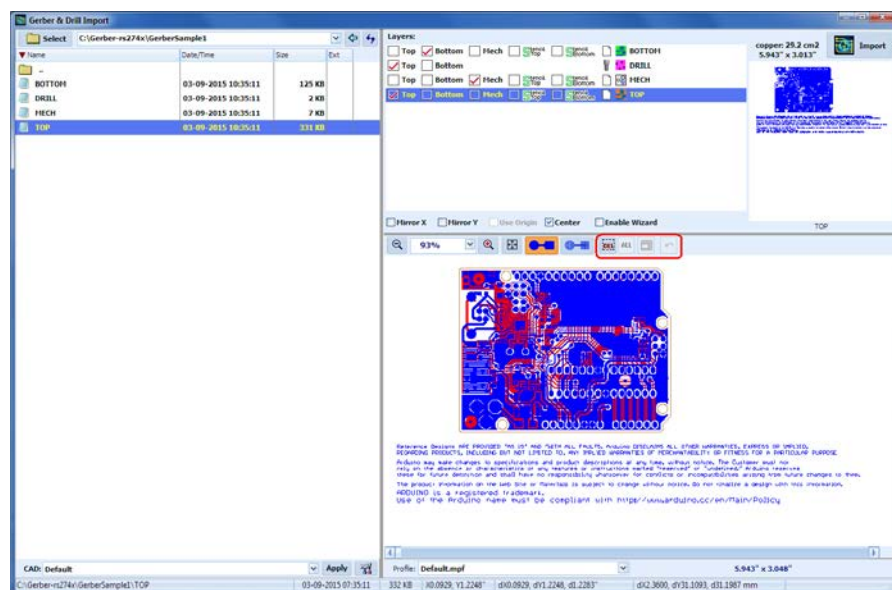




Layers for machining: "Top", "Bottom", "Drill", "Mechanical", "Stencil Top" (Top Paste), "Stencil Bottom" (Bottom Paste), also can be selected from popup menu.
After an import in IsoCAM3000, info layers can be shown/hidden the same way as the layers for machining.

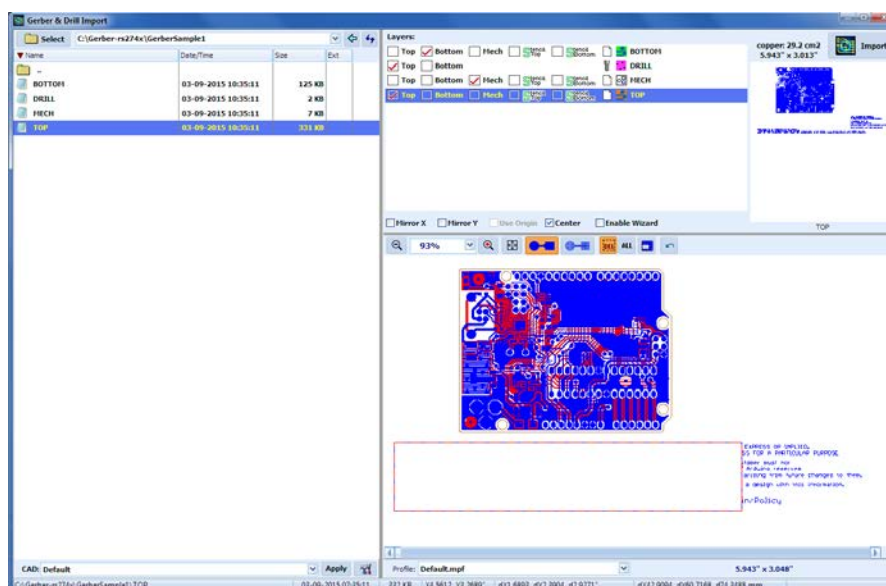




2.1.9 Gerber editor

IsoCAM3000 has a function that allows the operator to remove certain Gerber components when they are considered not needed for the machining. Typical examples include text, dimensional lines etc., that are (in some cases) included in the Gerber file. In the example below a large text field need to be removed from the Gerber file.



Activate the button . Using the mouse select the area to be deleted. All Gerber primitives that are completely included in the selected area will disappear from the selected file in the Layers list. If  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 IsoCAM3000. Please make sure that you are using full screen video in order to be able to see all details.

Gerber import example (editor) #1
(2.1.9.1) <https://youtu.be/3mD1id9C2hQ>

Gerber import example (editor) #2
(2.1.9.2) <https://youtu.be/9FPPrjE1djTc>

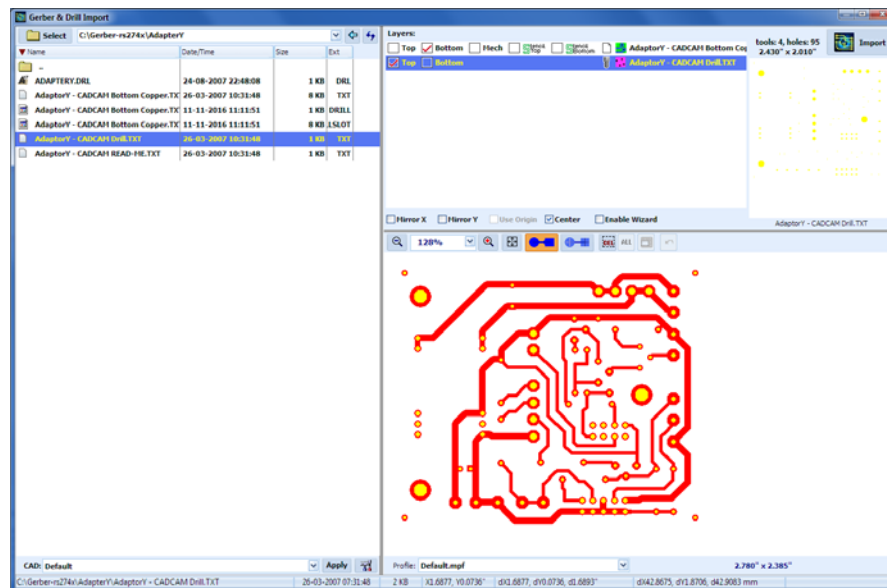
Gerber import example (editor) #3
(2.1.9.3) <https://youtu.be/9ajfbw8shKM>

2.1.10 Single layer PCB import

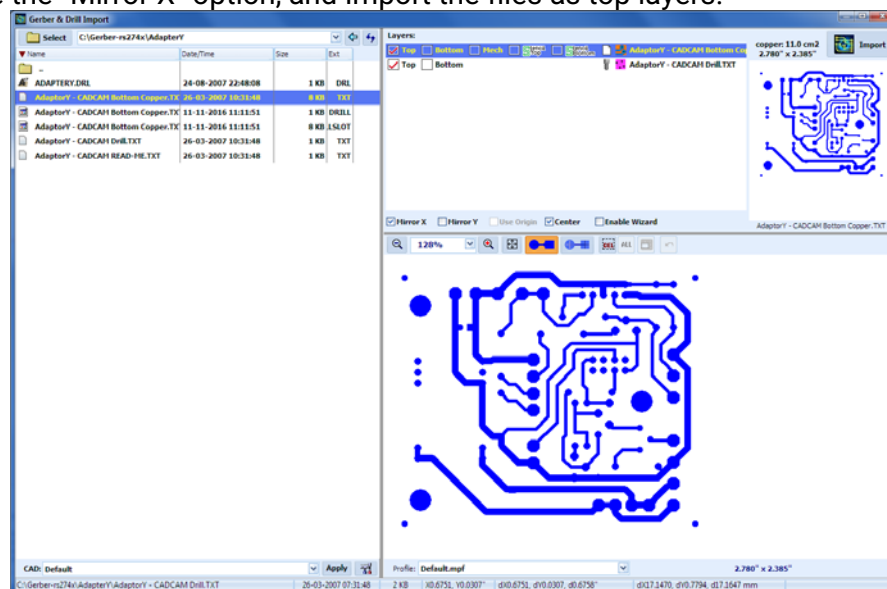
When you import single sided PCB designs the Gerber file for the solder layer (normally bottom layer) comes from your cad system as if you are looking at the board from the top. Our system mirrors this when you declare it "bottom layer", but this creates unnecessary difficulties in placing the project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

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

In the example shown below is the standard method of import



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



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

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

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

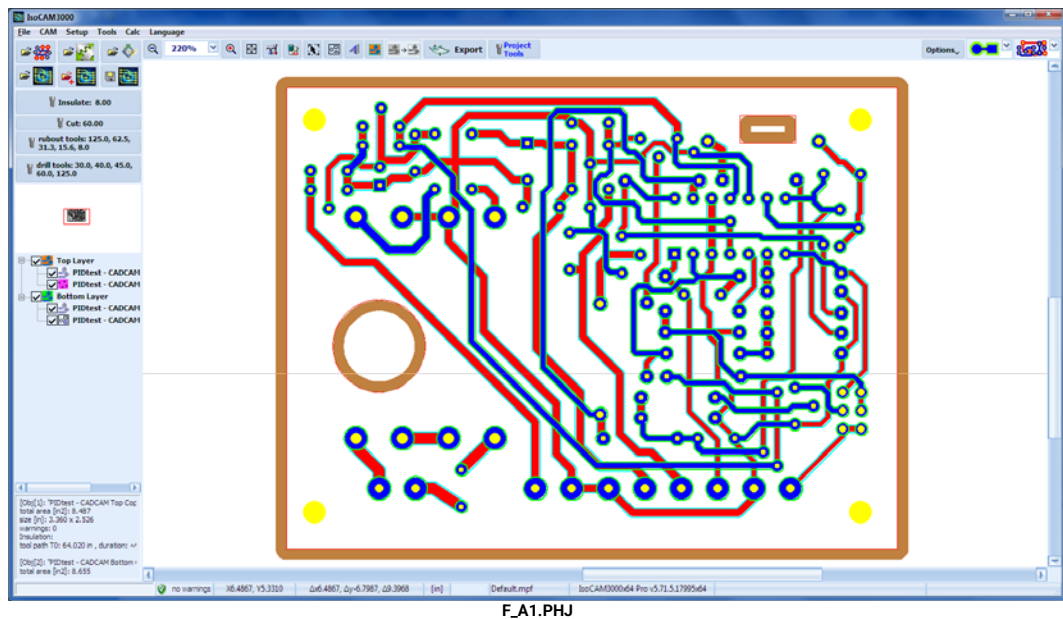
If you need to make more than one copy of any board, use clone (copy) function instead of loading your files again.

Cloned objects have some advantages:

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

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


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




2.2 IsoCAM3000 Project Files

IsoCAM3000 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.



IsoCAM3000 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 IsoCAM3000, 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  54%  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.

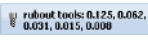
The button  is a **Very important setting!** It is for the insulation tools, used in the calculation of the insulation tool paths. If you change insulation tools, IsoCAM3000 automatically recalculates and redraws all insulation tool paths.

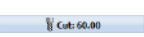
Important:

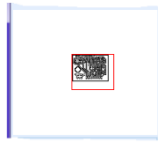
IsoCAM3000 insulation tool path calculations, **do not allow intrusion into the traces and pads**, if your tool diameter is bigger than the space between traces and/or pads they will be left non-insulated.

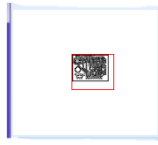


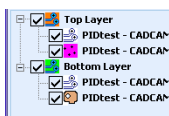
IsoCAM3000 detects and displays the problem areas (See 2.13 Design rules violation (DRV) detector). It is strongly recommended to view and check all critical zones of your design, using a high level of zoom!

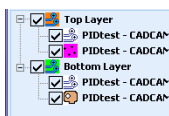
Button  is for Rubout tools selection (described later in this manual).

Drop down  is to set cutout tool diameter (if any). If you change this value, IsoCAM3000 automatically recalculates and redraws all cutout tool paths.



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



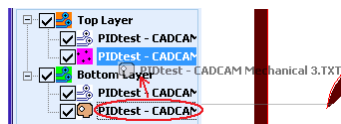
Window  is the loaded files window it shows all the loaded files and the layer they are on. Clicking on the checkmark ☒ will turn it into a cross-mark, disabling and hiding the corresponding layer.

Note:

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





Important:

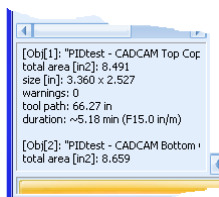
Disabled layers are not manipulated by the CAM edit functions. For example, if you like to select a rubout area in one of the layers, you can disable (hide) the other ones. If you need to move a file from one layer to another, click on file name, hold and drag.

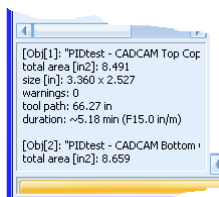


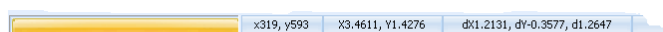
In the example above, the Mechanical layer is being moved from the Bottom to the Top Layer.

File type symbols:

-  - Gerber file
-  - Drill file
-  - Mechanical layer ("IsoCAM3000 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.

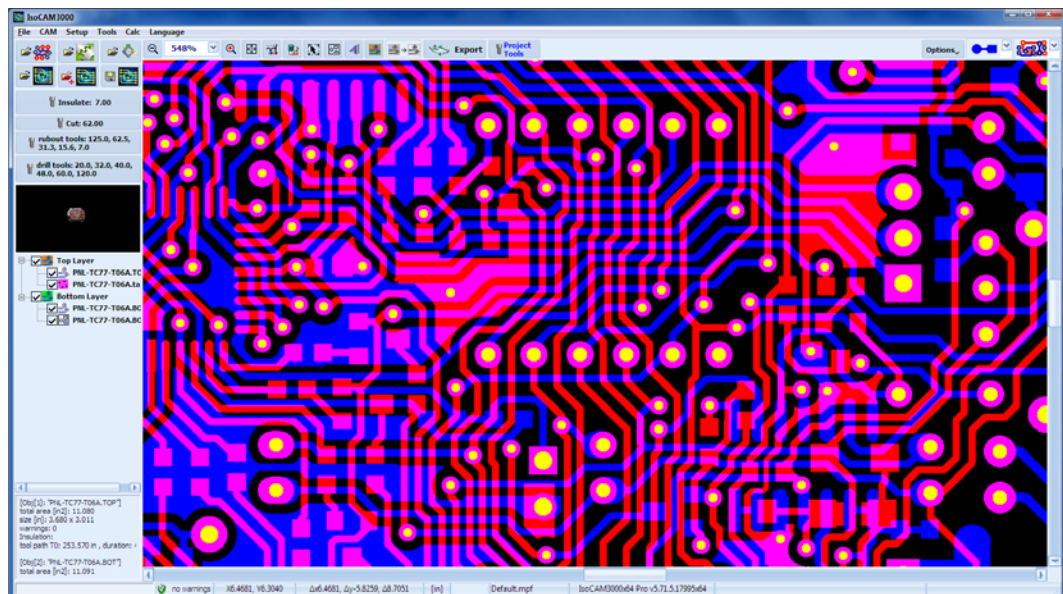
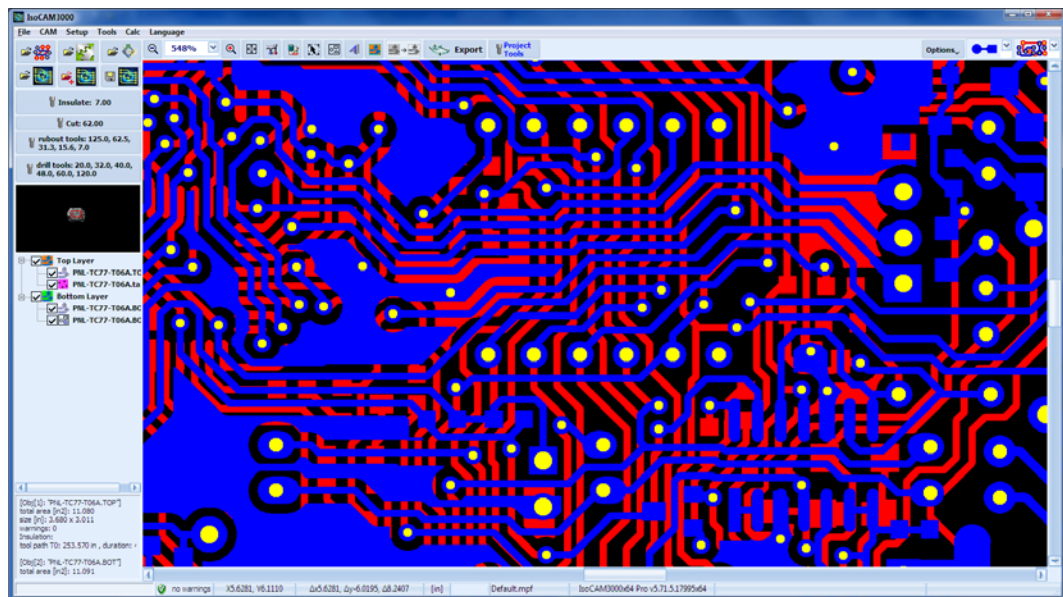
Drop down  is for controlling the way objects are displayed on the screen and will give you the following choices:



- 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 IsoCAM3000 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 IsoCAM3000 option will show / hide thumbtacks and is discussed in detail in section 2.14
- "Design rules violation" - this IsoCAM3000 option will activate/deactivate the DRV and is discussed in detail in section 2.13
- "Forced Insulation" - this IsoCAM3000 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 IsoCAM3000 option will activate/deactivate the full rubout and is discussed in detail section 2.15
- "Transparent Mode" - this IsoCAM3000 option will give you a transparent top layer and the ability to see top and bottom layers at the same time. This option is useful for review/inspection of double-sided projects. Below is an example of the usage.







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





Note:




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





Drop down  is for visualization modes.

Buttons  &  are for a special visualization mode called "copper view". In this mode the screen shows what should be the final result after executing all machining programs. It uses close to real colors to show copper and typical base material. In CNC mode of IsoCAM3000 these buttons are inactive!

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

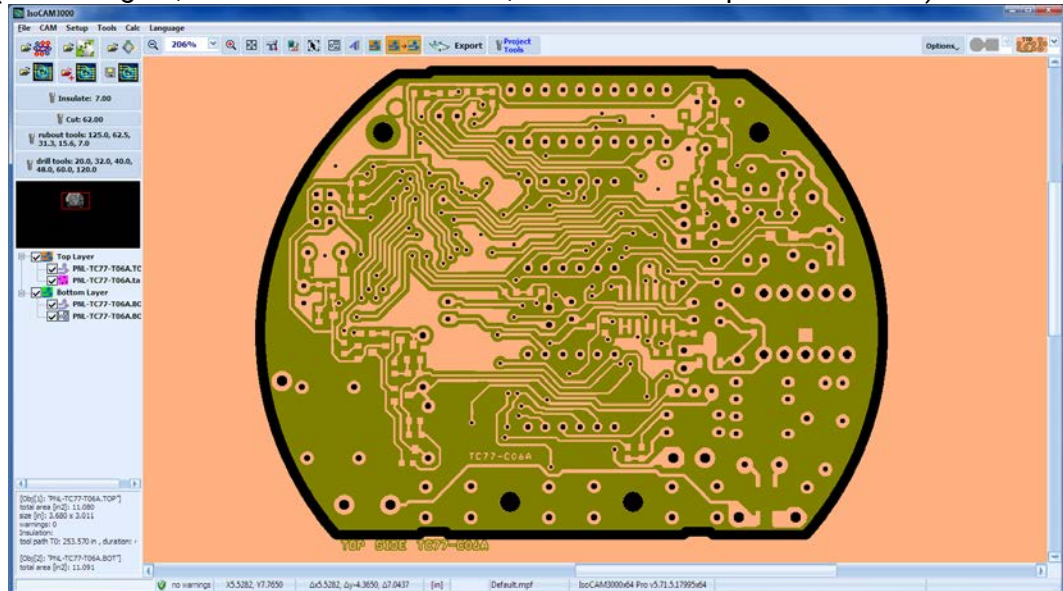
-  - top, bottom, drill & mechanical (natural layers order)
-  - bottom, top, drill & mechanical (bottom layer on top)
-  - top, drill & mechanical (bottom layer hidden)



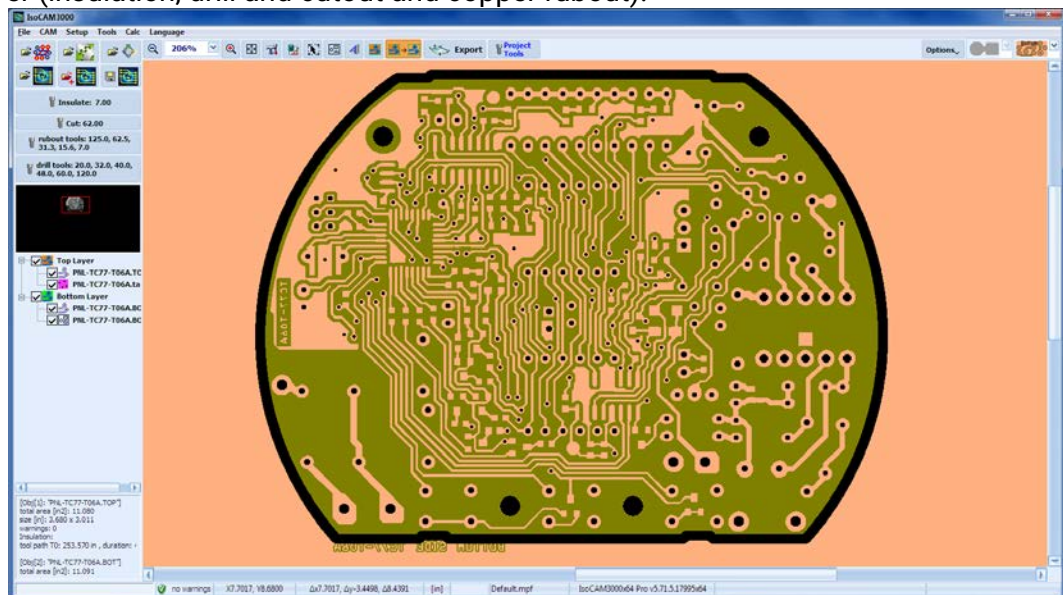
 - bottom, drill & mechanical (top layer hidden)
by default,  is active.
In CNC mode of IsoCAM3000 all these buttons are inactive!

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 IsoCAM3000
- Button  is to access the built-in calculator
- Button  is to enter select objects mode (selected objects can be moved, cloned, rotated, grouped in a single object, ungrouped, and deleted)
- Button  is to enter tool diameter compensation selection mode (used in the mechanical layer)
- Button  is to enter rubout regions selection mode
- Button  is to start the rubout regions tool paths calculation



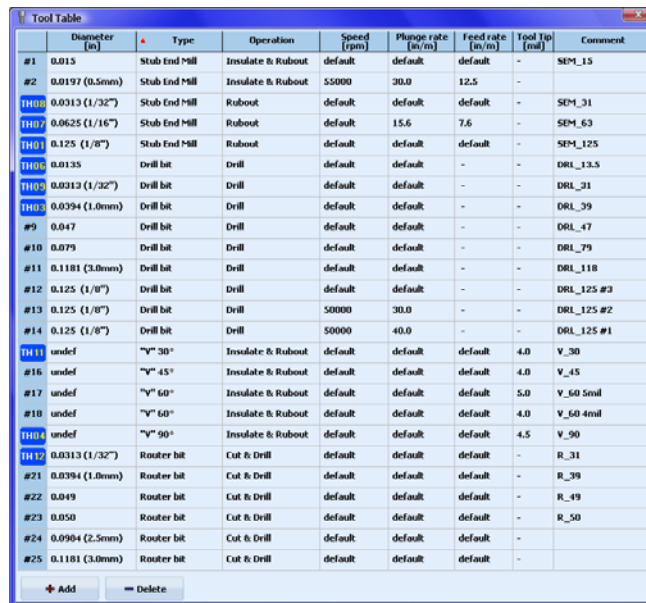
Button  is to switch between CAM/CNC modes

Button  is to enter the project tools list



2.4 Tools

In order to make the operation of IsoCAM3000 easier and to organize the tool settings we have tool tables. In the tool table you may enter all tools that are available for use. For each tool you may enter size (diameter [inch]), type, useful operations, Plunge Rate, Feed Rate, Speed [RPM]

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



#	Diameter [in]	Type	Operation	Speed [rpm]	Plunge rate [in/m]	Feed rate [in/m]	Tool Tip [mm]	Comment
#1	0.015	Stub End Mill	Insulate & Rubout	default	default	default	-	SEM_15
#2	0.0197 (0.5mm)	Stub End Mill	Insulate & Rubout	55000	30.0	12.5	-	
TH00	0.0313 (1/32")	Stub End Mill	Rubout	default	default	default	-	SEM_31
TH02	0.0625 (1/16")	Stub End Mill	Rubout	default	15.6	7.6	-	SEM_63
TH01	0.125 (1/8")	Stub End Mill	Rubout	default	default	default	-	SEM_125
TH00	0.0135	Drill bit	Drill	default	default	-	-	DRILL_135
TH05	0.0313 (1/32")	Drill bit	Drill	default	default	-	-	DRILL_31
TH03	0.0394 (1.0mm)	Drill bit	Drill	default	default	-	-	DRILL_39
#9	0.047	Drill bit	Drill	default	default	-	-	DRILL_47
#10	0.079	Drill bit	Drill	default	default	-	-	DRILL_79
#11	0.1181 (3.0mm)	Drill bit	Drill	default	default	-	-	DRILL_118
#12	0.125 (1/8")	Drill bit	Drill	default	default	-	-	DRILL_125 #3
#13	0.125 (1/8")	Drill bit	Drill	50000	30.0	-	-	DRILL_125 #2
#14	0.125 (1/8")	Drill bit	Drill	50000	40.0	-	-	DRILL_125 #1
TH11	undef	"V" 30°	Insulate & Rubout	default	default	default	4.0	V_30
#16	undef	"V" 45°	Insulate & Rubout	default	default	default	4.0	V_45
#17	undef	"V" 60°	Insulate & Rubout	default	default	default	5.0	V_60 Smil
#18	undef	"V" 60°	Insulate & Rubout	default	default	default	4.0	V_60 4mil
TH04	undef	"V" 90°	Insulate & Rubout	default	default	default	4.5	V_90
TH12	0.0313 (1/32")	Router bit	Cut & Drill	default	default	default	-	R_31
#21	0.0394 (1.0mm)	Router bit	Cut & Drill	default	default	default	-	R_39
#22	0.049	Router bit	Cut & Drill	default	default	default	-	R_49
#23	0.050	Router bit	Cut & Drill	default	default	default	-	R_50
#24	0.0904 (2.5mm)	Router bit	Cut & Drill	default	default	default	-	
#25	0.1181 (3.0mm)	Router bit	Cut & Drill	default	default	default	-	

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

- **Diameter** is the cutting diameter of the tool. You can select it from drop down list or enter it manually. Entries are in inches, if the entry is bigger than 1.0, it is accepted as mils. For example, if you enter 15 – this will be 0.015"
- **Type** is the type of the tool. It is recommended to fill this field. It helps during the selection of tools for certain operations. The Possible types are "undefined", "Drill bit", "Router bit", "Stub End Mill", "V 25°", "V 30°", "V 45°", "V 60°", "V 90°", "V 100°" and "V 120°"
For tools of type "V", the track size (diameter column) depends on the tool penetration into the material. It is recommended to select "undef" (or enter 0.0)!
- **Operation** is to show the possible usage of the tool. The choices are "undefined", "Insulate", "Rubout", "Cut & Drill", "Drill", and "Cut". For example, if the type of tool is a "Router Bit", then the possible choice of Operation will be limited to: "Cut & Drill", "Drill" and "Cut"; The reason for this is a router bit can be used for drilling holes larger than its diameter.
- **Speed** is for setting the RPM of the spindle for this tool. If the choice is "default", the setting speed is selected from global parameters table for this operation (see 3.4 CNC process control settings). Possible settings are 5,000...60,000 and 5,000...100,000 RPM depend of the model.
- **Plunge Rate** is for setting the speed of Z axis toward the table, e.g. the speed that tool will penetrate the material. If selection is "default" the setting is from global table for selected operation (see 3.4 CNC process control settings). Possible values are 40...0.1 in/min.
- **Feed Rate** is for setting the tool movement speed in XY plane (in/min). If "default" is selected it takes the speed setting from global table (see 3.4 CNC process control settings). Possible settings are 15...0.1 in/min. This parameter has no meaning for "Drill bit" type.



- **Comment** is a field for making notes, maximum length 256 characters.
- **Tool Life [in]/#** is a field that shows maximum tool usage acquired by the manufacturer specs or by own experience

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

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

It can be set for every tool (if needed) and will prompt the operator that the limit set is exceeded.

IsoCAM3000 records both (total number of material entries and total length of cutting) for routers and end mills. It records only the number of drill cycles for the drill tools.

Example:

	Diameter [in]	Type	Operation	Speed (rpm)	Plunge rate (in/min)	Feed rate (in/min)	Tool [in]	Tool Life [in]/#	Comment
#12	0.035	Drill bit	Drill	default	default	-	-	10000	DRL_30
#13	0.047	Drill bit	Drill	default	default	-	-	15000	DRL_47
#14	0.079	Drill bit	Drill	default	default	-	-	15000	DRL_79
#15	0.1101 (3.0mm)	Drill bit	Drill	default	default	-	-	15000	DRL_110
#16	0.125 (1/8")	Drill bit	Drill	default	default	-	-	15000	DRL_125 #1
#17	0.125 (1/8")	Drill bit	Drill	default	default	-	-	15000	DRL_125 #2
#18	0.125 (1/8")	Drill bit	Drill	default	default	-	-	15000	DRL_125 #3
#19	undef	"V" 30°	Insulate & Rubout	default	default	default	3.5	2000 in	V_30
#21	undef	"V" 45°	Insulate & Rubout	45000	default	default	6.5	2000 in	V_45
#22	undef	"V" 45°	Insulate & Rubout	default	default	default	4.5	2000 in	V_45
#23	undef	"V" 60°	Insulate & Rubout	default	default	default	4.0	2000 in	V_60
#24	undef	"V" 90°	Insulate & Rubout	default	default	default	4.0	-	V_90
#25	undef	"V" 90°	Insulate & Rubout	default	default	default	4.5	500 in	V_90
#26	0.0313 (1/32")	Router bit	Cut & Drill	default	default	default	-	-	R_31
#27	0.0394 (1.0mm)	Router bit	Cut & Drill	default	default	default	-	10000 in	R_39
#28	0.049	Router bit	All	default	default	default	-	-	R_49
#29	0.055	Router bit	Cut & Drill	default	default	default	-	-	R_50
#30	0.0904 (2.5mm)	Router bit	Cut & Drill	default	default	default	-	-	R_90
#31	0.1101 (3.0mm)	Router bit	Cut & Drill	default	default	default	-	-	R_110

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.

Tool Life

Max Life Cycles

Max Life Path [in]: 2000 in

Max # of holes/plunges: 1000

Current Life Cycles

Current Life Path [in]: 2427 in

Current # of holes/plunges: 1512

OK Cancel New Tool

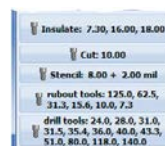
Important:

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).

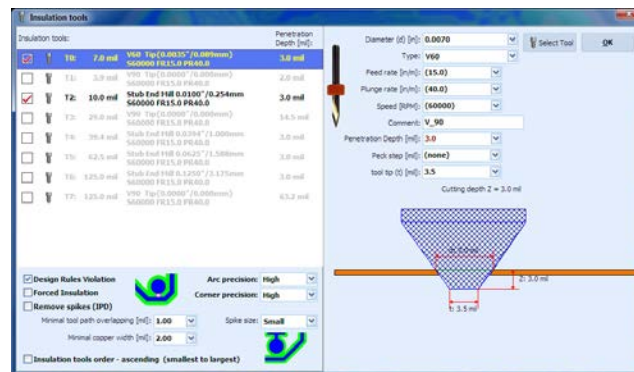
In the left portion of the screen (IsoCAM3000 in CAM mode) you will find buttons for tools and tool tables for each operation.



2.4.1 "Insulate" Tools

IsoCAM3000 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, IsoCAM3000 is using only the selected tools (☑).

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

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

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

IsoCAM3000 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, IsoCAM3000 will always use the smallest tool available.

Additional options related to insulation process

• Design Rules Violation (DRV)

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

• Forced Insulation

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

• Remove Spikes (IPD)

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

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

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

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

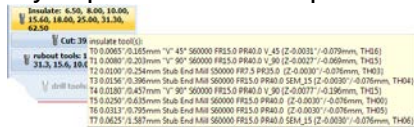
When you select an insulation tool from the main table, only tools that have "Insulate & Rubout" and "Insulate" in the "Operation" column will be present in the list.



	Diameter [in]	Type	Operation	Speed [rpm]	Plunge rate [in/min]	Feed rate [in/min]	Tool ID [in]	Tool Life [min]	Comment
[H01]	0.010	Stub End Mill	All	50000	35.0	7.5	-	3000 in	
[H02]	0.016	Stub End Mill	All	default	default	default	-	-	SEPI_15
[H03]	0.0313 (1/32")	Stub End Mill	Insulate & Ribout	default	default	default	-	-	
[H04]	0.0625 (1/16")	Stub End Mill	All	default	default	default	-	-	SEPI_63
[H05]	0.125 (1/8")	Stub End Mill	All	default	default	default	-	-	SEPI_125
[H06]	0.0135	Drill bit	Drill	default	default	-	-	7500	DRL_13.5
[H07]	0.022	Drill bit	Drill	default	default	-	-	10000	DRL_21.65
[H08]	0.025	Drill bit	Drill	default	default	-	-	10000	
[H09]	0.0313 (1/32")	Drill bit	Drill	default	default	-	-	10000	DRL_31
[H10]	0.035	Drill bit	Drill	default	default	-	-	10000	DRL_39
[H11]	undef	"Y" 45°	Insulate & Ribout	default	default	default	4.5	2000 in	V_45
[H12]	undef	"Y" 60°	Insulate & Ribout	default	default	default	5.5	2000 in	V_60
[H13]	undef	"Y" 90°	Insulate & Ribout	default	default	default	4.0	-	V_90
[H14]	0.055	Router bit	Cut & Drill	default	default	default	-	-	R_50

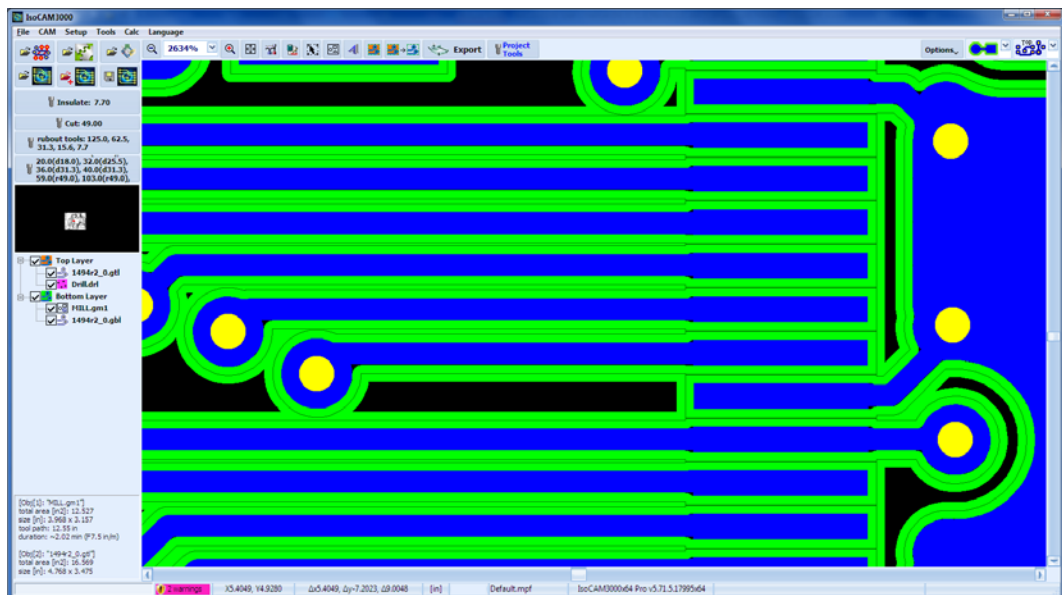
The information contains actual parameters (If "default" is selected you will see the value from the global table). These are the parameters that will be used for the program generation for this process.

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



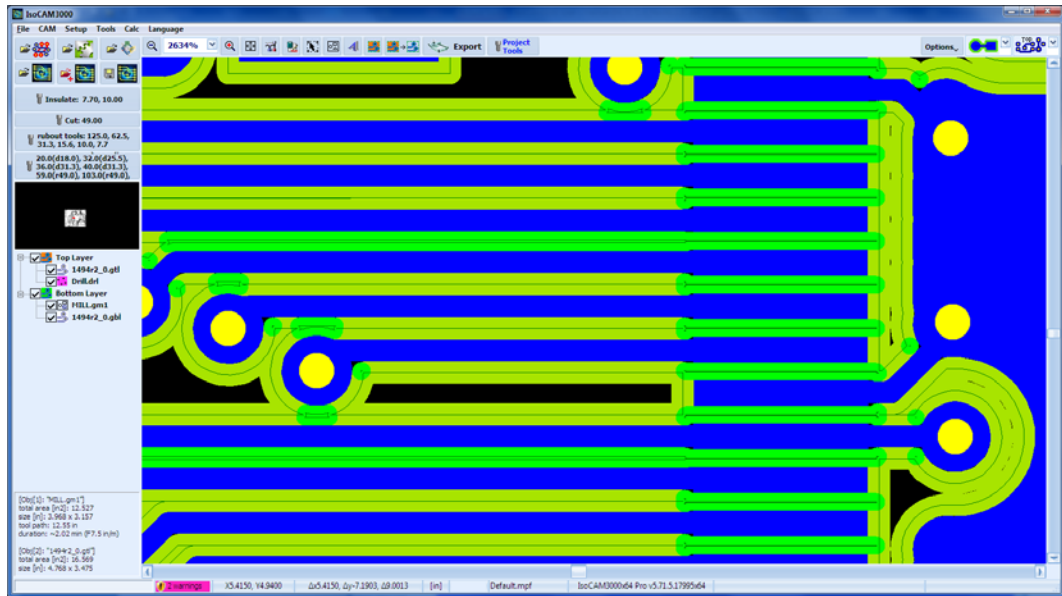
As we mentioned above a PCB design can be insulated with one two or more tools. Shown below is a single design machined with 1,2 and 3 tools

Single tool (7.7 mil V60):



tool path T0: 489.670 inch

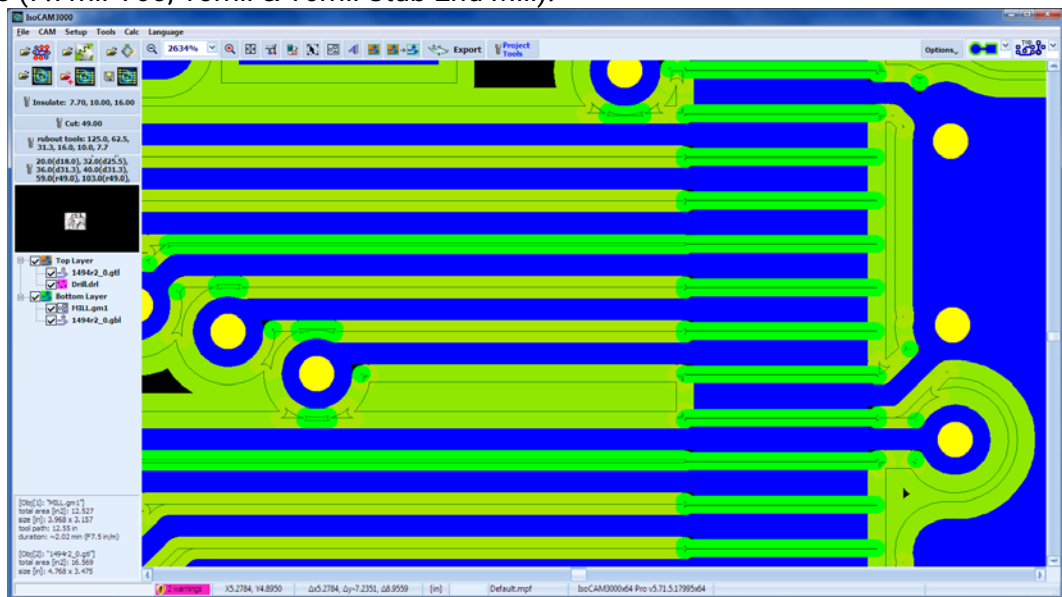
Two tools (7.7mil V60, 10.0mil Stub End Mill):



tool path T0: 120.960 inch

tool path T1: 388.900 inch

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



tool path T0: 120.960 inch

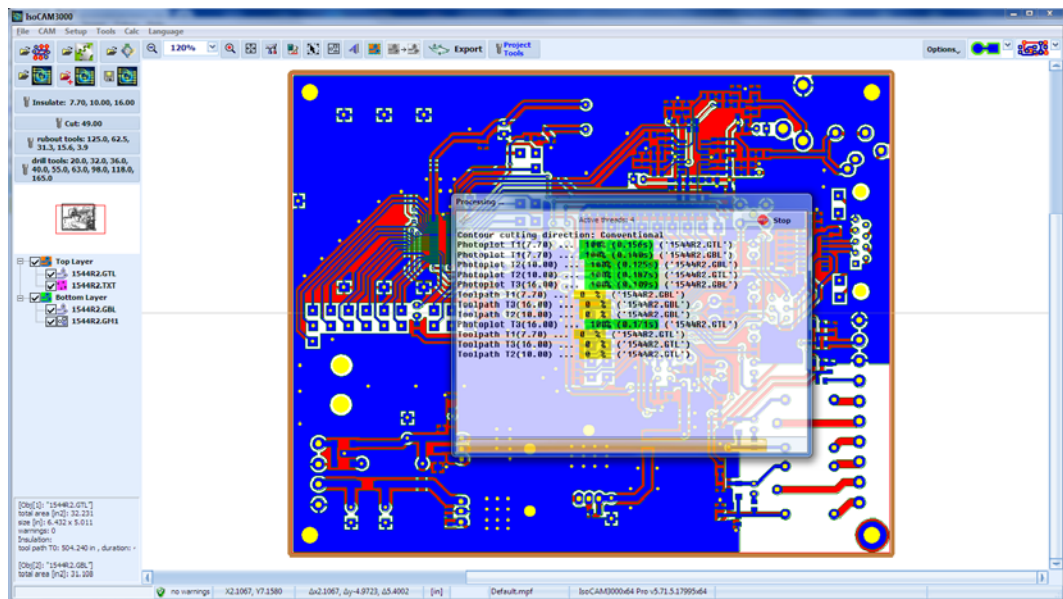
tool path T1: 184.330 inch

tool path T2: 291.000 inch

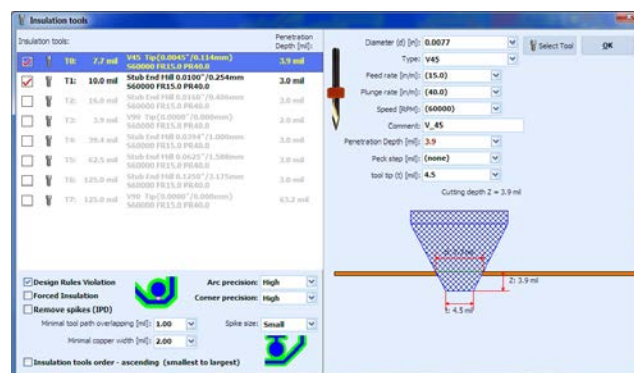
Important:

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

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



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



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

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

2.4.2 "Cut" Tool

To select the cutout tool left click on the button.

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

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

Tool Table										
	Diameter [m]	A	Type	Operation	Speed [rpm]	Plunge rate [m/min]	Feed rate [m/min]	Tool Tip [mm]	Tool Life [min]	Comment
[H03]	0.010		Stub End Mill	All	50000	35.0	7.5	-	3000 in	
[H04]	0.016		Stub End Mill	All	default	default	default	-	-	SEM_15
[H06]	0.0625 (1/16")		Stub End Mill	All	default	default	default	-	-	SEM_63
[H07]	0.125 (1/8")		Stub End Mill	All	default	default	default	-	-	SEM_125
[H14]	0.055		Router bit	Cut & Drill	default	default	default	-	-	R_50

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

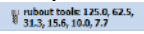
If you place the mouse pointer on the tool button, you will see the information for the selected tool in a popup window.

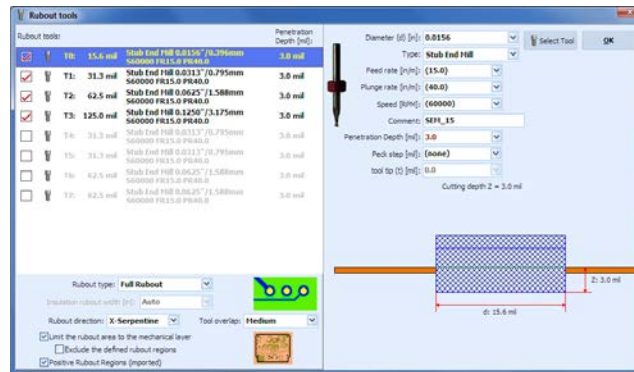


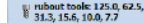
The information contains the actual parameters. If the selected tool uses default parameters, its actual value is in brackets. For example, if the RPM is set to default (60000) will be shown. This means that, according to current tool profile, the spindle speed will be 60,000 RPM because the global table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

2.4.3 "Rubout" Tools

IsoCAM3000 now allows usage of up to 8 additional tools for the "copper rubout" process in addition to 8 tools available in "insulation" process. This way the maximum number of tools used in rubout process can go up to 16 (8 insulation tools and 8 rubout dedicated tools). Copper rubout process is actually an extension to the process of insulation and can be considered or rejected depend on the final goals of the design. We consider the total number of tools quite excessive but like to have it in order to meet the requirements in technology that may come up in the future. This will allow us to meet more complicated needs without making structural changes in the program. Currently a medium complicated project needs total of 4 to 5 tools to execute the processes of Insulation and Copper rubout.

To select the Rubout tools table click on the  button.



Important: In the table above, you have to enter the desired copper rubout tools. IsoCAM3000 software will use all insulation tools declared as the smallest rubout tools. If the rubout tools table contain any tool smaller than largest insulation tool it will not be used in the process. In the rubout tools button  you can see the list of the tools that will be used in the process. The requirement for the tool diameters is the same as for Insulation tools. The difference in diameters must be larger than 1 mil. Only tools with check mark will be included in the process.

The process has 3 options for the rubout direction

- X serpentine
- Y serpentine
- Conical

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

IsoCAM3000 offers 2 types of copper rubout:

- Full rubout

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

- Insulation rubout

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



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

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

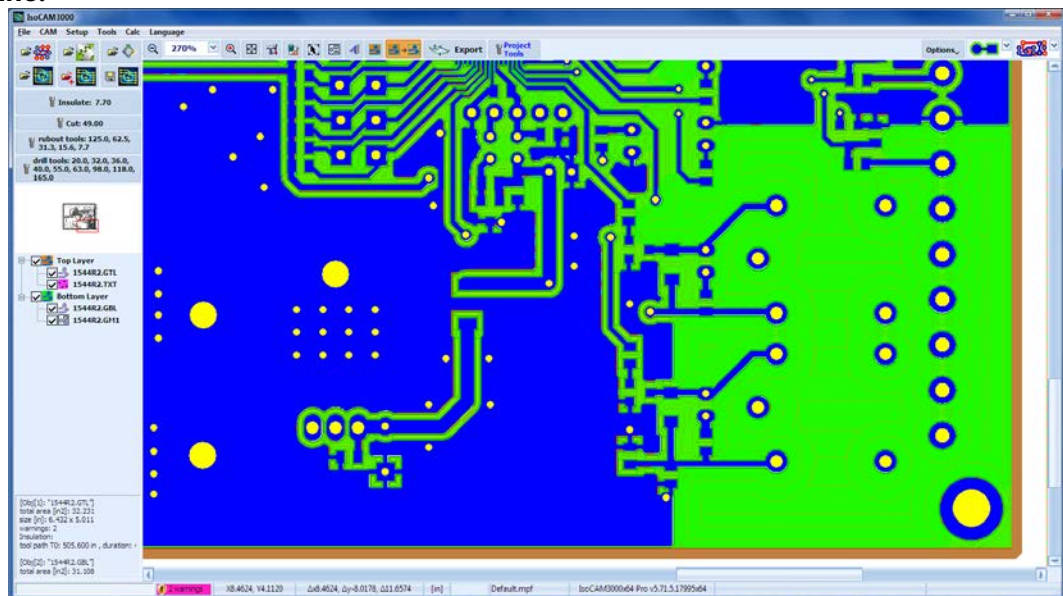
Important:

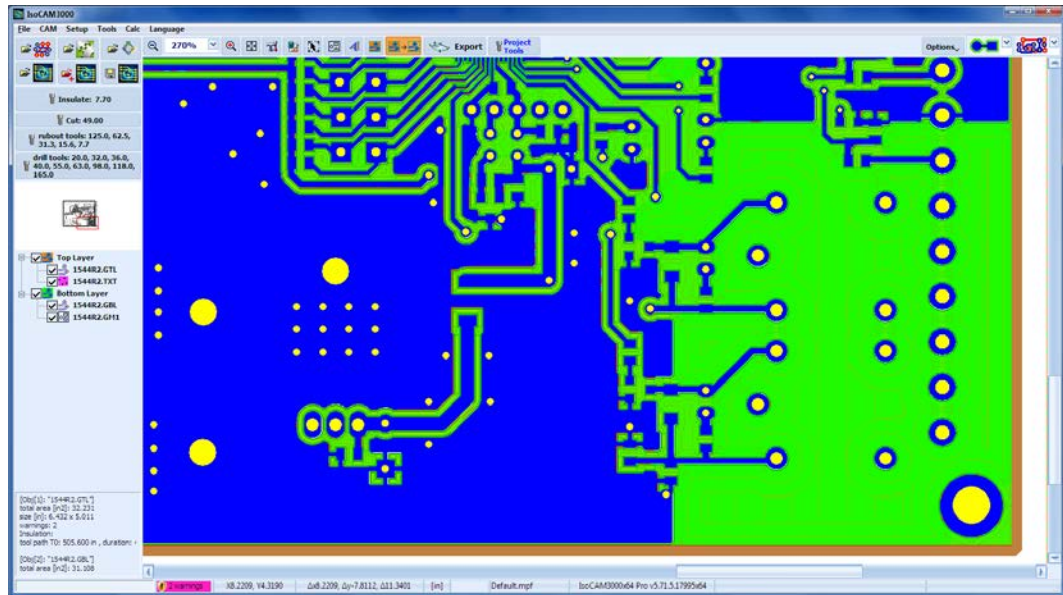
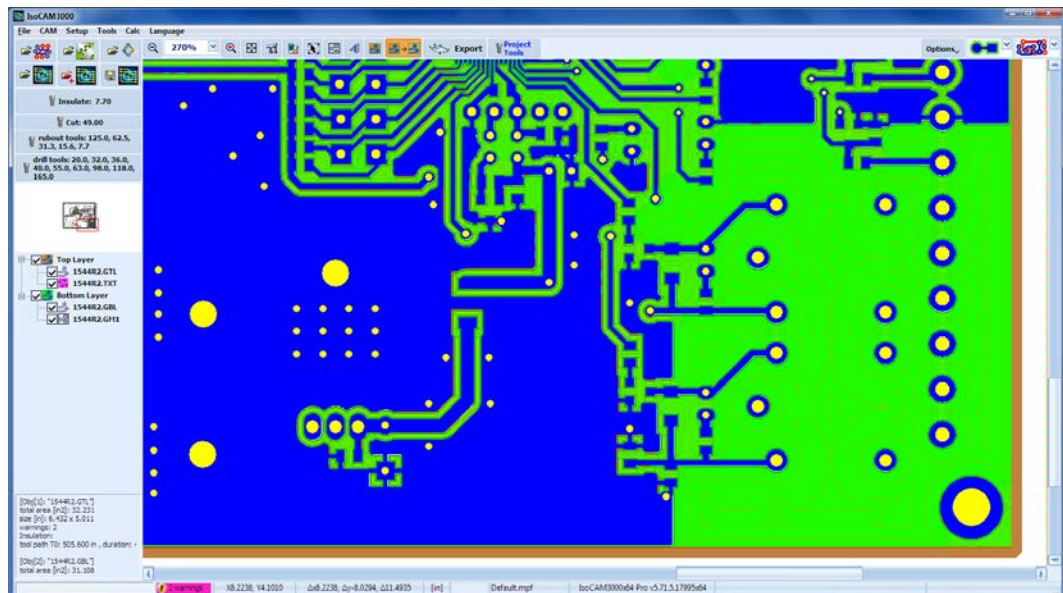
When Full rubout is selected, it is not needed to select the option Remove spikes. Full rubout covers Remove spikes process. If you change your mind after performing Remove spikes this will not create any problem. Simply the time used for the Remove spikes will be a wasted time.

Examples:

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

X-Serpentine:



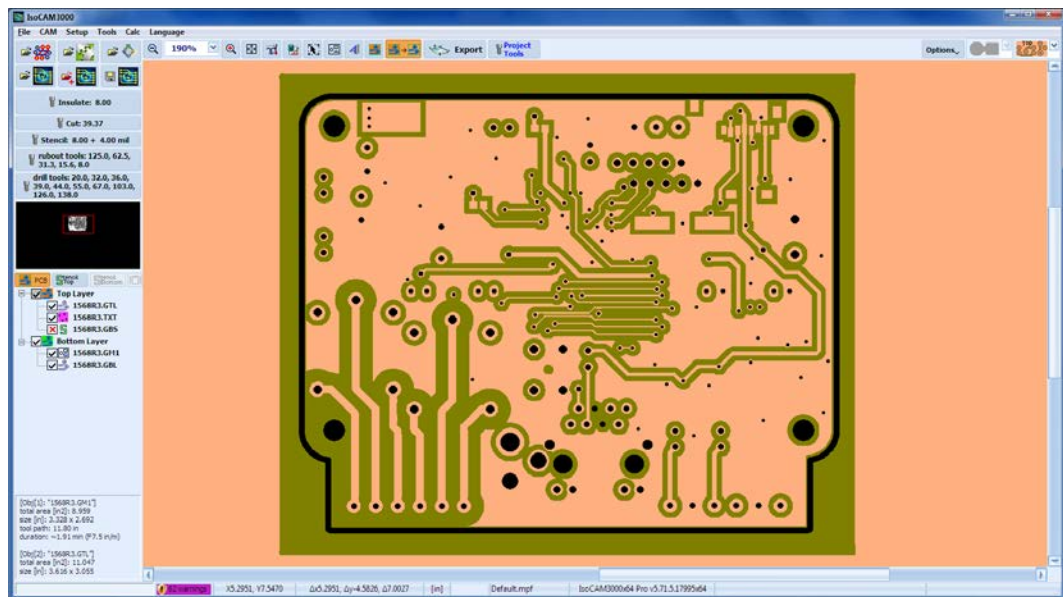
**Y-Serpentine:****Conical:****Notes:**

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

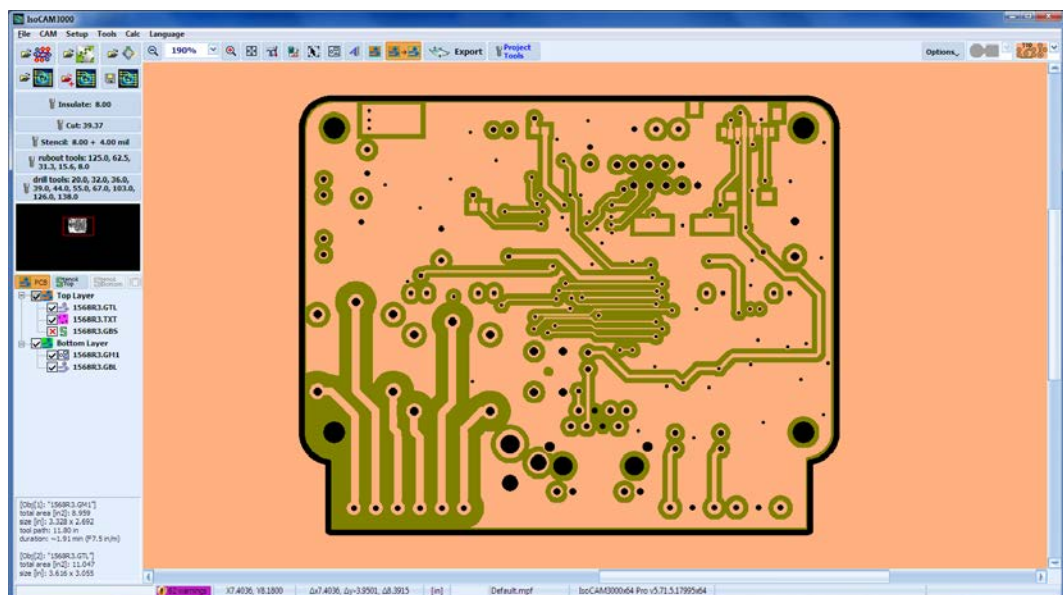
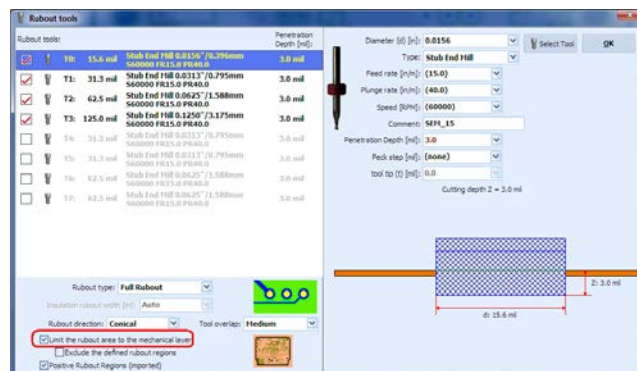
"Conical" is more efficient for designs that have "poured copper" or when "Insulation rubout" has been used.

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

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

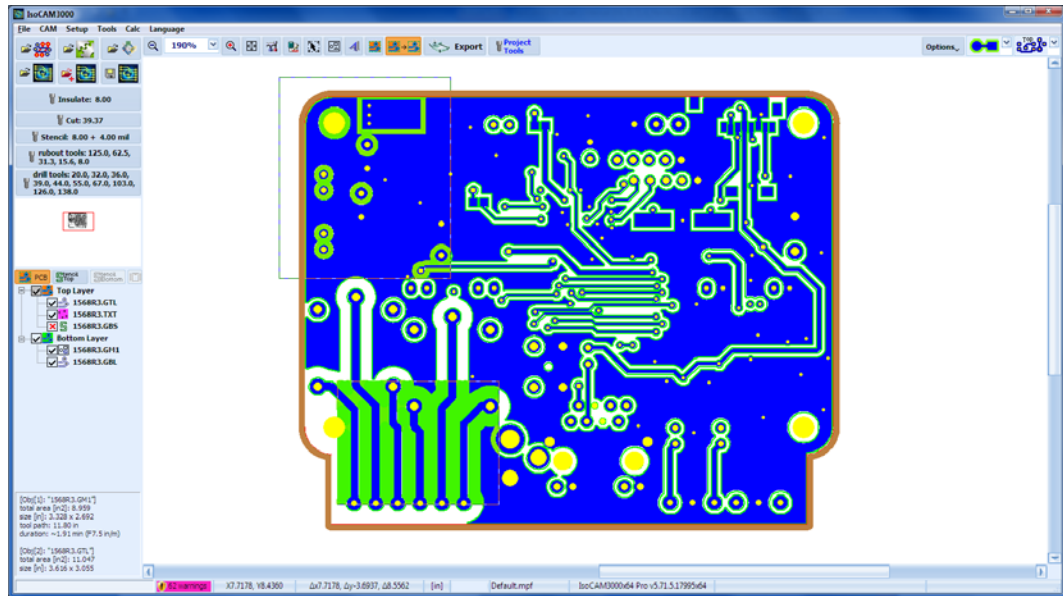


When "Limit the rubout area to the mechanical layer" is checked, the mechanical layer is used like rubout region:

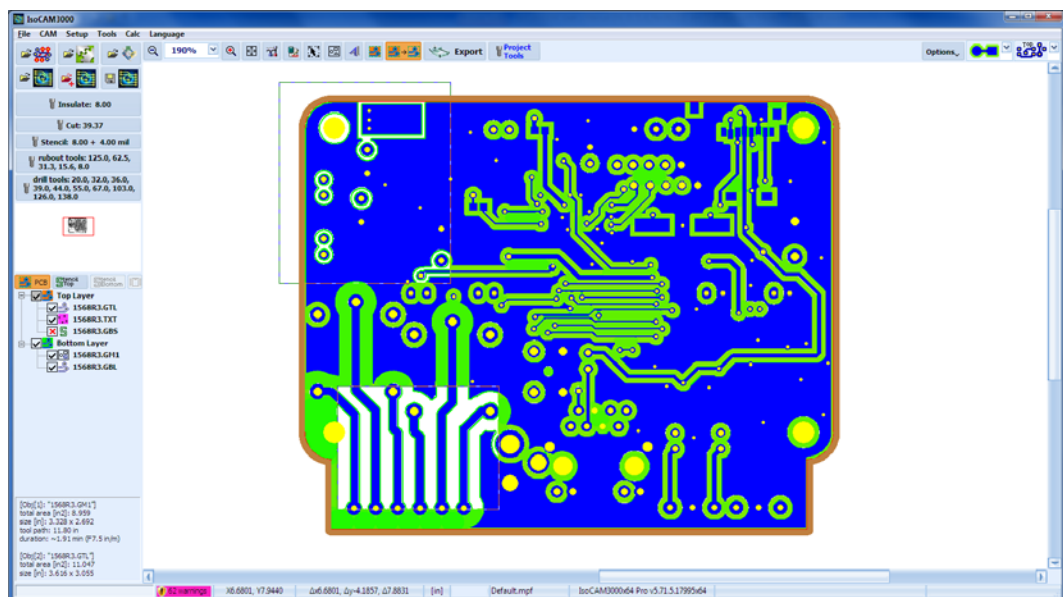
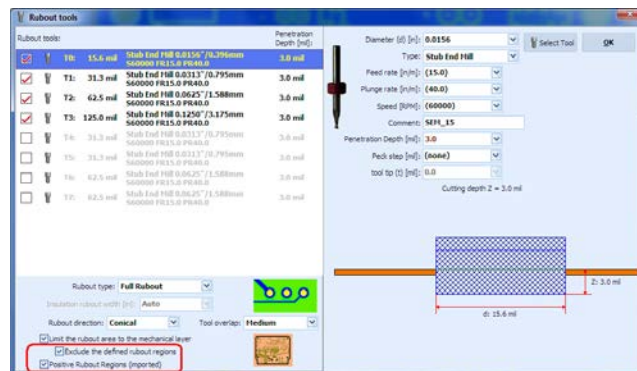


The option "Exclude the defined rubout regions" has meaning, only when "Limit the rubout area to the mechanical layer" is checked.

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

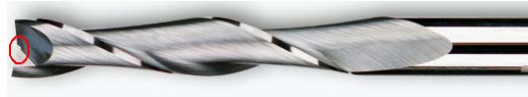


Example, when "Limit the rubout area to the mechanical layer" is checked and "Exclude the defined rubout regions" is checked:

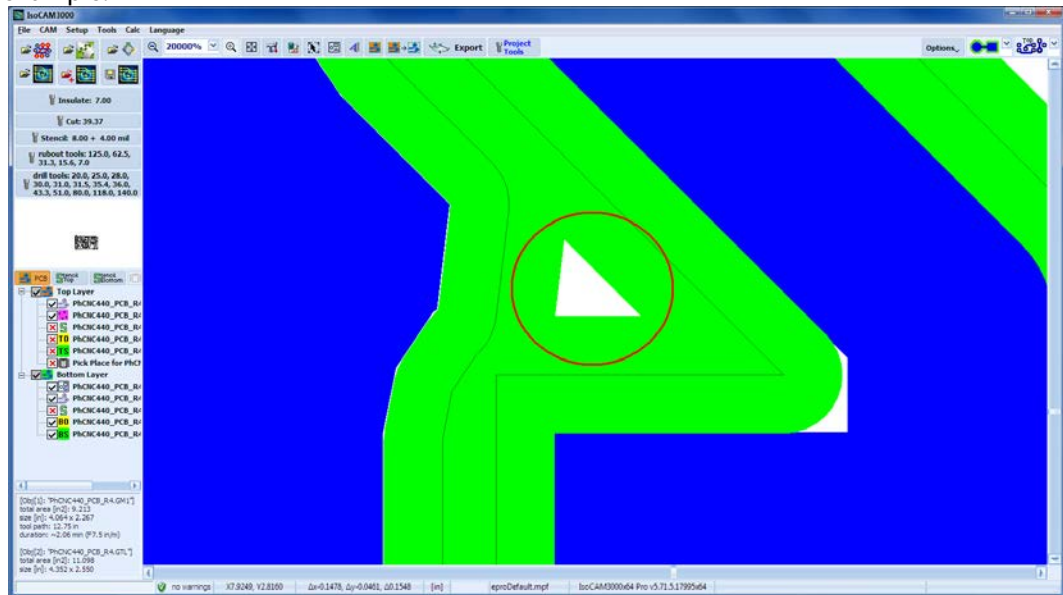


"Fish-tail SEM optimizations" menu option

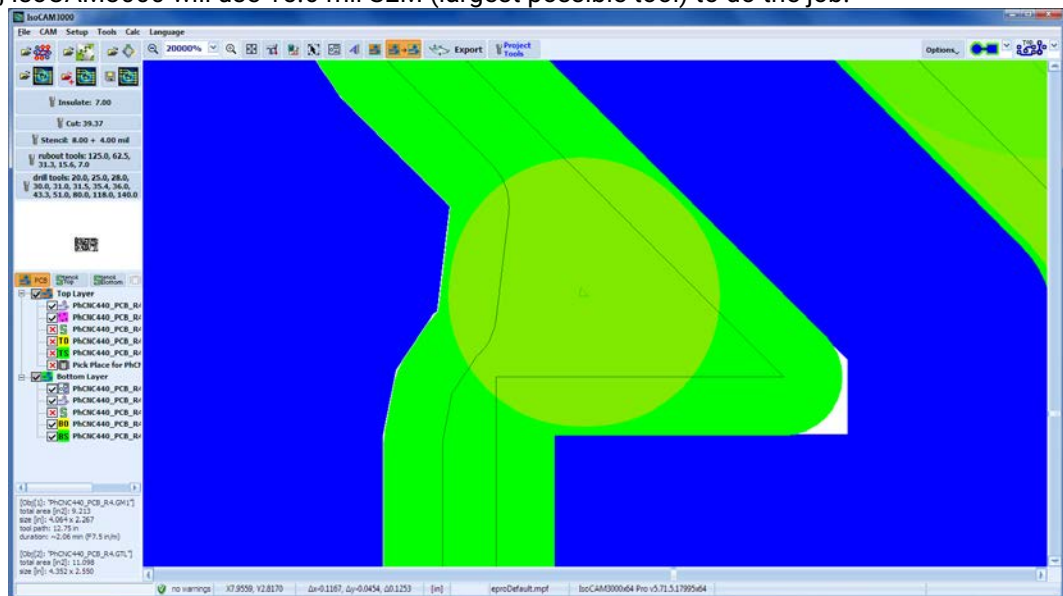
IsoCAM3000 always is trying to use the largest possible tool for insulation & rubout programs, but in case of using Fish-tail Stub End Mills (SEM, shown below) because of its geometry this can create some problems.



One rubout example:

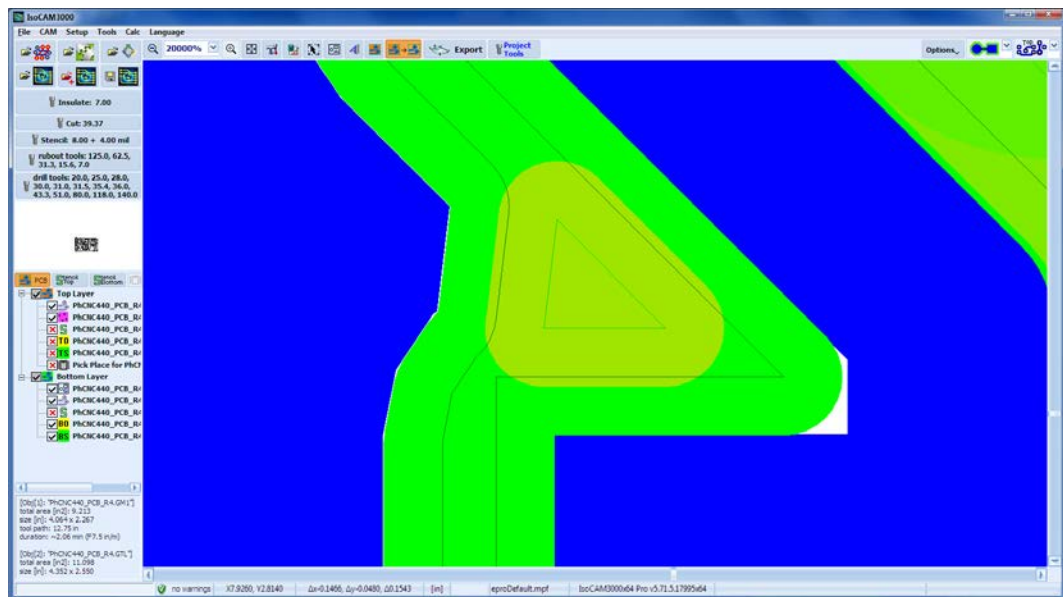


In this case, IsoCAM3000 will use 15.6 mil SEM (largest possible tool) to do the job:



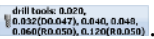
Because of fish-tail geometry - in the center of the very small regions is possible the copper to stay (Fish-tail tool needs movement of at least one R to work like standard SEM).

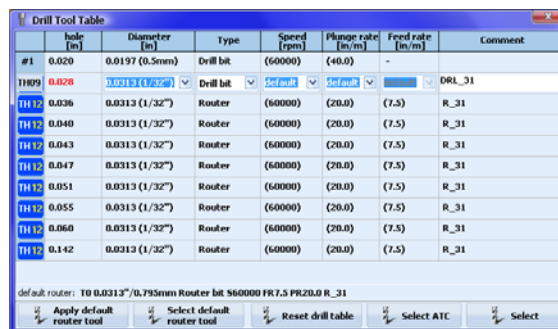
To avoid such problems, IsoCAM3000 will use smaller tool for the problem regions:



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

2.4.4 "Drill" Tool Table

To select the Drill tools table left click on .



	Hole [in]	Diameter [in]	Type	Speed [rpm]	Plunge rate [in/m]	Feed rate [in/m]	Comment
#1	0.020	0.0197 (0.5mm)	Drill bit	(60000)	(40.0)	-	
H109	0.028	0.0313 (1/32")	Drill bit	(60000)	(20.0)	(7.5)	DRILL_31
H112	0.036	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
H112	0.040	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
H112	0.043	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
H112	0.047	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
H112	0.051	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
H112	0.055	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
H112	0.060	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31
H112	0.142	0.0313 (1/32")	Router	(60000)	(20.0)	(7.5)	R_31


default router: TO 0.0313"/0.793mm Router bit S60000 FR 7.5 PR20.0 R_31


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

You must select a tool for each hole size. In some cases, the operator may select to drill the hole using router bit. To use this option, you have to enter a router bit with diameter smaller than the hole. The example above shows what happens if you select a drill bit larger than the hole. The 2nd tool in the tool 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 Router related to usage of router tool for drilling of larger holes. If your project has a small number of larger holes, it is more efficient to drill them using router tool, thus saving tool expenses and tool changes during drill operations.

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

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

Button  resets all tool selections.

**Important:**

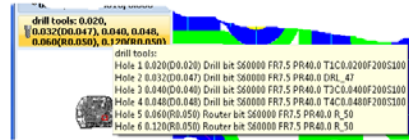
All tool tables and selections will be saved in the project files (.PHJ).

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

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

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

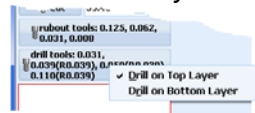
Drill tools table usually is specific for each project. Therefore, you have to select your tools and fill out the drill tools table for every new or imported project.



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

Note:

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

**2.4.5 "Stencil" Tool (Pro version only)**

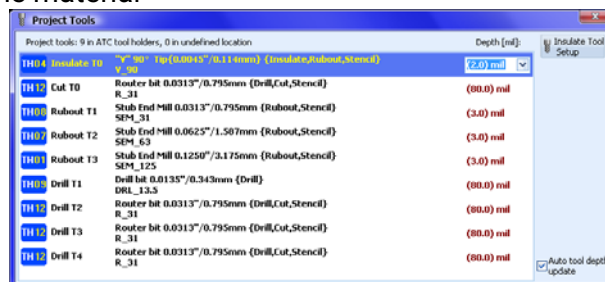
"Stencil" button allows you to select a tool and set the parameters for it. For more details see 2.4.1.

2.4.6 Project Tools

The window "Project Tools" contains a list with all used tools in the project (for all operations).

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

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



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

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

Tool penetration values are entered as positive values to cut into the material. A negative value will not cut the material and will display a warning message before you start this is handy for test runs. A value of Zero "0" will recall the default value for this operation or tool. For changes of tool type, you may use

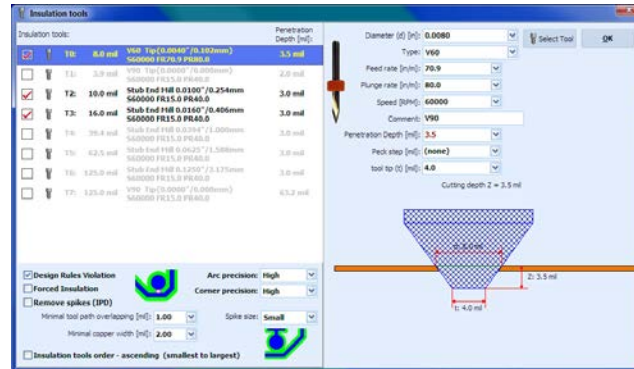


the button “Xxxx Tool Setup” (where Xxxx is the type of operation). For example, you will be able to change your “V” insulation tool with a small stub end mill.

Some examples of tool selection:

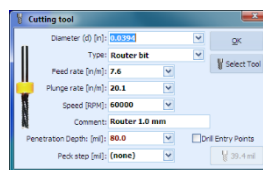
“Insulation Tools”

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

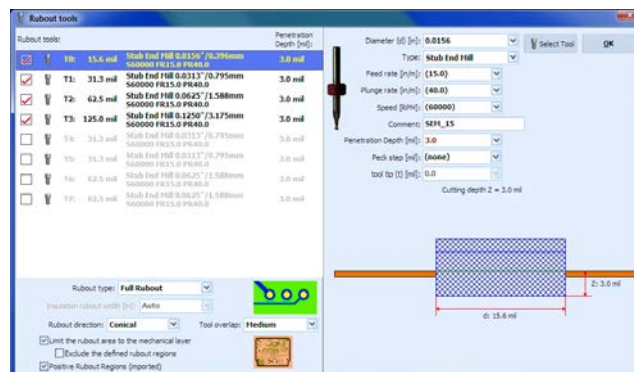


If you have (or already arrange) the necessary tool in the machine magazine simply click on “Select Tool ATC”, it will show you only the tools that are currently in your machine magazine. Using similar technique you can make your selection for the tools for other operations.

“Cut Tool”



“Rubout Tools”



“Drill Tools”

Tool	Diameter (in)	Type	Speed (rpm)	Plunge rate (in/m)	Feed rate (in/m)	Comment
#1	0.0200	Drill bit	(60000)	(40.0)	-	T1F00500C.02000
#2	0.0250	Drill bit	(60000)	(40.0)	-	T2F00500C.02500
#3	0.0280	Drill bit	(60000)	(40.0)	-	T3F00500C.02800
#4	0.0300	Drill bit	(60000)	(40.0)	-	T4F00500C.03000
#5	0.0310	Drill bit	(60000)	(40.0)	-	T5F00500C.03100
#6	0.0315	Drill bit	(60000)	(40.0)	-	T6F00500C.03150
#7	0.0354	Drill bit	(60000)	(40.0)	-	T7F00500C.03543
#8	0.0360	Drill bit	(60000)	(40.0)	-	T8F00500C.03600
#9	0.0433	Drill bit	(60000)	(40.0)	-	T9F00500C.04331
#10	0.0510	Drill bit	(60000)	(40.0)	-	T10F00500C.05100
#11	0.0800	Drill bit	(60000)	(40.0)	-	T11F00500C.08000
#12	0.1180	Drill bit	(60000)	(40.0)	-	T12F00500C.11800
#13	0.1400	Drill bit	(60000)	(40.0)	-	T13F00500C.14000

default router: ☐ not defined ☒ Enable Harking Drill V90 Tool 0.0118" / 0.300mm V90 S40000 Harking Drill Tool P640.0 V_90


Apply default router tool

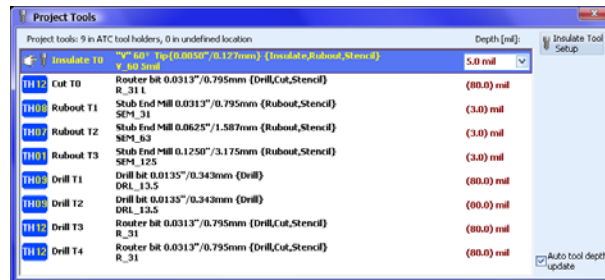
Note:



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 IsoCAM3000 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.

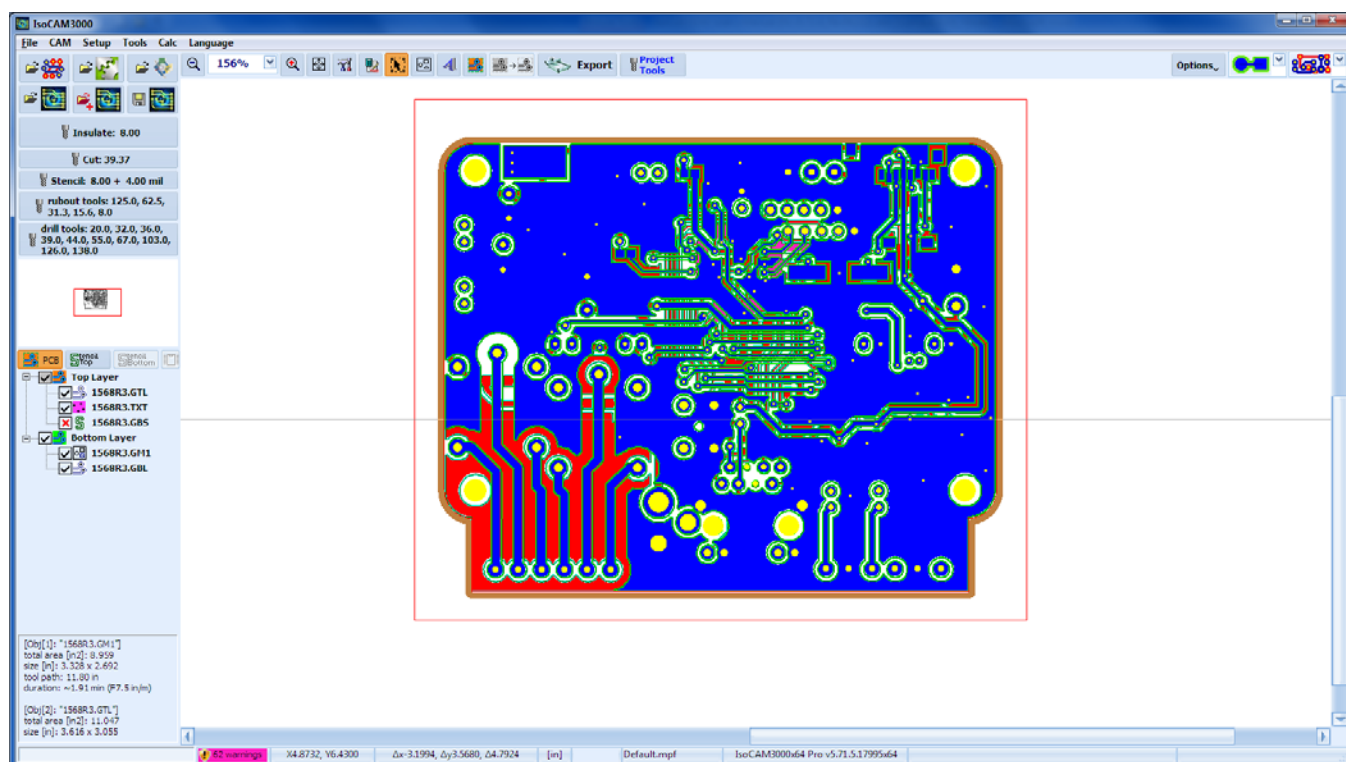
Important:

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

IsoCAM3000 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.

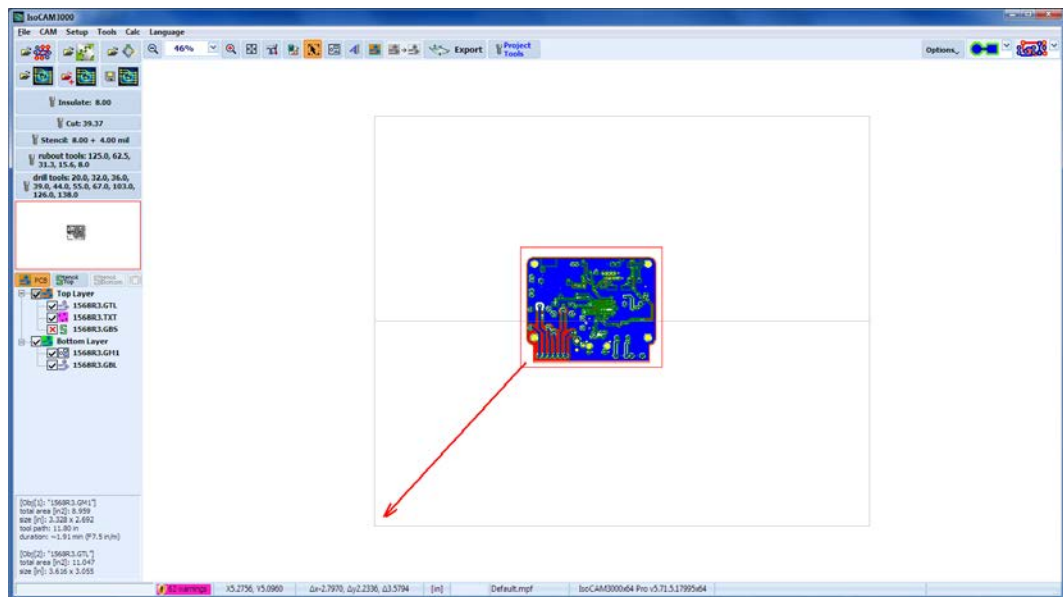
2.6.1 Moving objects using the mouse

In CAM and CNC mode of IsoCAM3000, you can push and hold the left mouse button. Mouse movements now will scroll the screen (convenient option, especially when you have high zoom).

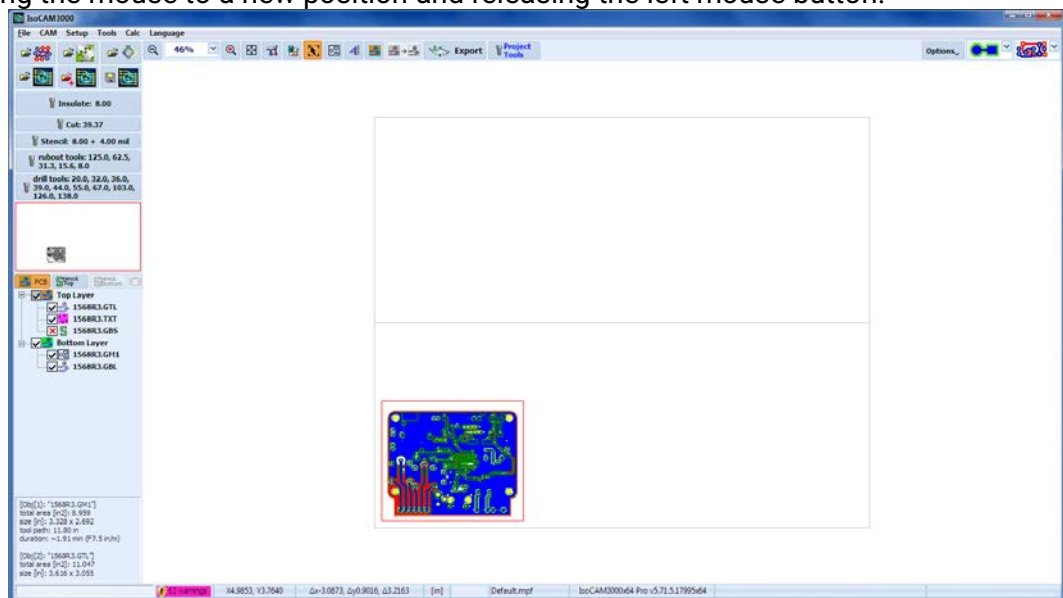
In CAM mode left clicking and holding the left mouse button on a selected object(s) will move the object(s) together with the mouse cursor (known as click and drag).

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.

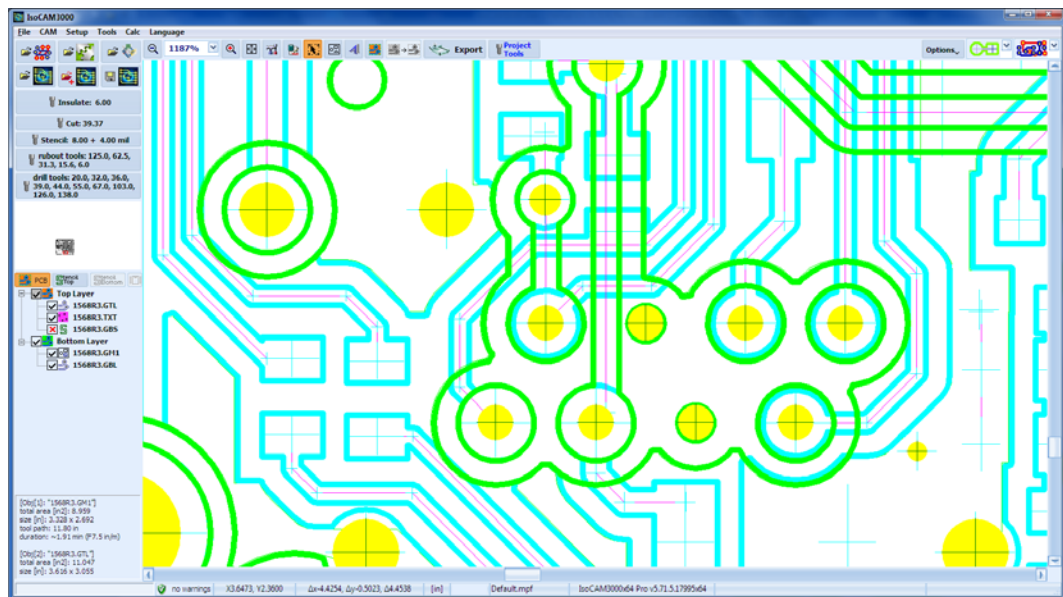


After moving the mouse to a new position and releasing the left mouse button:



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).

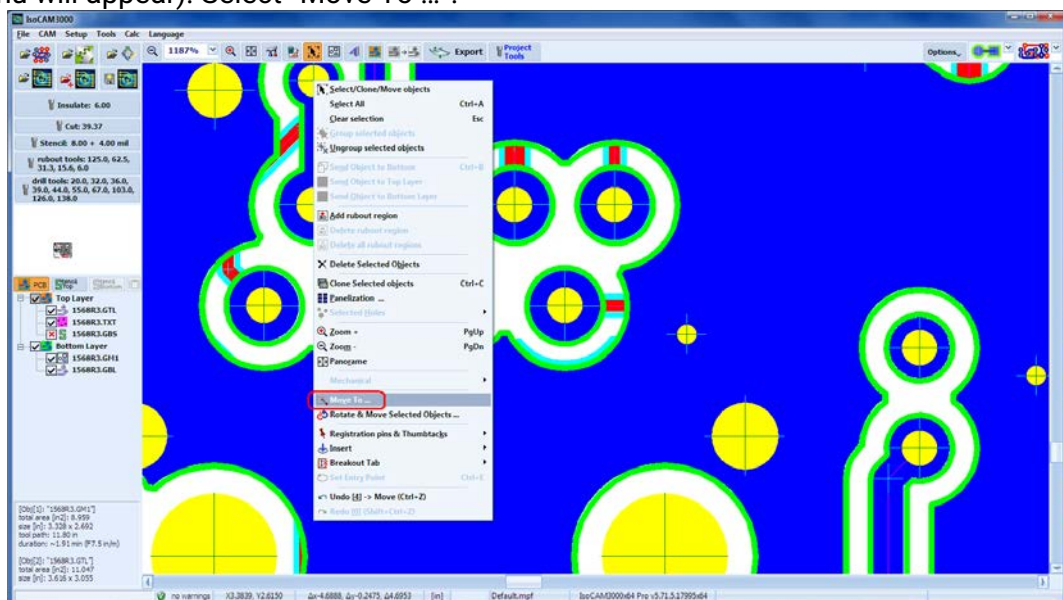


2.6.2 Move using “Rotate & Move Selected Objects ...”

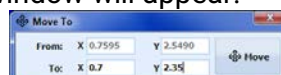
This is the best mode when you need to move an object(s) to a predetermined distance. Described in paragraph 2.8.

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

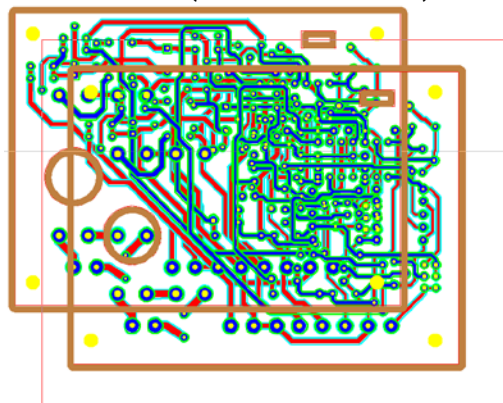
- Select an object or group of objects
- Select "Clone Selected objects" from popup menu

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.

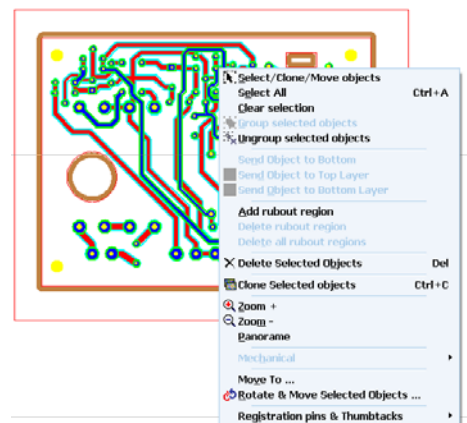
To move a cloned object, click, hold, and drag the object.

Note:

See also the Panelization (2.29 Panelization)!



Cloned object

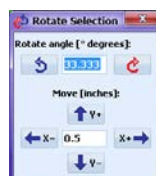


2.8 Rotate objects

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



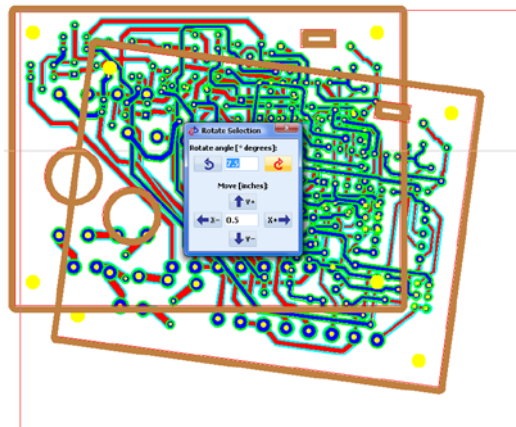
Rotate Selection Window will appear:



You may set any angle and distance.

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

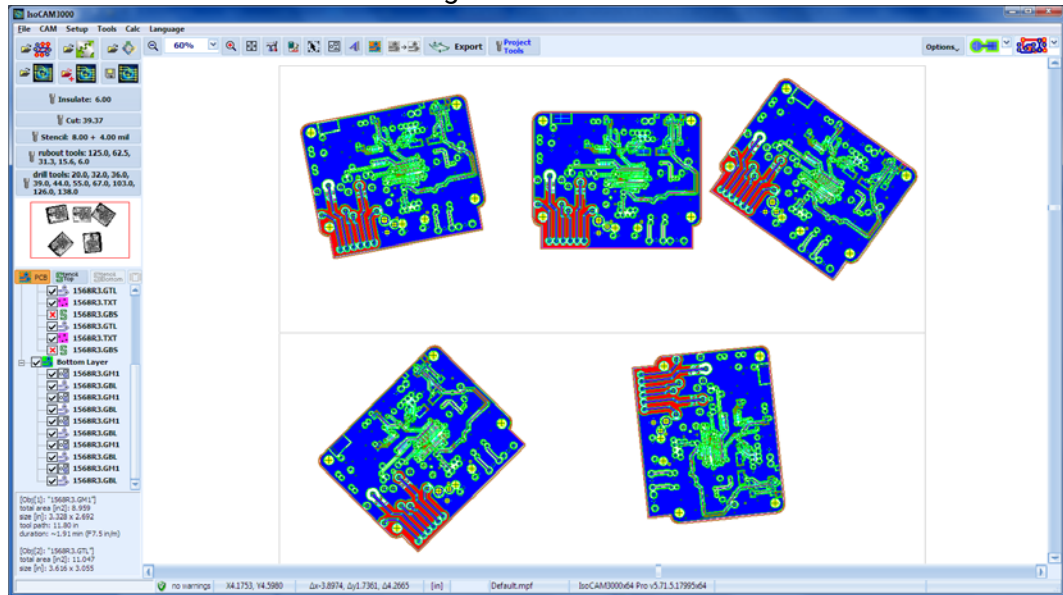
For example, the object below has been rotated at 7.5° CW



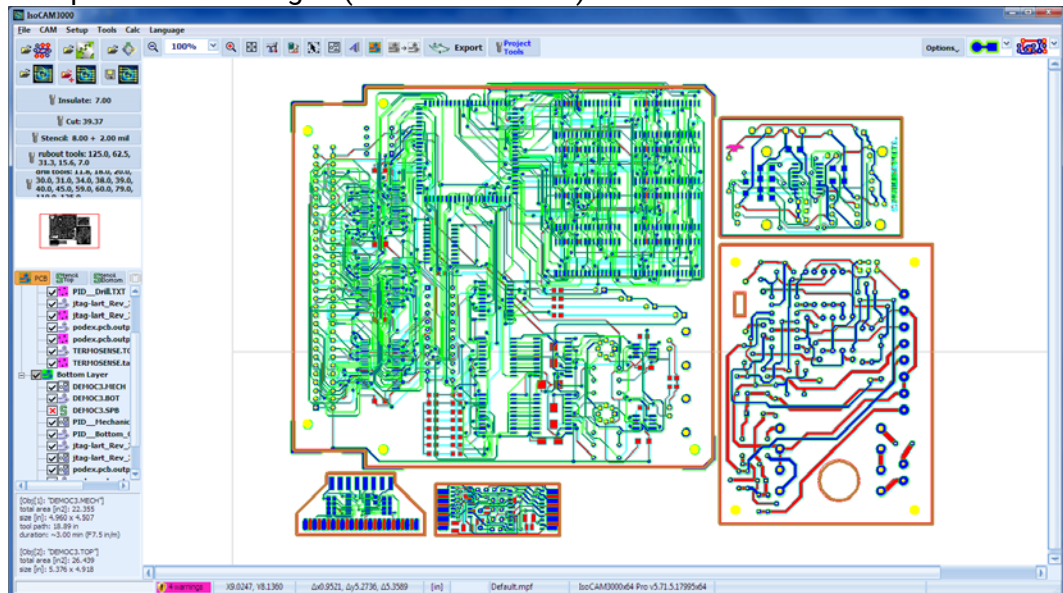
Example 2:



Five copies of a board rotated at different angles:



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



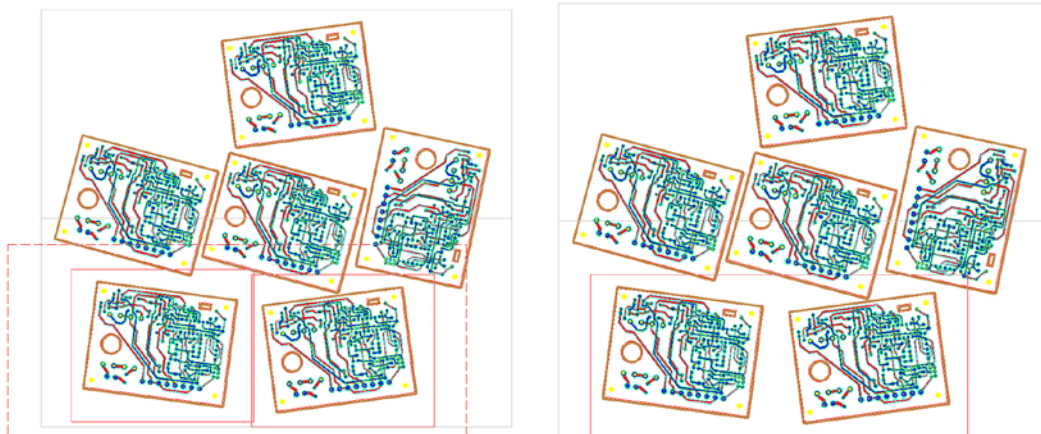
F_A5.PHJ

2.9 Group/ungroup objects

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



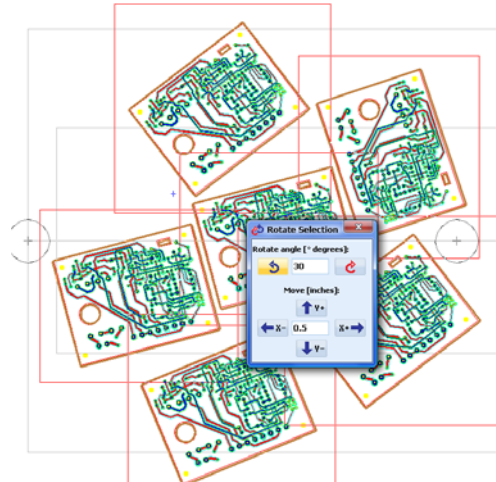
All grouped objects behave as a single object and they maintain their relative position during further manipulations.





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

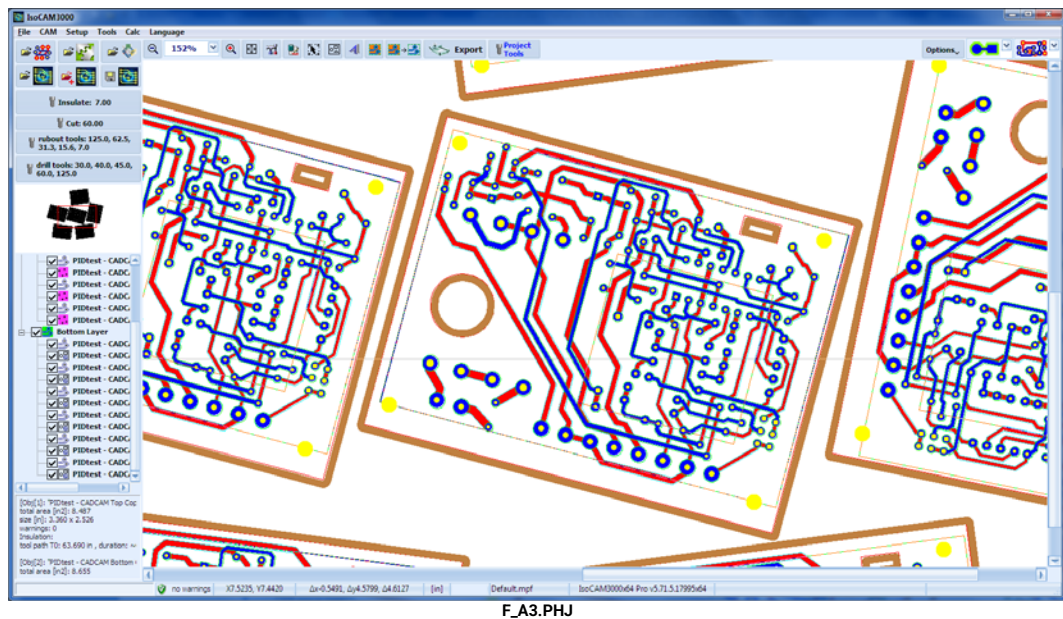
To delete objects, select them, and then click on “Delete Selected objects” from the popup menu.

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.

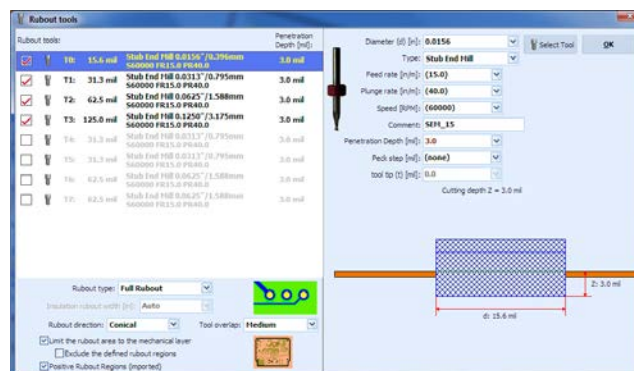


Example of rubout region creation (short dash line):



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

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

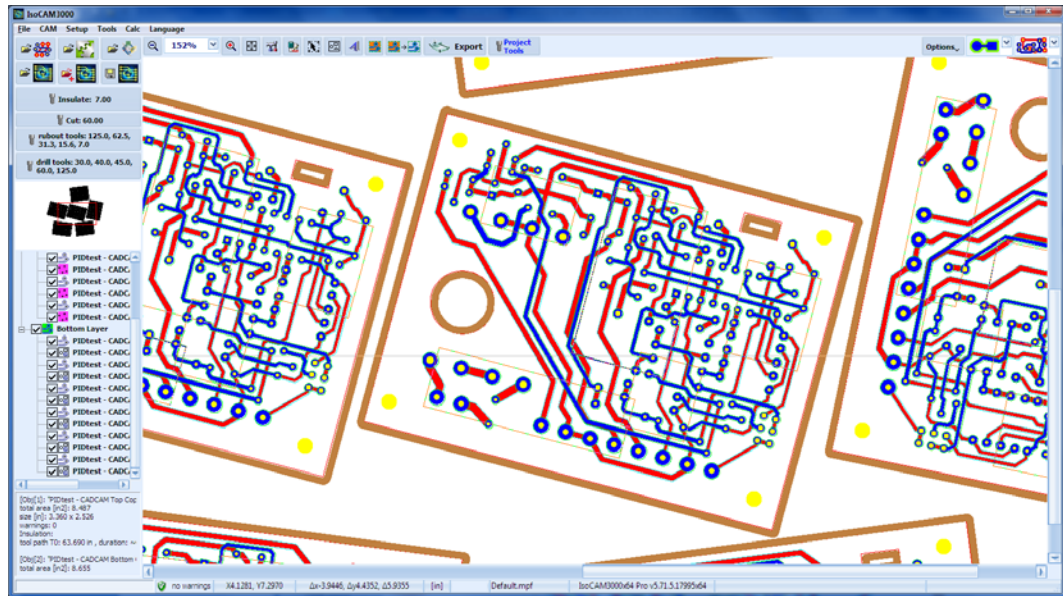


The first and the smallest tool in the rubout process will always be the insulation tool. The rest depends on your design and you can create any combination you want. Enter 0.0 for the positions that will not be used (or press "Delete"). Enter the right sizes for the tools you will use. Rubout will sort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


Important:

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

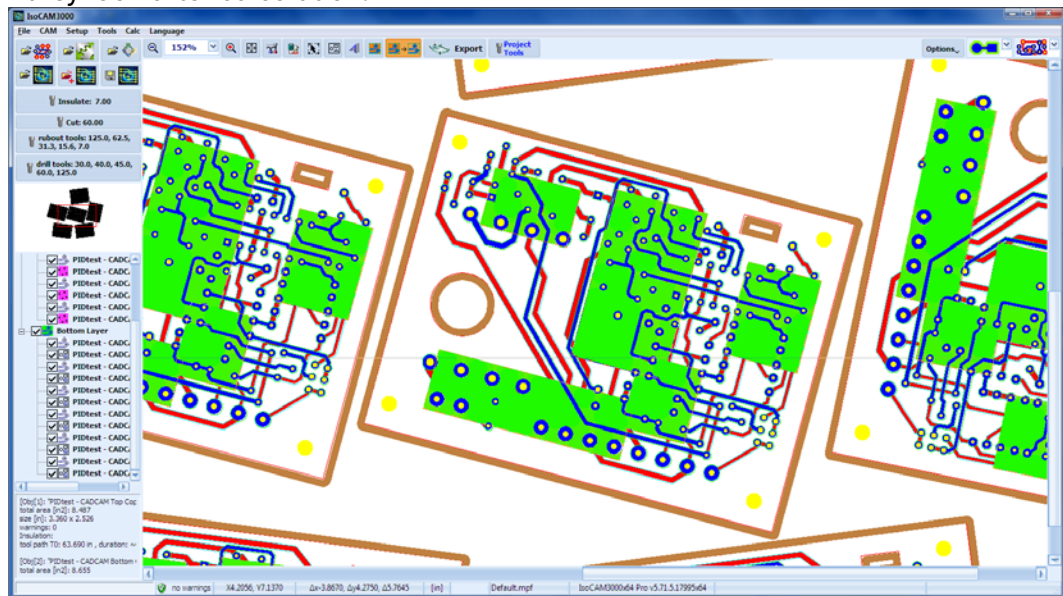
In the example below 5 regions are created:



F_A4.PHJ

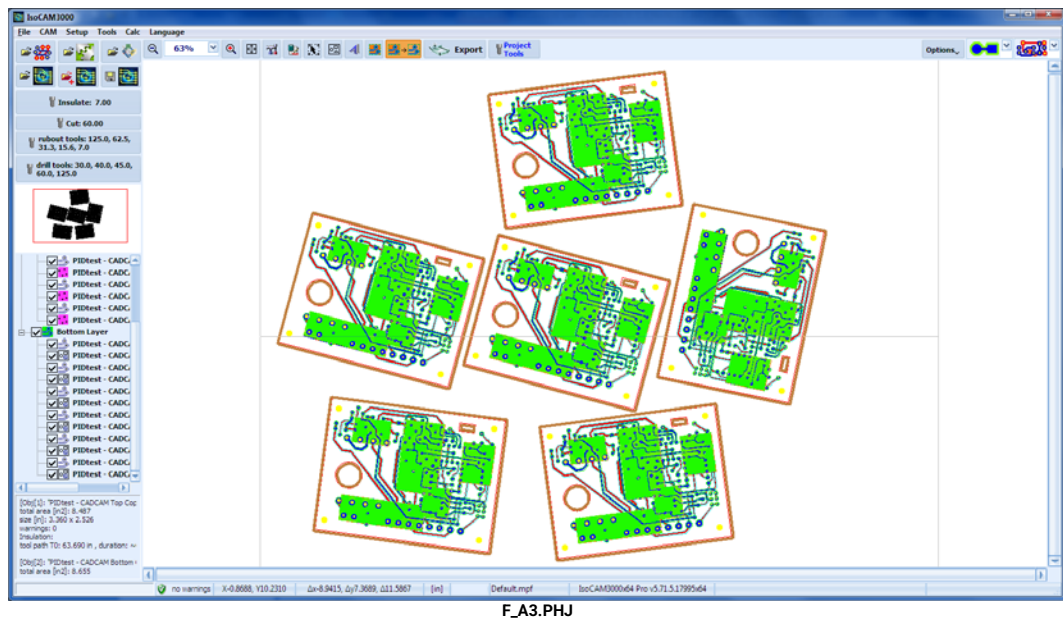
To calculate regions, click on the  button. First click will calculate the “Rubout regions” and all consequence clicks will toggle between hide/show them on the screen (v1.17.5+)

Here is how they look after calculation:



F_A4.PHJ

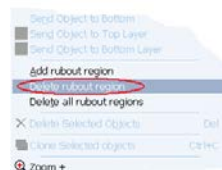
Here is the example above spread to all copies:



F_A3.PHU

If for any reason you do not calculate the regions before entering CNC mode, IsoCAM3000 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.




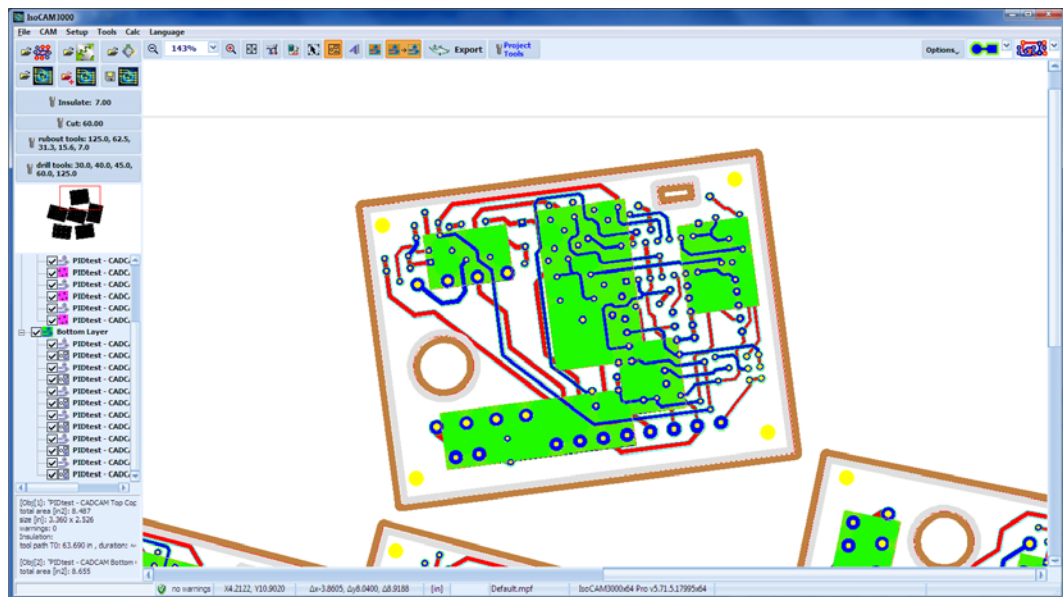
If you would like to delete all rubout regions click on "Delete All rubout regions" from the popup menu.

2.12 Editor for the mechanical layer

2.12.1 Contour cutting selections


If you have loaded a mechanical layer for contour cutting, IsoCAM3000 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:



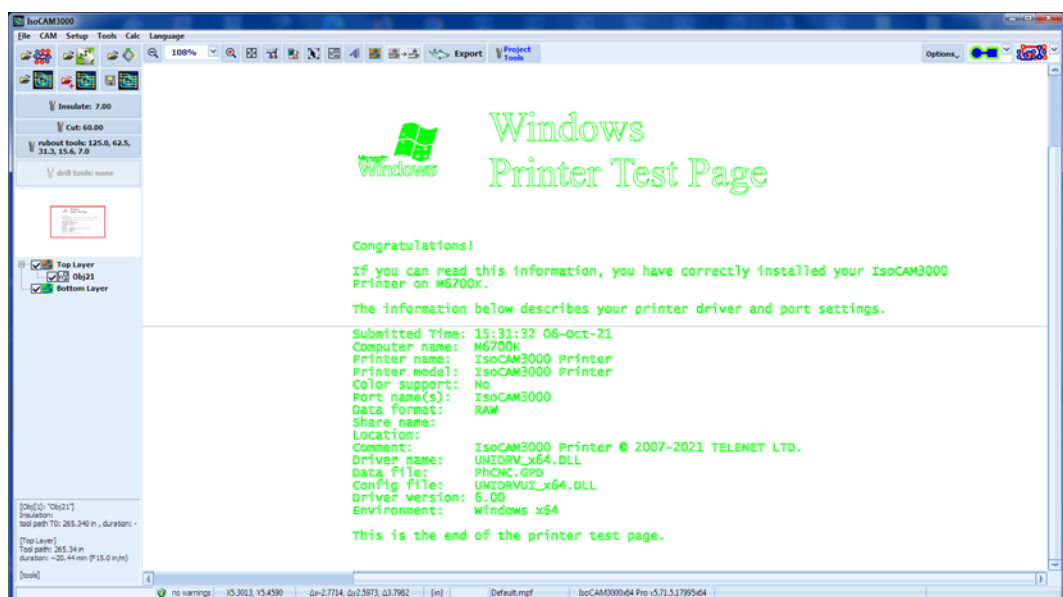
Selected contours are in light brown and the possible, but not selected, opposite compensation contours are in light gray. You may change the selection with a mouse click on the contour. When you leave contour cutting selection mode, unselected contours will become invisible and will not be executed.

2.12.2 Contours selection in the mechanical layer

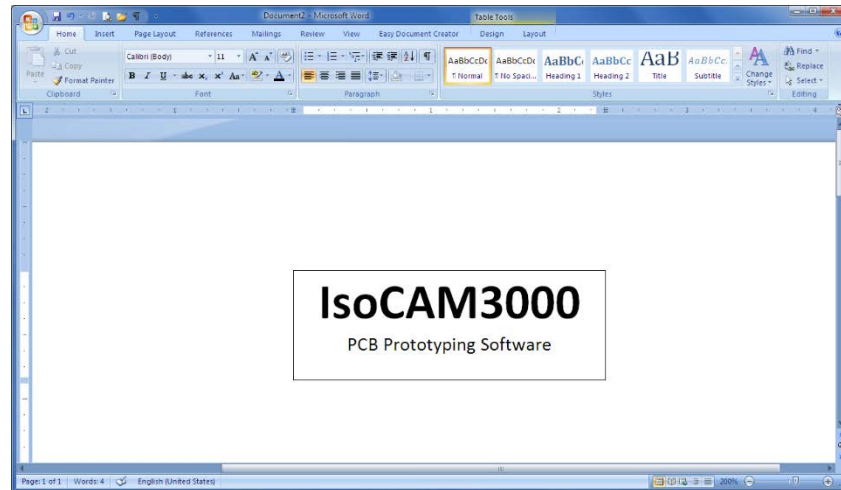
In this mode, you can perform the following operations with the contours generated by “IsoCAM3000 Printer” and  (import of text and graphics).

2.12.3 Contours generation with “IsoCAM3000 Printer”

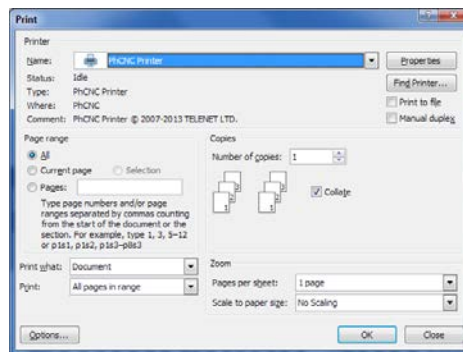
Upon starting IsoCAM3000, the “IsoCAM3000 Printer” driver is installed, which allows you to import vector information in IsoCAM3000 from any application or CAD/CAM system. On the figure is shown the test page of the printer.



Example:
Name plate generated with MS Word:



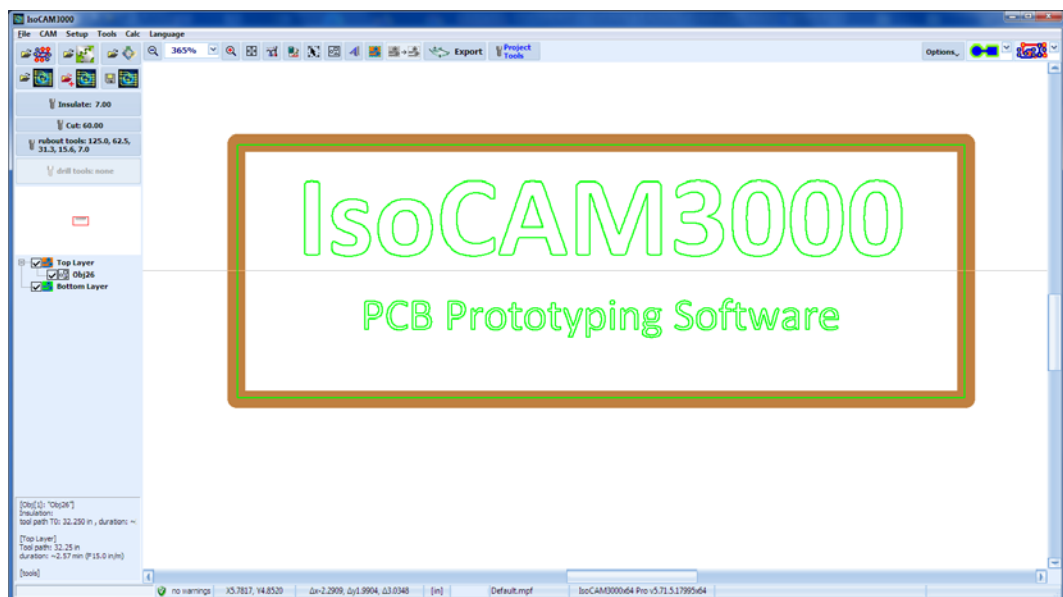
To import a name plate into IsoCAM3000 – simply select the printer “IsoCAM3000 Printer” and press “Print”.




Remark:

IsoCAM3000 must be started (you can also start it after the printing)!

The vector image is imported automatically in IsoCAM3000 (as a separate object):

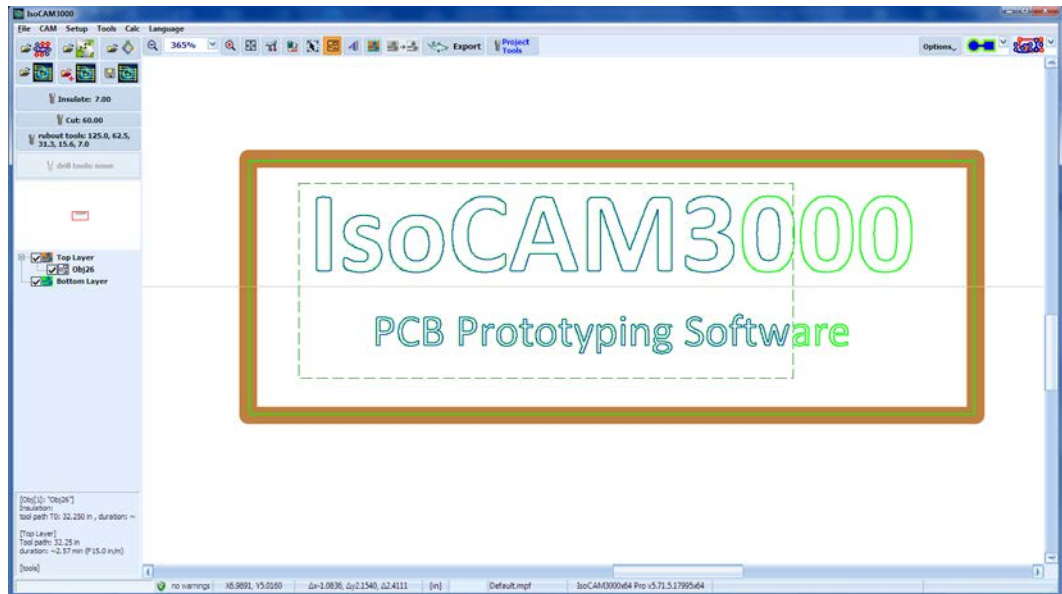


By default, all contours will be processed with the tool for insulation (T0). If you want to select contours for cutout (for example the external contour), enter the mode for editing of mechanical contours by left clicking the  button.

There are two possibilities for selection of contours:



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



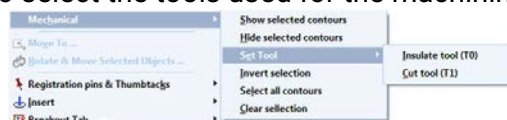
Right clicking on selected contours allows you to enter the “Mechanical” menu.



This menu will allow you to select the following options:

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

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

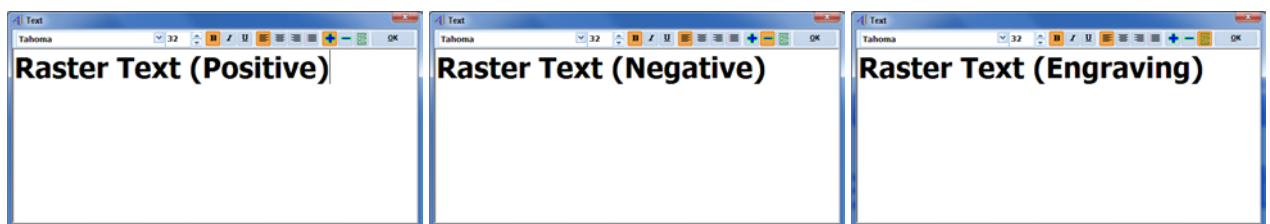


Here, T0 is the current tool for insulation and T1 is the current instrument for cut-out.

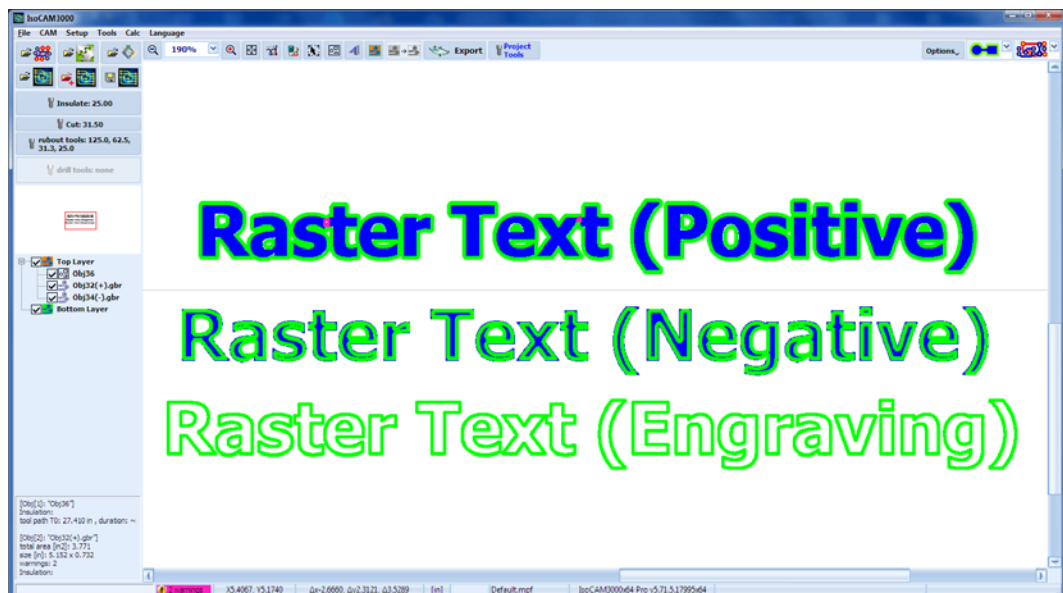
On the figure below, the frame and the inscription IsoCAM3000 will be processed with T1 (cut-out) and the remaining text will be processed with the tool for insulation T0:



2.12.4 Generation of contours with ("Insert Text")

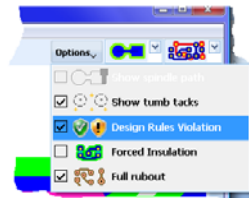



The result is shown on the figure below:

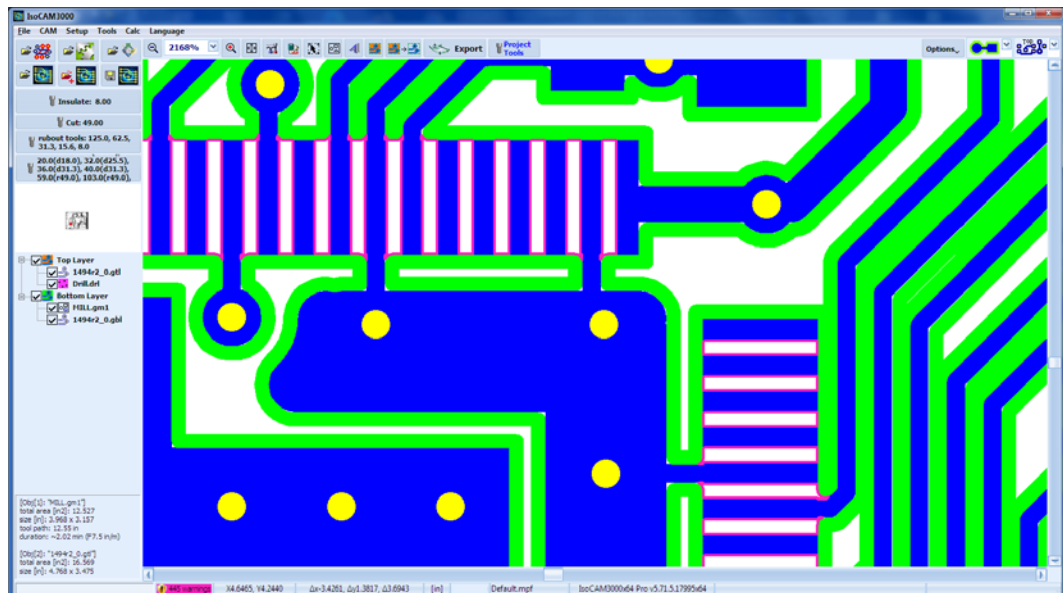



2.13 Design Rules Violation (DRV) detector


IsoCAM3000 utilizes DRV calculations. The main function of DRV is to check all tool paths for compatibility with the selected insulation tool (diameter). If the selected tool is bigger than the distance between the traces/pads, DRV detector will prompt you.




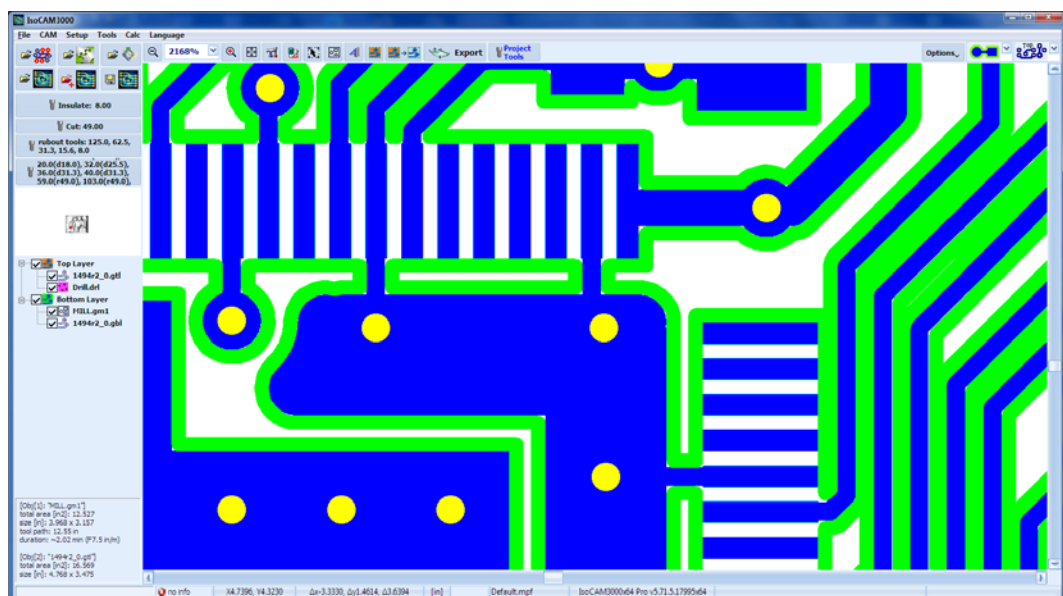
When this option  is checked DRV detector is active. It checks for DRV compatibility for each generated tool path. The DRV spots are presented with easy to see default pink:





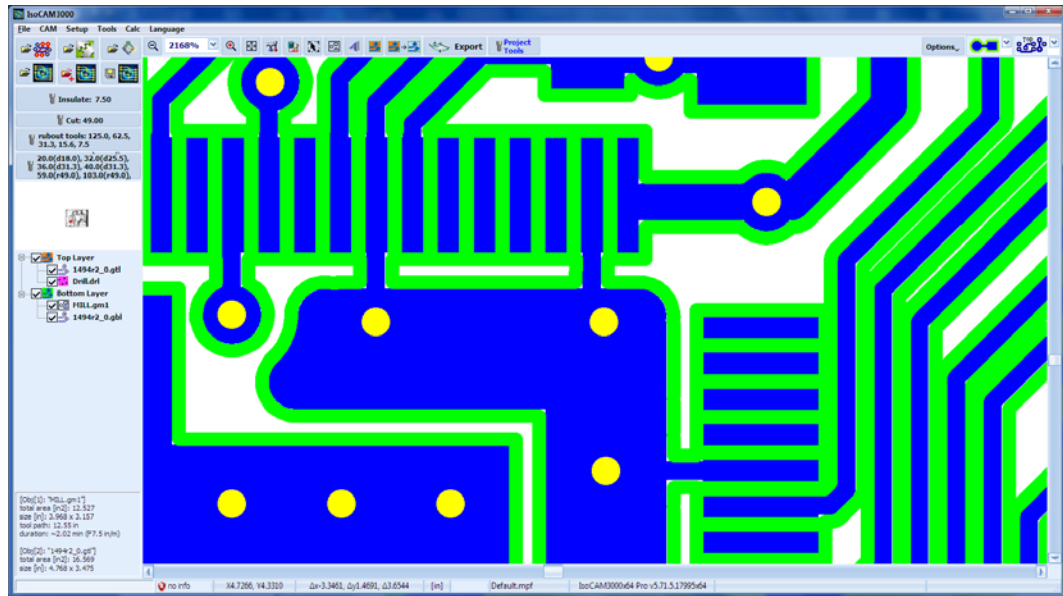
The status bar displays information () about DRVs and their number (for all layers on).

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

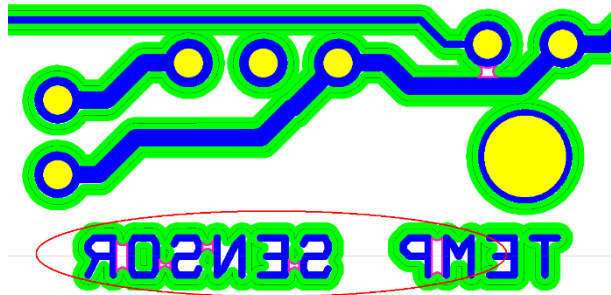
Button  also shows/hides DRV spots. If it is not activated during the tool paths generation, you may turn it on later and DRV detector will check for DRVs.



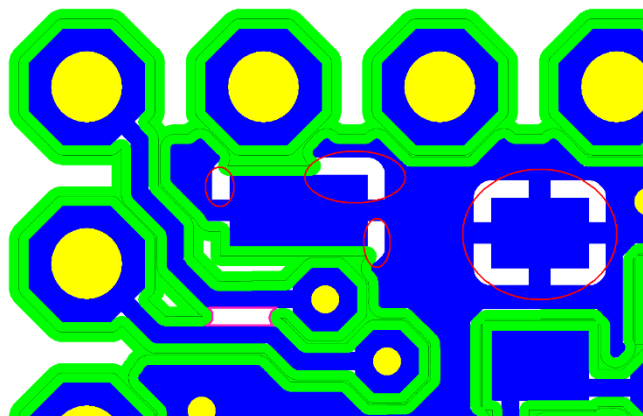
If there are no DRVs the status bar will show the  no warnings indicator. If DRV detector was not activated during the tool paths generation the status bar will show the  no info indicator. If DRVs exist the only practical solution is to reduce the diameter of insulation tool (See 2.23 Forced Insulation).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



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




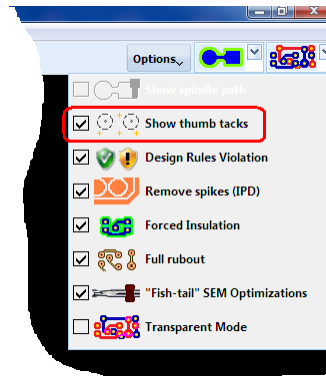
DRV detector will prompt you for shorts (non-insulated area that belongs to two different contours/pads/traces). If the area belongs to the same circuit (has a common contour) it will not appear as a DRV. See the case below (in red circles). The traces belong to the same circuit and are not DRV's.



2.14 Registration Pins and Thumbtacks

IsoCAM3000 allows you to add and display registration pins and thumbtacks visible in CAM & CNC modes of the software.

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

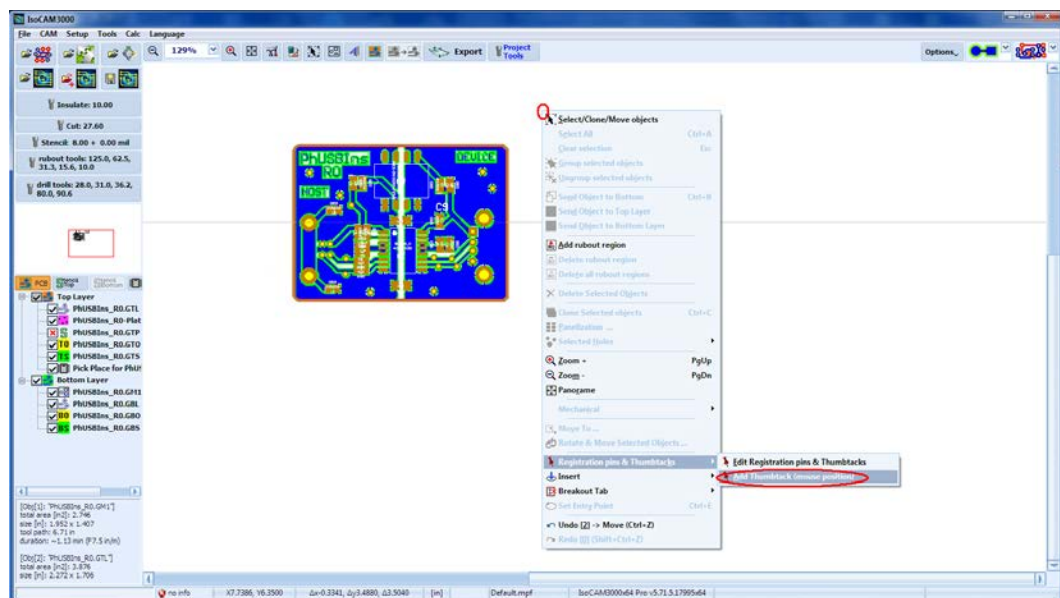


2.14.1 Thumbtacks

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

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

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



2.14.2 Registration Pins

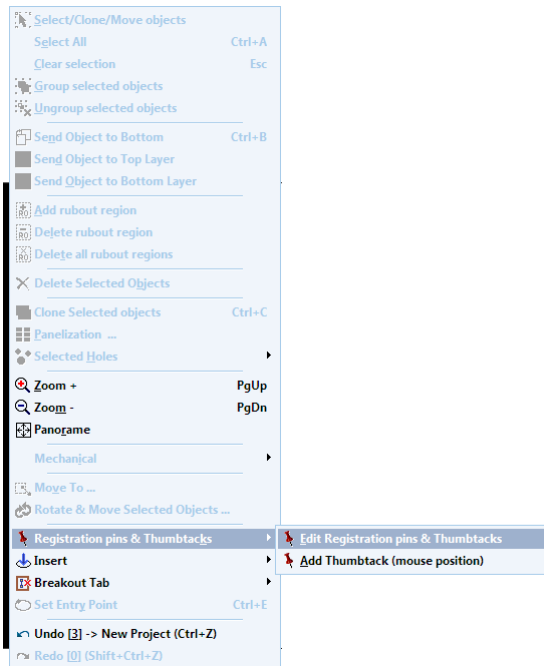
Registration pins show the position of the (normally two) pins that create the mirror axis for double sided jobs.

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

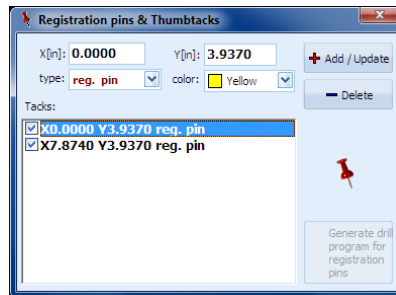
During the setup process you should not place any objects closer (than 0.5 inch) to the registration pins. Using this option, you will be able to see the real relative position between any pin and your design. Around each pin you will see a circle (dashed line) with 1.0 inch diameter that represents the area that should be clear of project objects. The registration pins must be located on the center line in the middle of Y coordinates and strictly parallel to X axis. This is the line that creates a mirror image and tool paths for the machining of the bottom layer (The Y coordinate of the mirror line can be adjusted with special command).



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

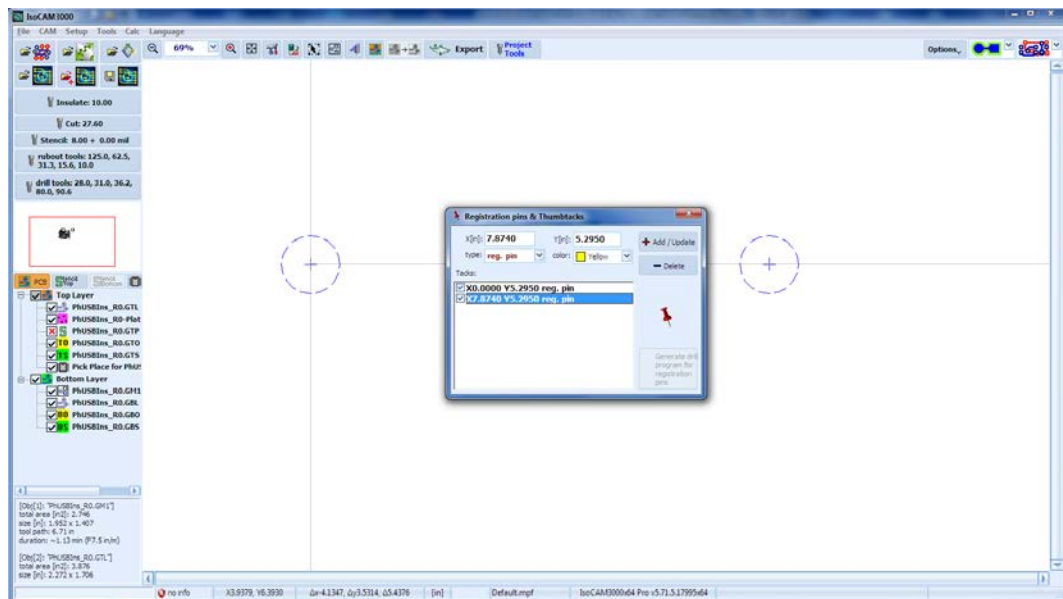


Below is the window that allows you to add/edit registration pins and thumbtacks:



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

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



2.15 Insulating copper rubout.

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

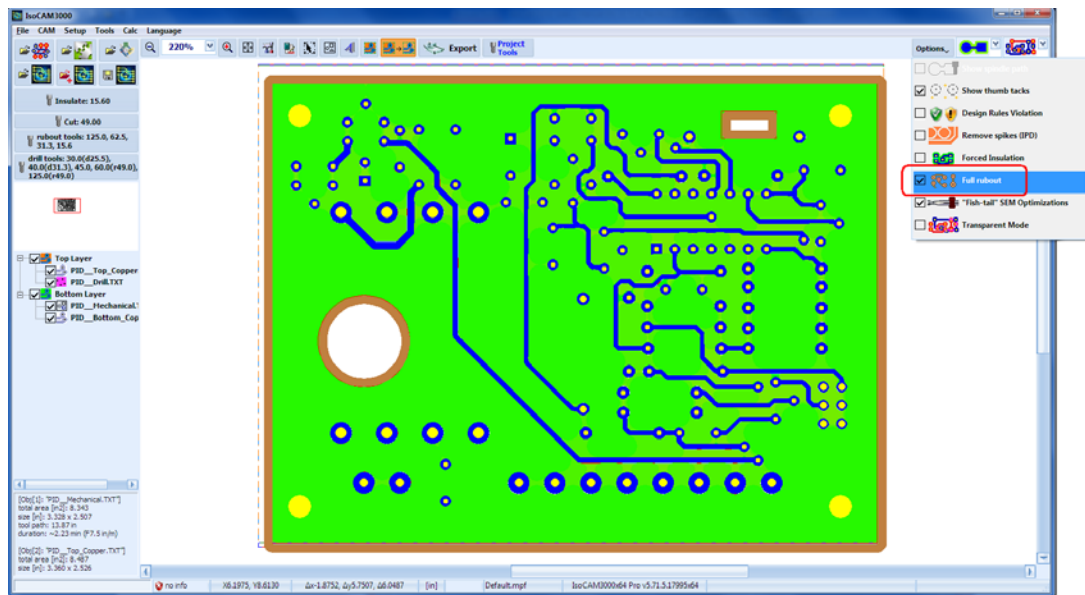
Rubout has two modes:

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

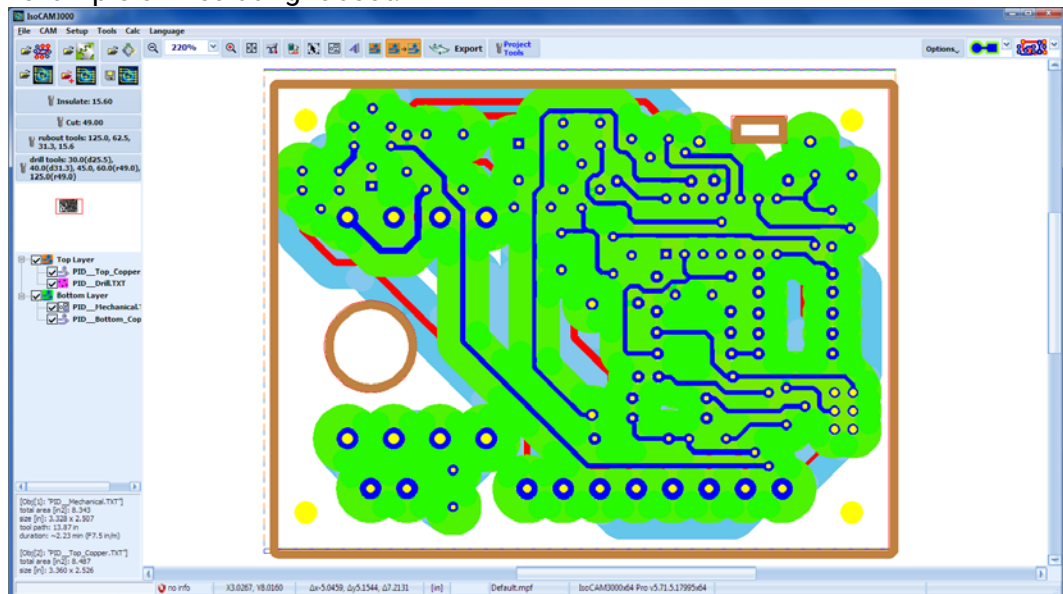
Complete or insulating rubout can be selected by this toggling button

- IsoCAM3000 calculates tool paths for complete rubout
- IsoCAM3000 calculates tool paths for insulating rubout

Below is an example of complete rubout:



Below is an example of Insulating rubout:



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



In the example above 100 mils is selected. This means all pads and traces in the selected region will have an insulation of 100 mils (when possible). The algorithm that creates the tool paths is very



effective and uses the smaller tool only in places that next (larger) tool cannot penetrate (without interfering with design features).




2.16 CAM350 import

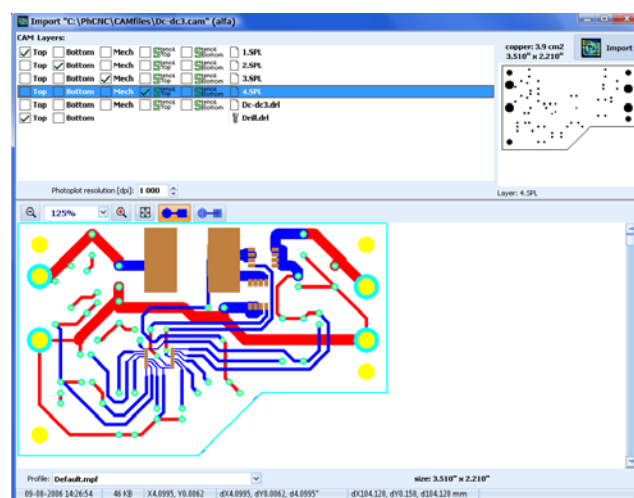
IsoCAM3000 is able to import CAM350 files without the need to export them (from CAM 350) in Gerber and/or Excellon formats.

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.

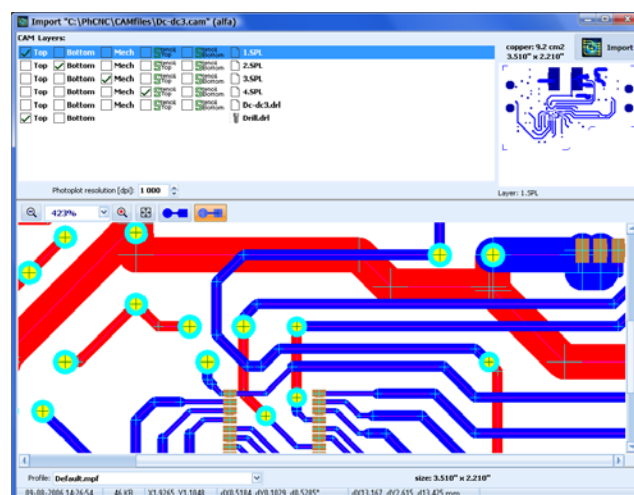
Button  will allow you to select a CAM350 file (.CAM).

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, IsoCAM3000 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

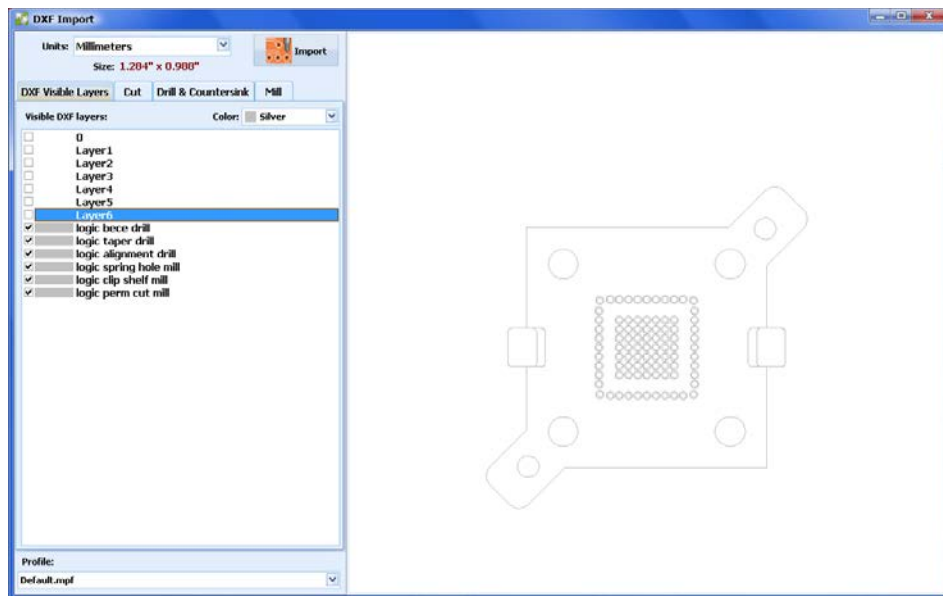
IsoCAM3000 imports 2D DXF files (AutoCAD).



Button  initiates DXF files import in IsoCAM3000. 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 IsoCAM3000.

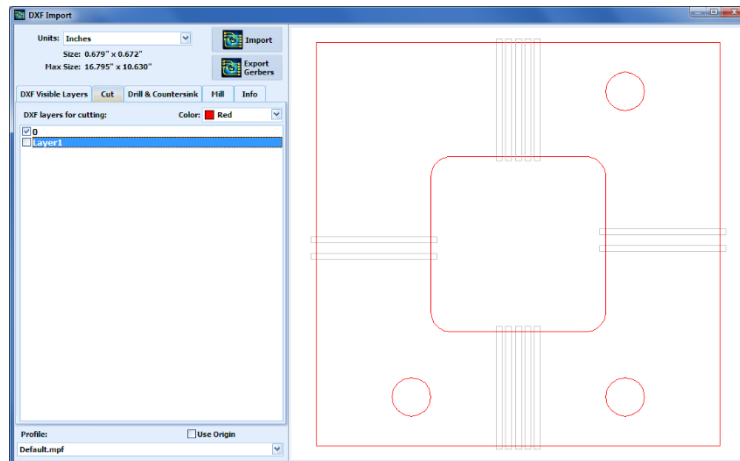
- Each operation (cutting, drilling, countersinking, and milling) must have a separate layer.
- Only closed contours are recognized for machining.
- In a layer that has 2 objects (for example lines) closer than 0.001", it is approximated that they are connected.
- Drill/countersink information is extracted 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.





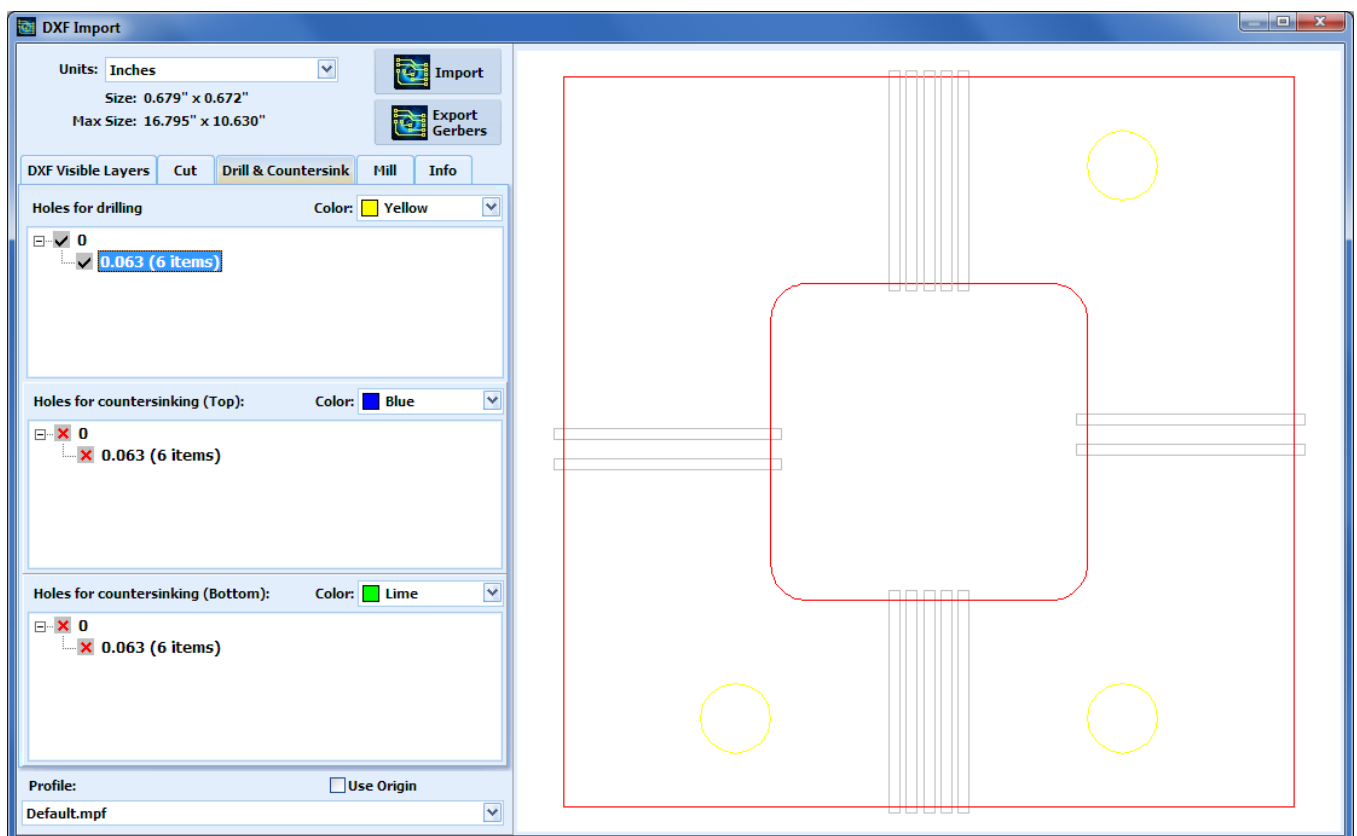
2.17.1 Selecting DXF layers for “Cut” operation



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.

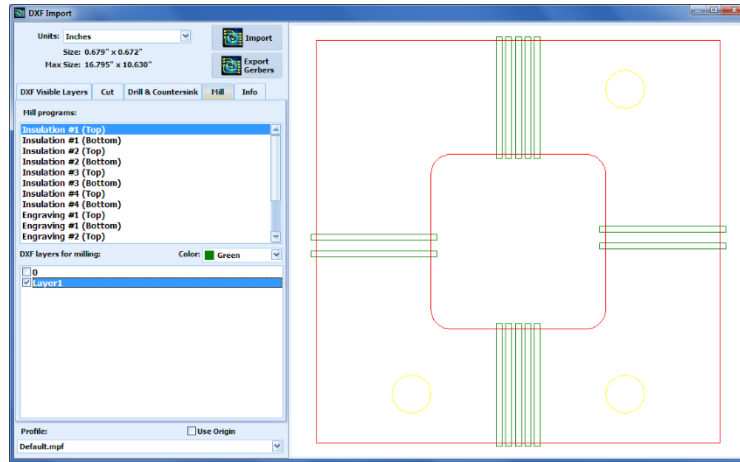
- Drill
- Countersink Top Layer
- Countersink Bottom Layer
- Milling
- Cutout



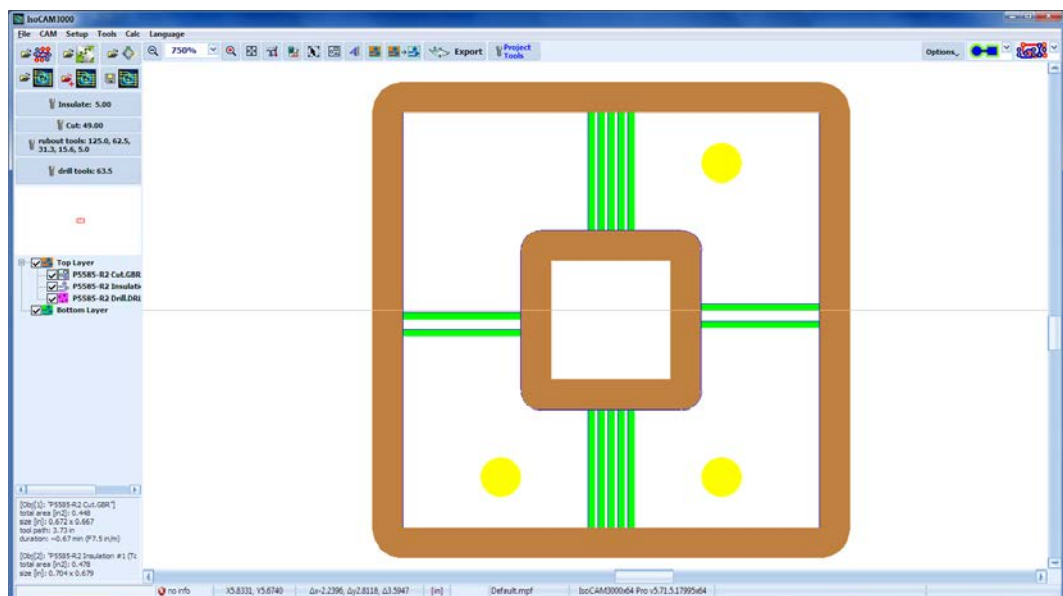
To assign a layer to desired operation replace the red “x” mark with check mark.

2.17.3 Selecting DXF layers for milling (MILL)

IsoCAM3000 is able to mill closed contours:



Example of an imported DXF file:

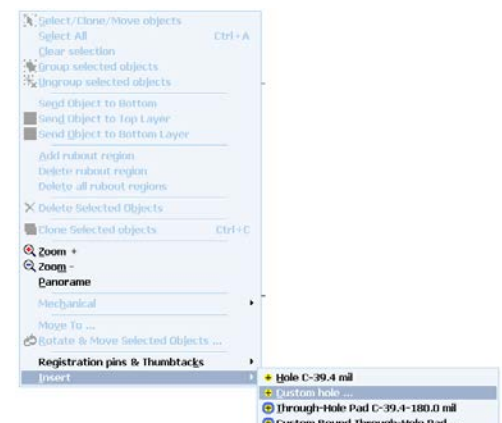


2.18 Holes and pads insertion

CAM mode of IsoCAM3000 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 IsoCAM3000.

2.18.1 Holes insertion

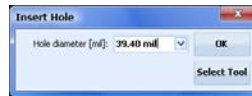
The submenu of “Insert” (from main popup menu) will provide you with two options





- 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).

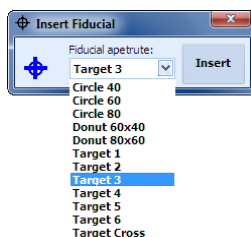


Notes:

- 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 than 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.

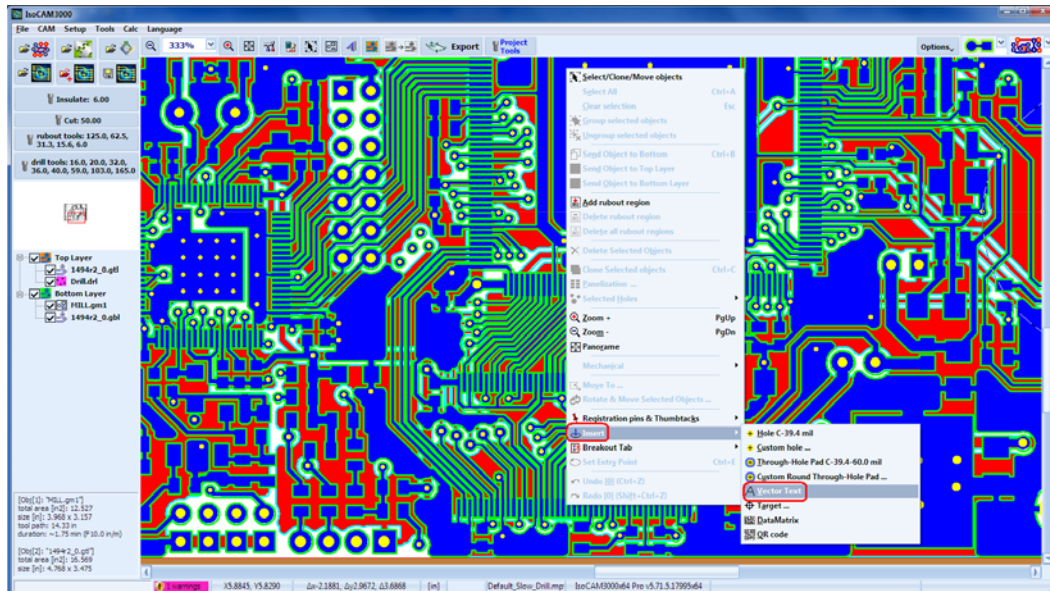


All apertures are predefined but adding new ones can be done by request.



2.19 Vector Text insertion function

IsoCAM3000 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.

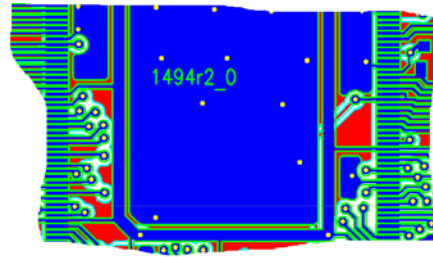
Buttons are for text slant (left or right).

Buttons are to change symbols width.

Buttons are to change the spacing between the symbols.

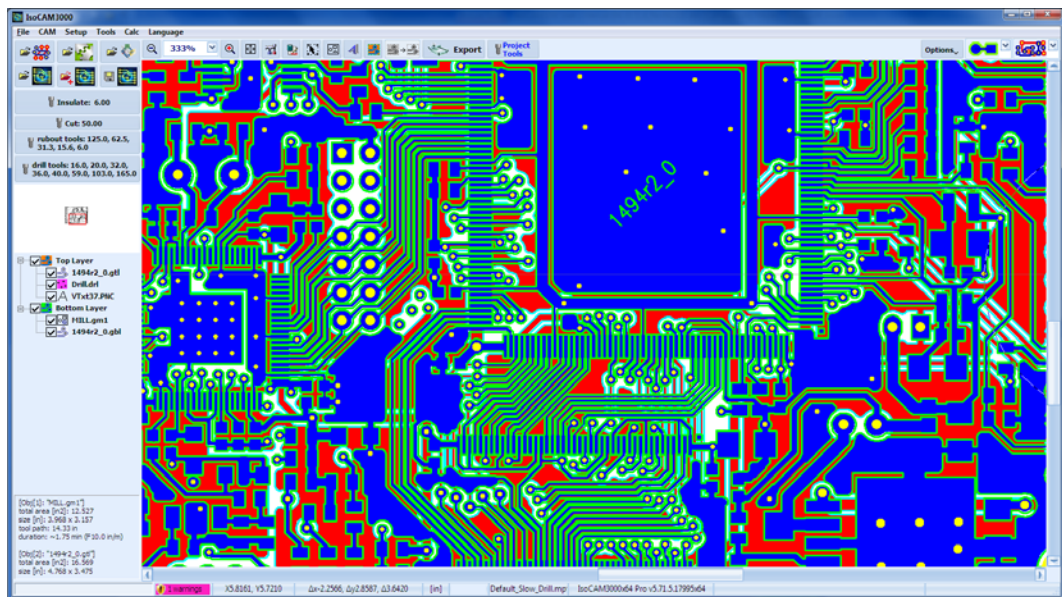
Buttons are to change the spacing between the lines (if there is more than one).

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.



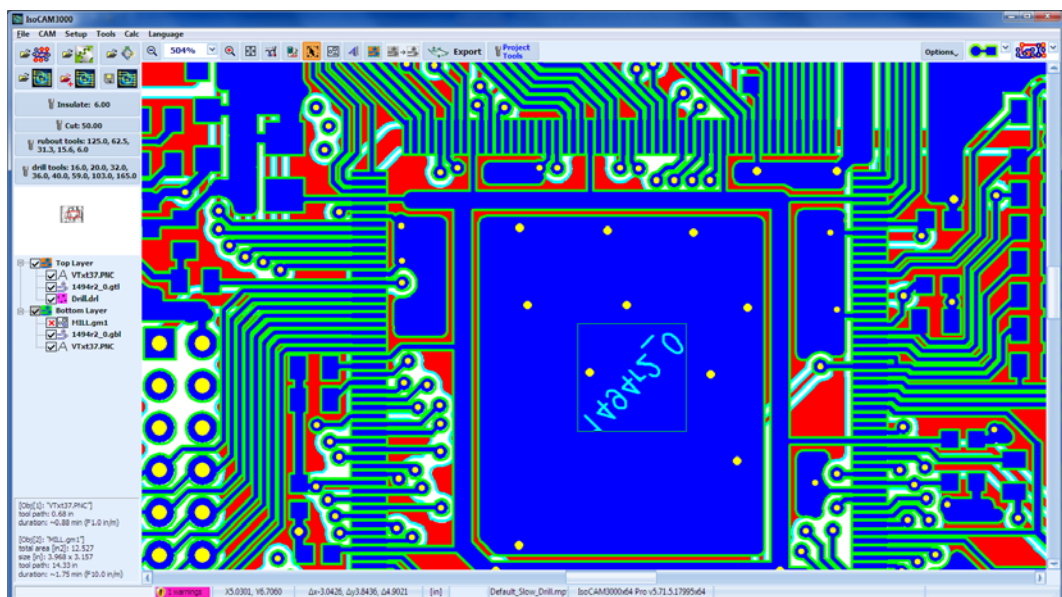
Inserted text objects can be manipulated just like other objects can they can be cloned, rotated, moved, grouped etc.

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.



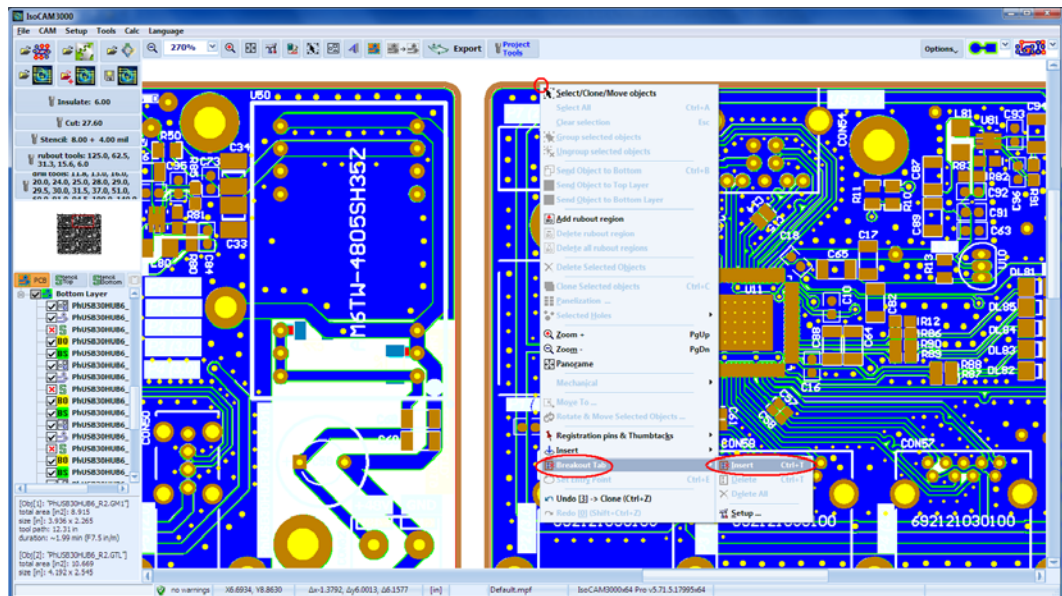
2.20 Breakout Tabs

IsoCAM3000 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.

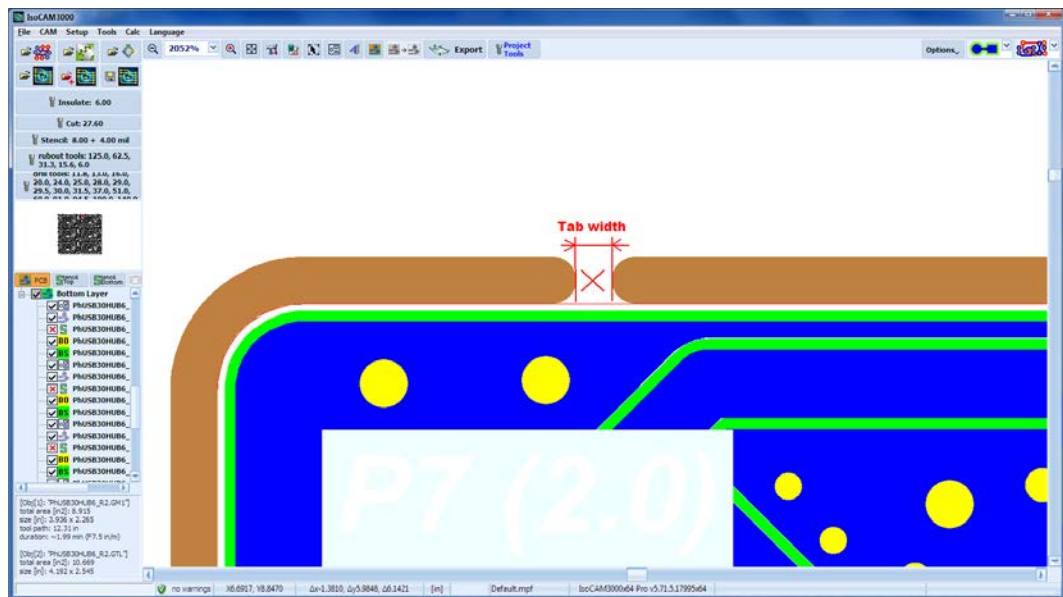
2.20.1 Breakout tabs insertion

Breakout tab insertions are possible in IsoCAM3000 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:



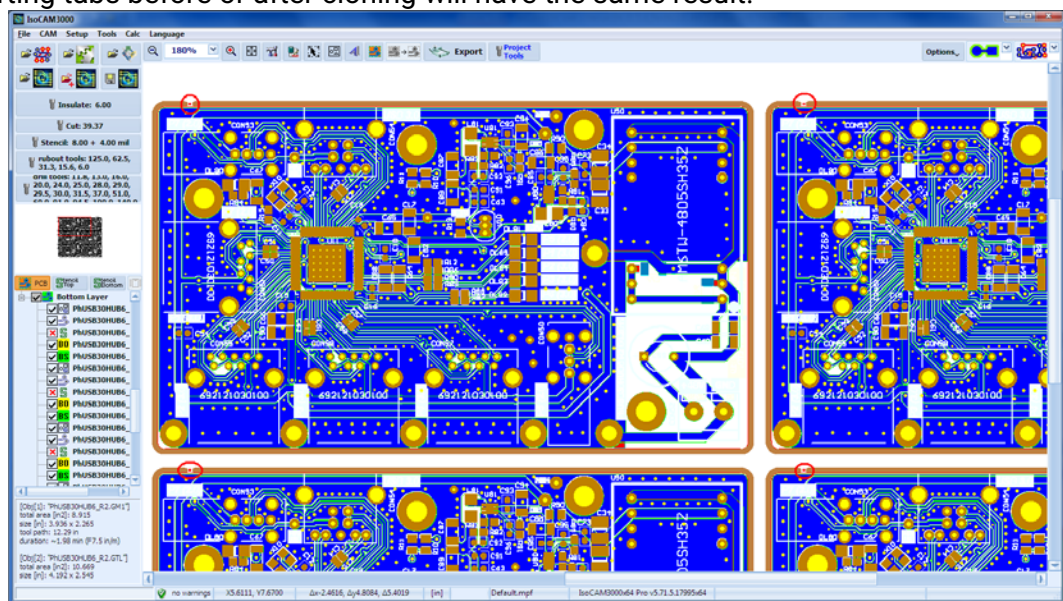
The "Insert" option is allowed only when the mouse pointer is on a cutout contour (mechanical layer).

Shown below is the inserted breakout tab:



Tab width is calculated by IsoCAM3000 and is independent from other contour parameters e.g. if you set 20 mils, the remaining material (at the closest point) will be 20 mils.

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



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

Note:

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

Note:

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



2.20.2 Breakout tabs removal

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

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

Note:

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

2.20.3 Breakout tab width setting

Select “Breakout tabs” then select “Setup”.



By “default” tab width is 20mil (~0.5mm) but can be changed to meet certain material requirements. When the width is changed IsoCAM3000 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, IsoCAM3000 will remove all tabs. Therefore it is recommended to insert tabs after the selection of a cutout tool.

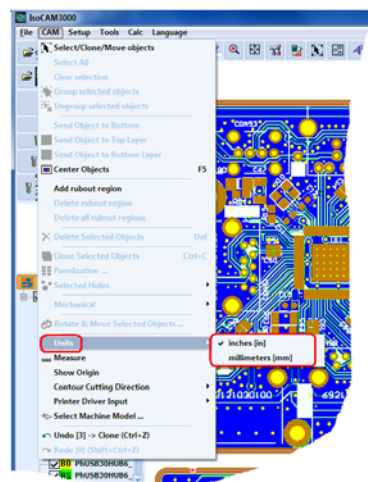
2.20.4 Breakout tabs complete removal.

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

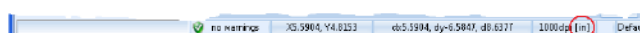
2.21 Units of measurement [mm]/[in]

IsoCAM3000 (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

IsoCAM3000 has an UNDO/REDO function. The size of UNDO/REDO buffer is 32 commands deep.

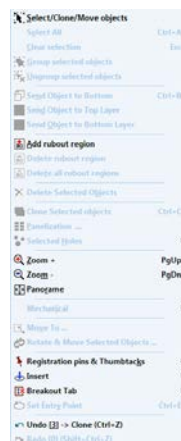
The following operations can use the UNDO/REDO function:

- MOVE (objects move)
- IMPORT (import of project or gerber/drill files)
- CLONE (copy/clone of objects)
- GROUP (group of objects)
- UNGROUP (ungroup of objects)
- SEND_TO_BOTTOM (Send to Bottom)
- ROTATE (rotation of objects)
- DELETE (removing of objects)
- INSERT_HOLE (insertion of drill holes)
- INSERT_PAD (insertion of pads)
- INSERT_TTF_TEXT (insertion of TTF text)
- INSERT_VECTOR_TEXT (insertion of vector text)
- NEW_PROJECT (new project)
- FIDUCIAL_MOVE (positioning project - CNC mode - point one) available in IsoCAM3000 only
- FIDUCIAL_ROTATE (rotating project - CNC mode – point two) available in IsoCAM3000 only

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

Note:

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



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

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

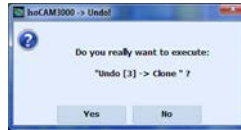
Most of Undo commands activated by Ctrl+Z need additional conformation.

Commands that are executed without conformation are:

MOVE, GROUP, UNGROUP, ROTATE & SEND_TO_BOTTOM



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

**Note:**

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

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

Important:

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

- Load Project
- Save Project
- Tool size change (insulation; cut; stencil) or project dpi

2.23 Forced Insulation

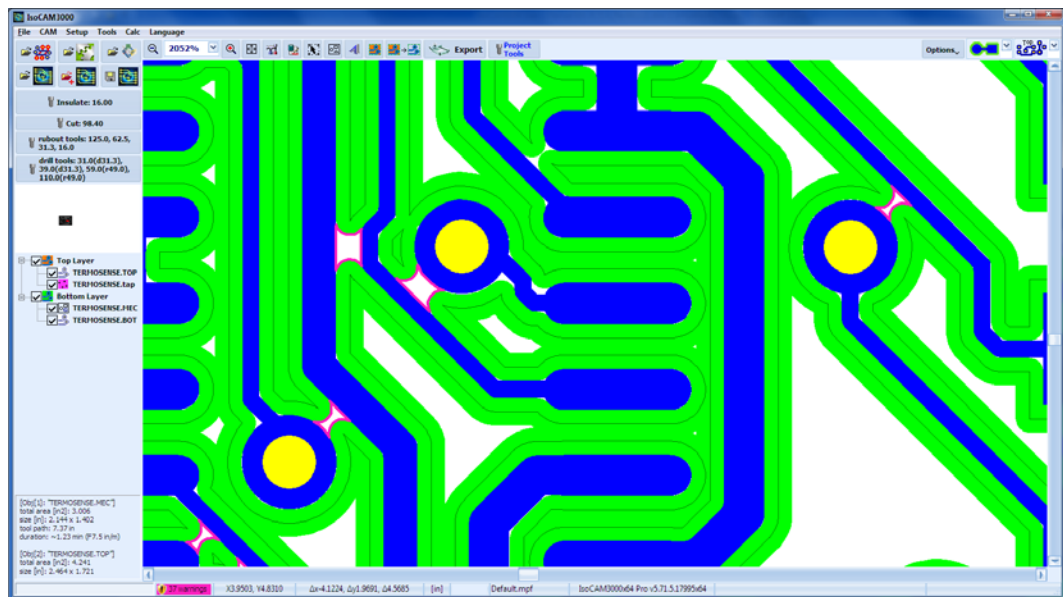
IsoCAM3000 has a function "Forced Insulation" that allows the system to handle sections of the design marked by DRV (See 2.13 Design Rules Violation) as impossible to insulate. The standard response of IsoCAM3000 tool path generation doesn't allow the tools to penetrate inside the traces/pads contours 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:

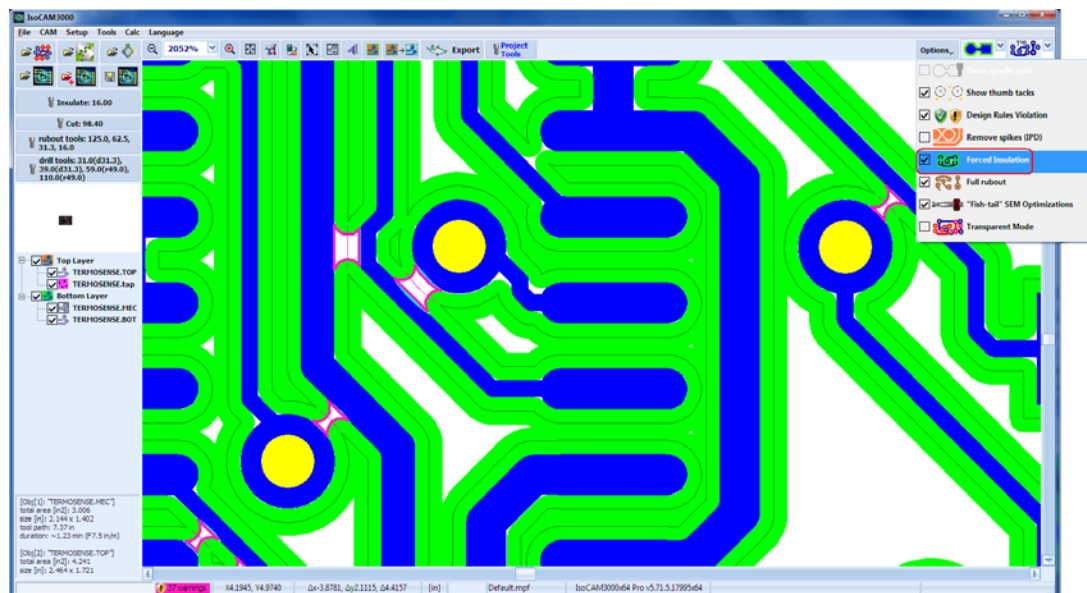
- Calculates tool paths used in "Forced Insulation". If the tool paths are never calculated the first activation will do so. If DRV function wasn't activated it will start it.
- Shows/hides tool paths related to "Forced Insulation".

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:

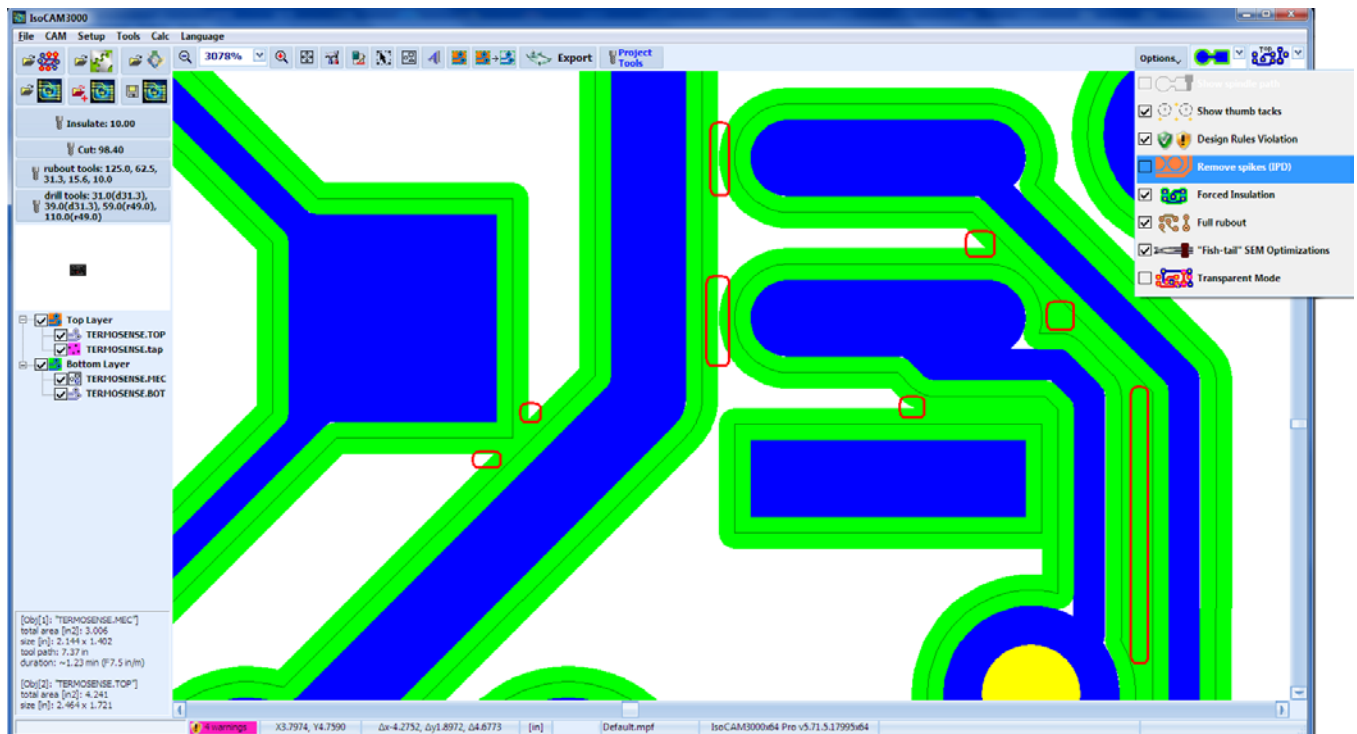


Light blue (default) strips shows additional copper removed from the design features when zoomed in to view the “Forced Insulation” paths.

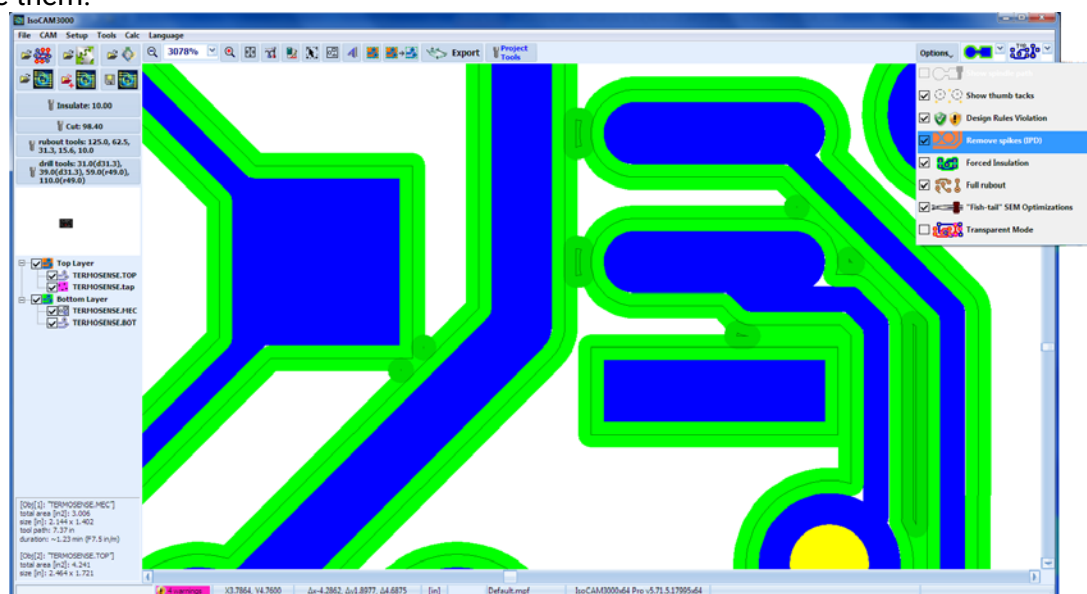
Button  will also serve as a switch to include/exclude the problematic sections in the machining process. “Forced insulation” tool paths are not automatically recalculated when you change the insulation tool diameter. If you change the insulation tool diameter you need to run the DRV function again to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

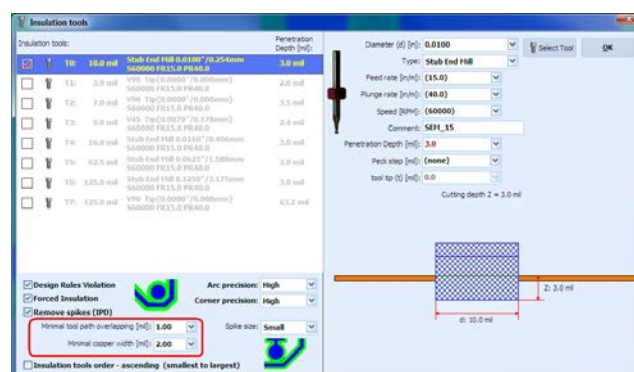
When the insulation only is used (not full copper rubout) very tiny copper strips may be left over from machining (hair sized). They do not have enough foundation to the board substrate and usually become free moving copper hairs that can create a condition for fine (hard to see) shorts on the board. 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. IsoCAM3000 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:



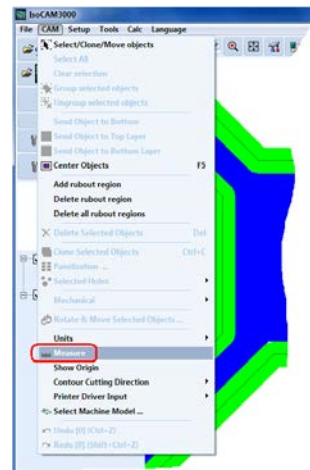
- minimal distance between tool paths that can be considered problematic (usually 0.5 mil).
- minimal residual copper width that can be considered source of potential problems (usually 2 mil).

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

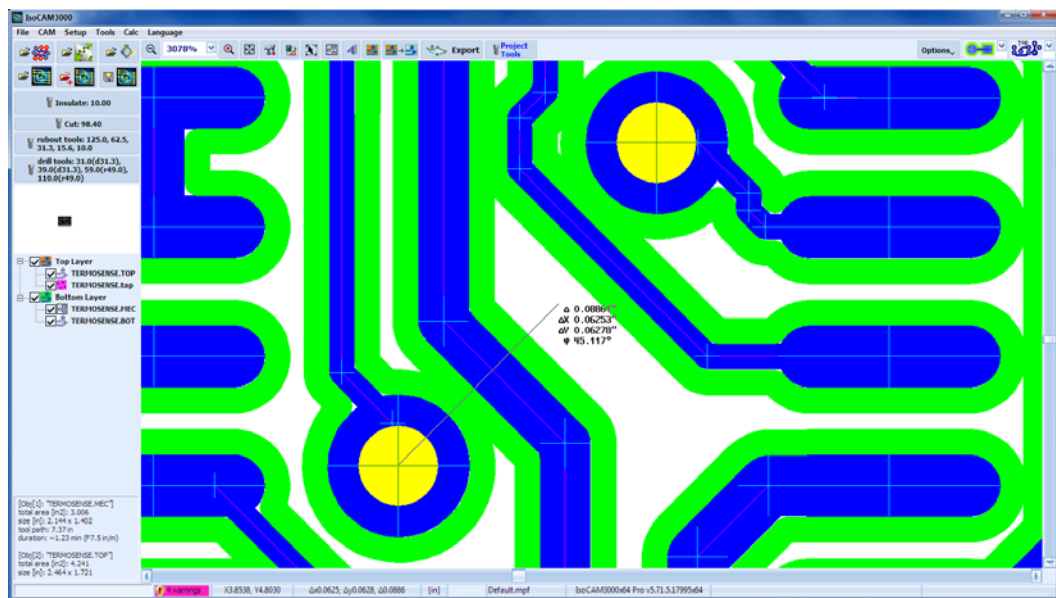


2.25 Measure

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.




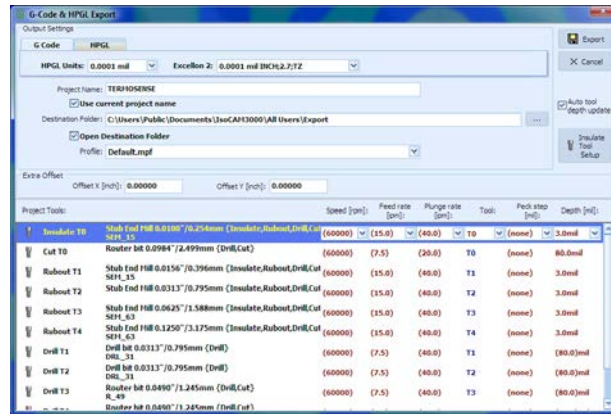
On screen information contains the distance, X difference, Y difference and the angle related to X axis. It is shown in the measurement units selected (inch/mm).



2.26 G-Code / HPGL Export from IsoCAM3000

The software revision named IsoCAM3000 generates G-Code or HPGL that can be executed on almost any CNC machine that accepts G & M codes or HPGL / Excellon2.

The button  in IsoCAM3000 calls a window with two tabs for G-Code / HPGL export:



IsoCAM3000, generates a G-Code / HPGL program for each machining process needed in your design:

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

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

Basic parameters that you must determinate are:

G-Code Units:

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

HPGL Units:

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

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

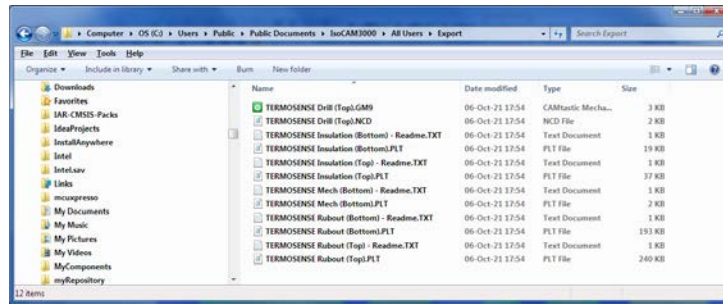
Project Name:

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

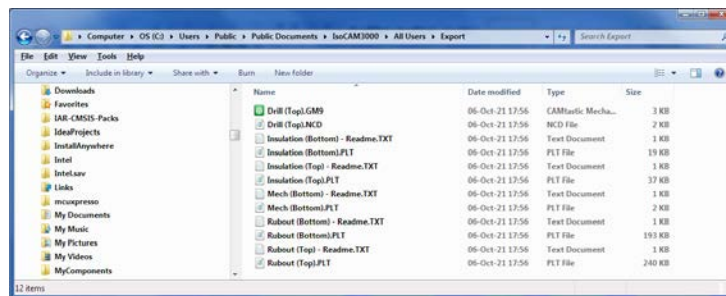
Example:

Project name: THERMOSENSE,

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.

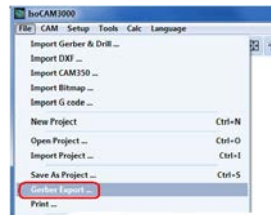
“Open destination folder”

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

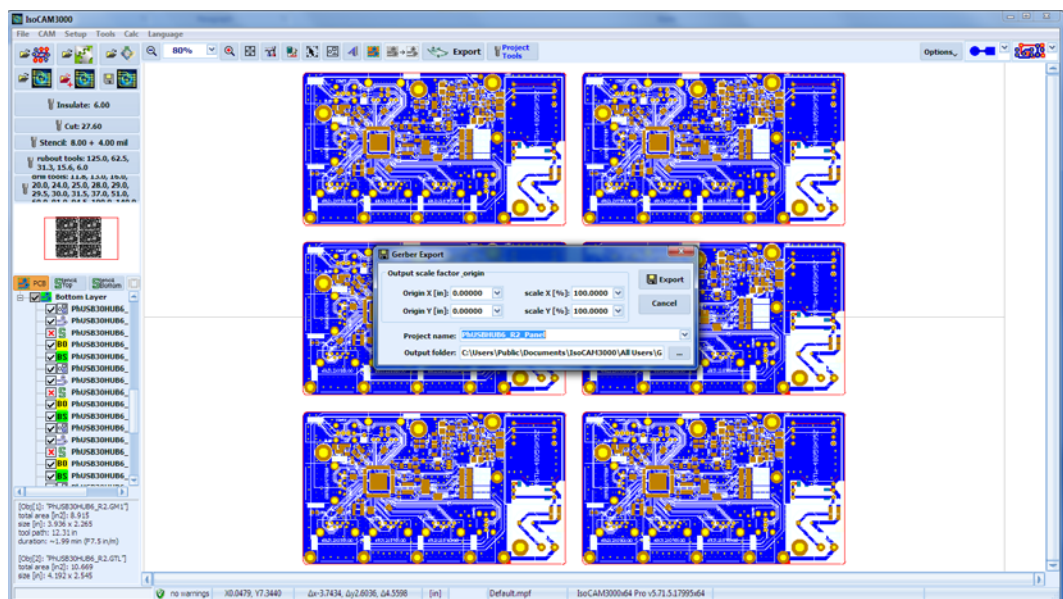


2.27 Gerber Export ...

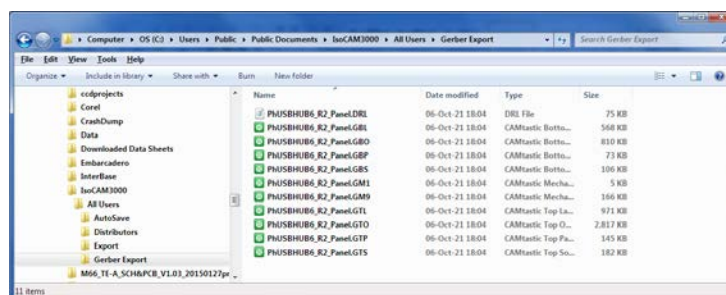
IsoCAM3000 allows an export of all Gerber and Excellon drill files in the current project . All visible files are exported including the info files that are not involved in the machining process.



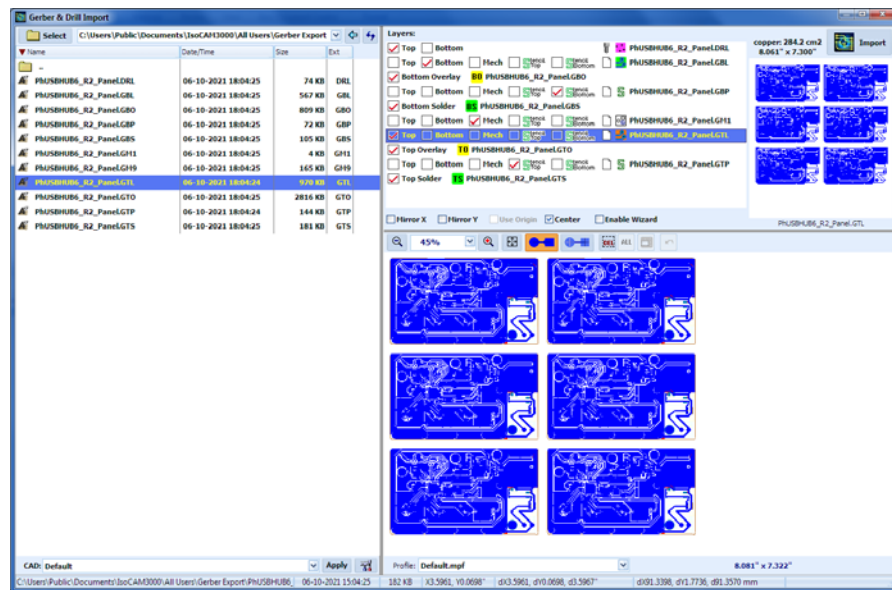
Gerber Export window allows to scale the files exported in Gerber and Excellon format (if needed).



After the export IsoCAM3000 automatically opens the folder that contain the exported files.



Exported Gerber and drill files can be imported back in IsoCAM3000 (for example after scaling on X/Y axes) or used in another CAD/CAM software.



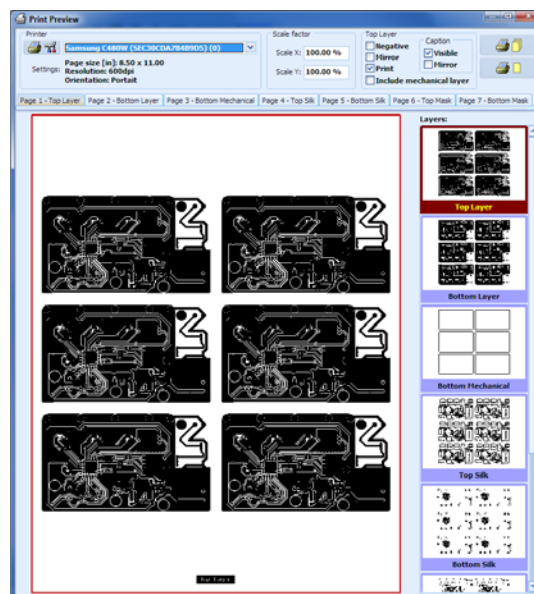
Import & Export Gerbers & Info layers - video:
(2.27) <https://youtu.be/IVOVg7DZEgg>



2.28 Print


IsoCAM3000 has an additional capability to print the layers of the project. Print Preview window has the following options:

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





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

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

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

Individual settings for each layer (Negative, Mirror, Caption, Caption mirror) and the printer used for IsoCAM3000 are saved in order to shorten the time needed in the next usage of Print option.

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

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

Note:

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

Print preview video:

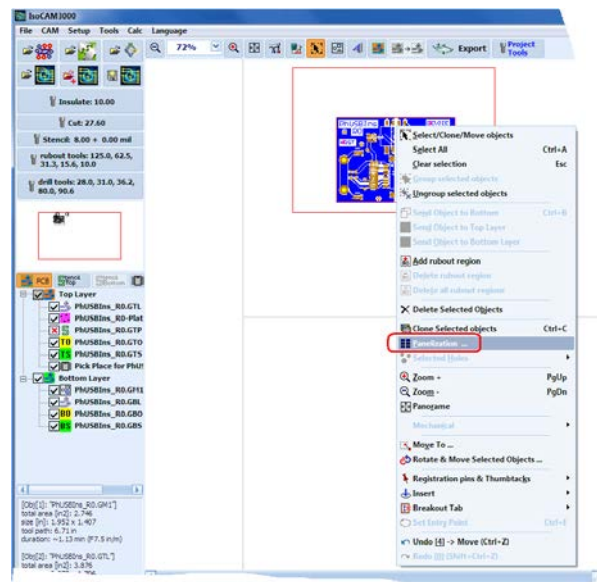
(2.28) <https://youtu.be/XWJisXLZyzI>



2.29 Panelization

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

This IsoCAM3000 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:

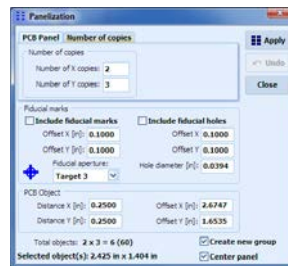
- size of the object(s) (Selected objects(s))
- required distance between them (PCB Object->Distance X/Y)

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



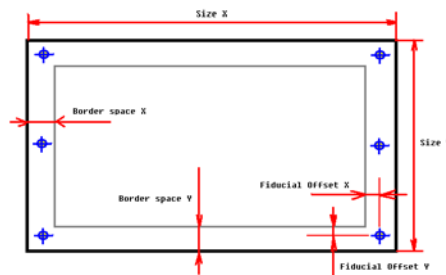
This is the easiest way to create the desired number of copies of your project entering the repeats on X and Y axes.



2.29.3 Fiducial marks

IsoCAM3000 allows inserting of targets & holes as fiducial marks around the panelization objects. Settings include: Offset (Fiducial marks->Offset X/Y); holes diameter (Fiducial marks->Hole diameter); aperture type (selected from the list).

The offset (Fiducial marks->Offset X/Y) depend of the mode selected.
In "PCB Panel" mode, the offsets are defined in relation to Border space:

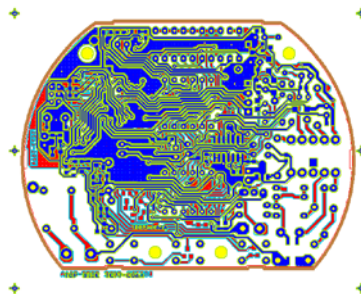


If negative values are used the marks will enter the machining space.

In "Number of copies" mode, the offsets of the fiducial marks are defined related to the area of placement of the objects. Again, if negative values are used the marks will enter the machining space.

"Number of copies" mode allows easy adding of fiducial marks even on a single object. To do so the Number of copies X/Y should be set to 1.

Example:



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

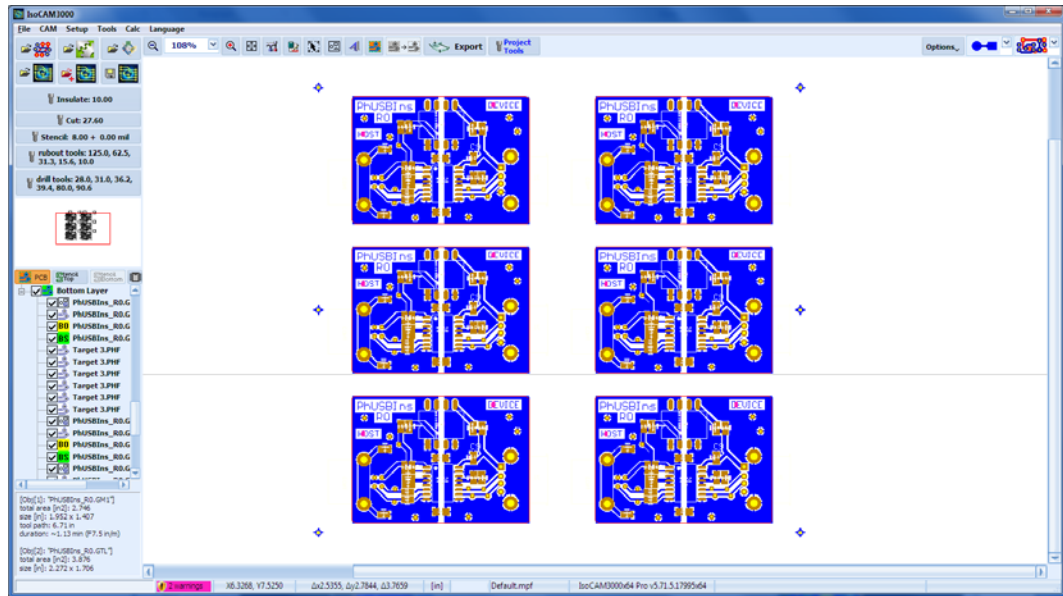
2.29.4 Buttons "Apply" & "Undo" (Ctrl+Z)

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

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

2.29.5 Create new Group

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

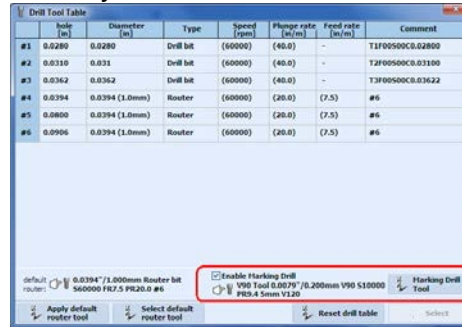




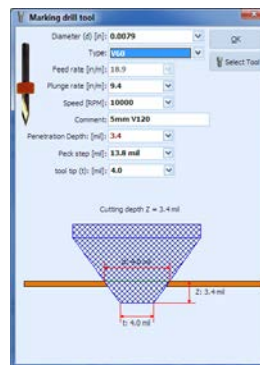
2.30 Marking Drill

IsoCAM3000 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.

If the larger holes will be cut with router bit, they are excluded from the Marking Drill program!

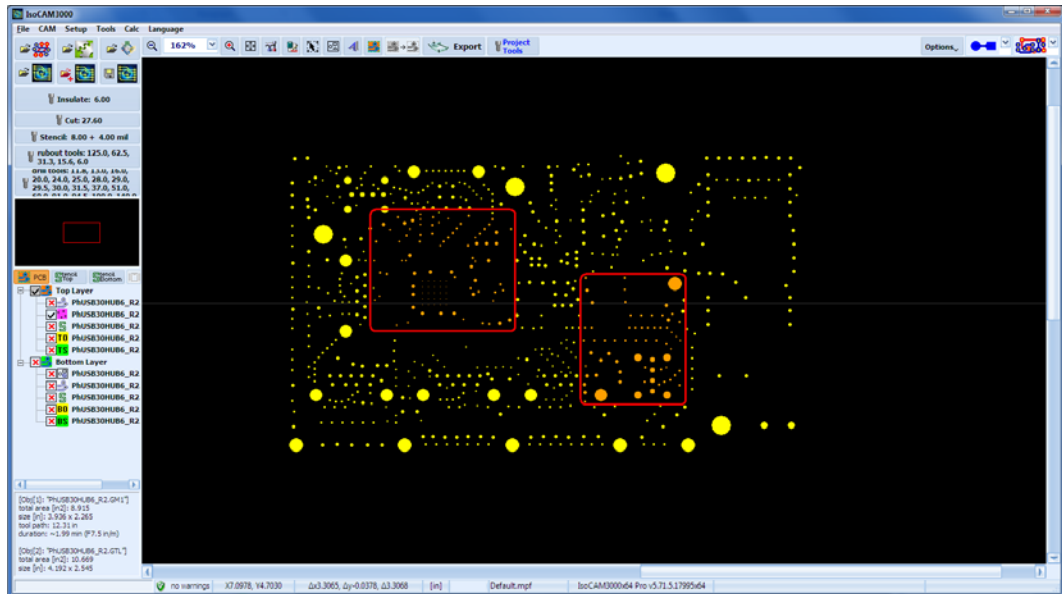


2.31 Holes Selection / Modifications / Removal

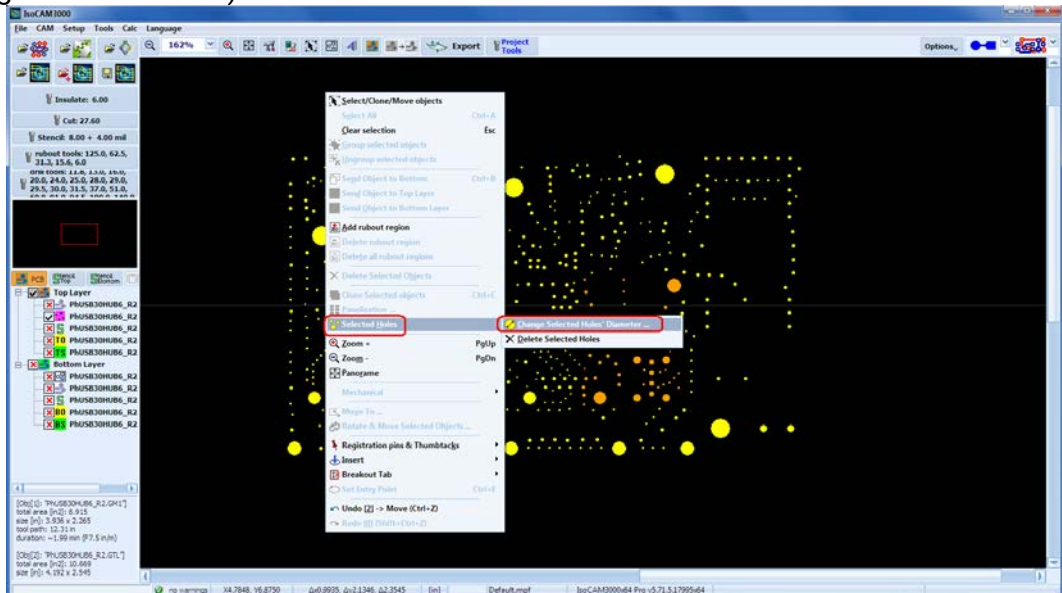
IsoCAM3000 allows to select holes from the project and modify/delete them.

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

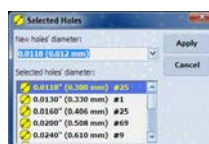
Example – on the screen below 2 rectangular regions are selected (the gerber files are hidden for clarity):



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



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





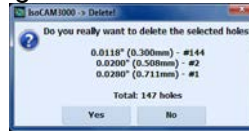
In the list you can find the holes' current diameters and the number of the selected holes with this diameter.

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

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

- If the holes removal is selected, IsoCAM3000 will require additional confirmation for the process to begin.

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



Notes:

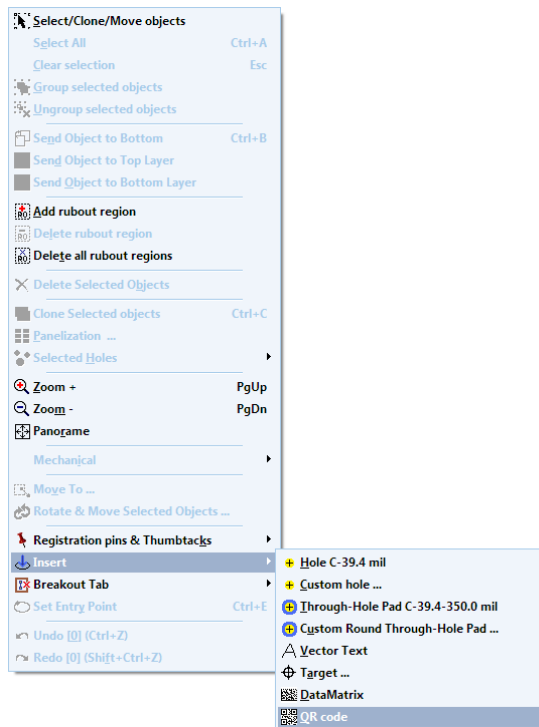
- In case of mistakes related to the above two functionalities, you can use "Undo/Redo".
- If the project has multiple copies the changes will affect all of them, IsoCAM3000 does not allow the copies of the same project to be different.

(2.31) <https://www.youtube.com/watch?v=MKYtO4IRoY0>



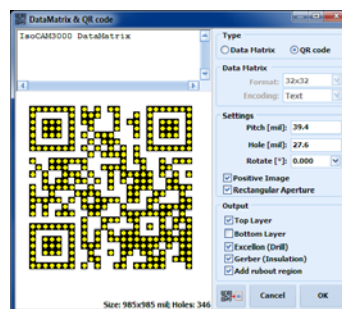
2.32 QR Code

IsoCAM3000 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:

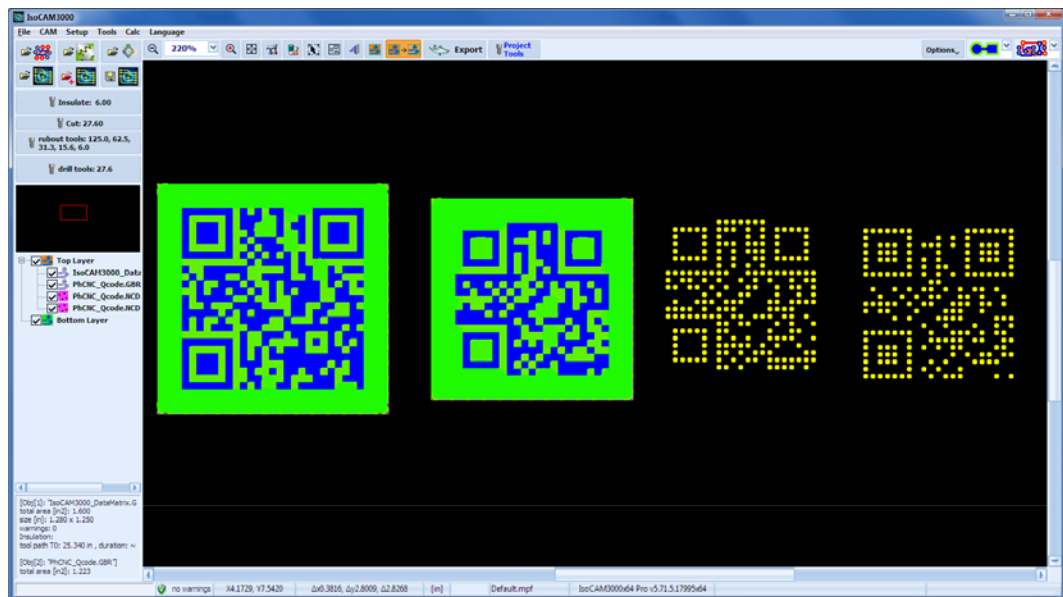
Positive image QR Code with Gerber & Drill:



Positive image QR Code with Gerber (engraving / insulation)

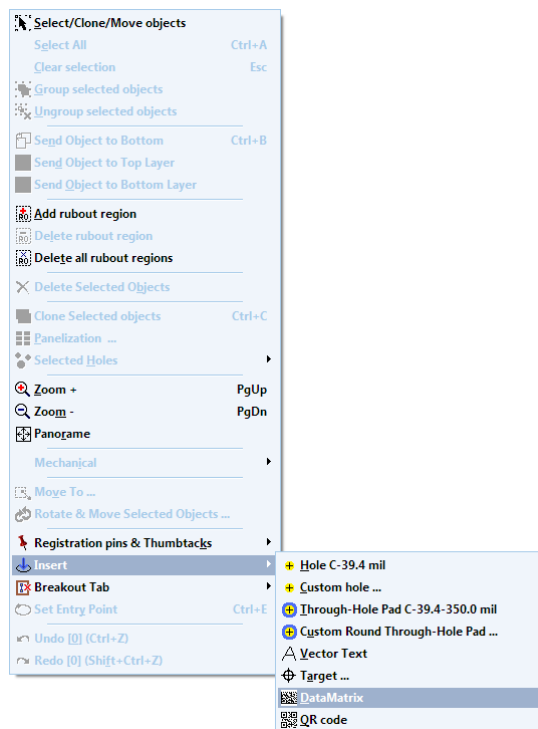


The result of several inserted QR Codes with negative, positive and drill only images:



2.33 DataMatrix

IsoCAM3000 allows to insert DataMatrix at the mouse position from the Pop-up menu (right mouse click):

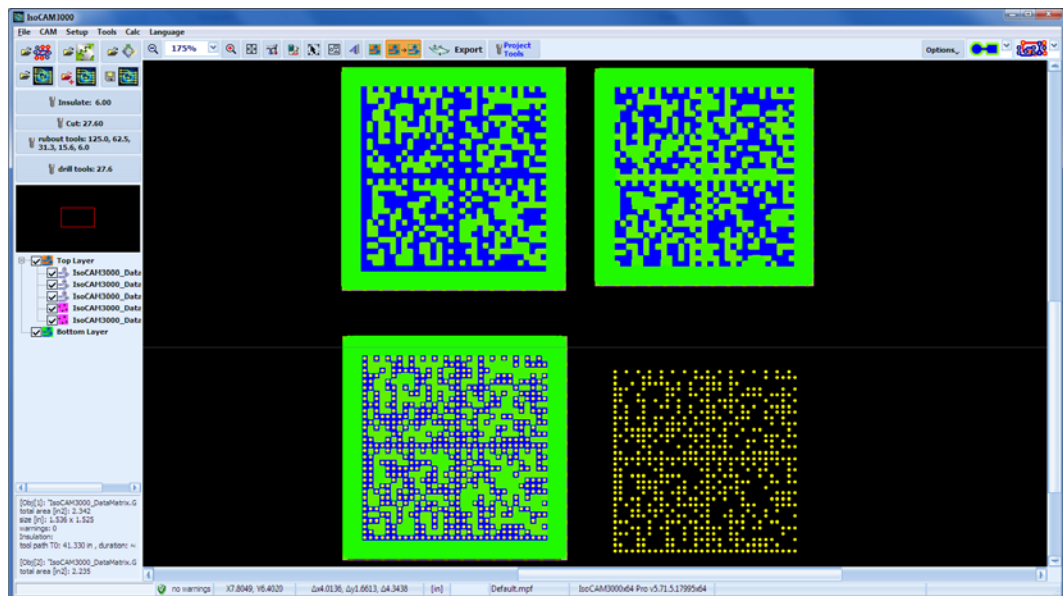


A lot of options are available depending on the methodology of manufacturing, the pitch size and the image size:

Positive DataMatrix with Gerber (engraving / insulation):



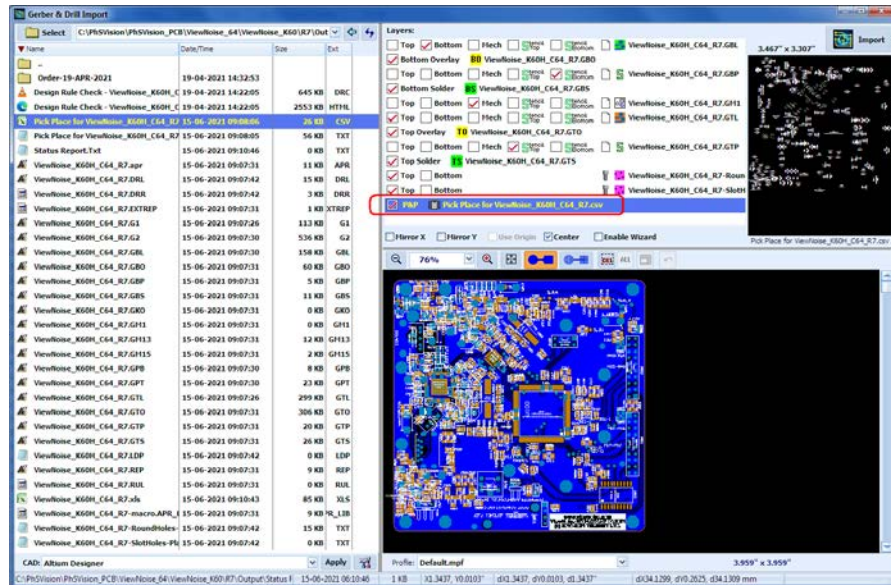
The result of several inserted DataMatrix with negative, positive and drill only images and with format 80x80:



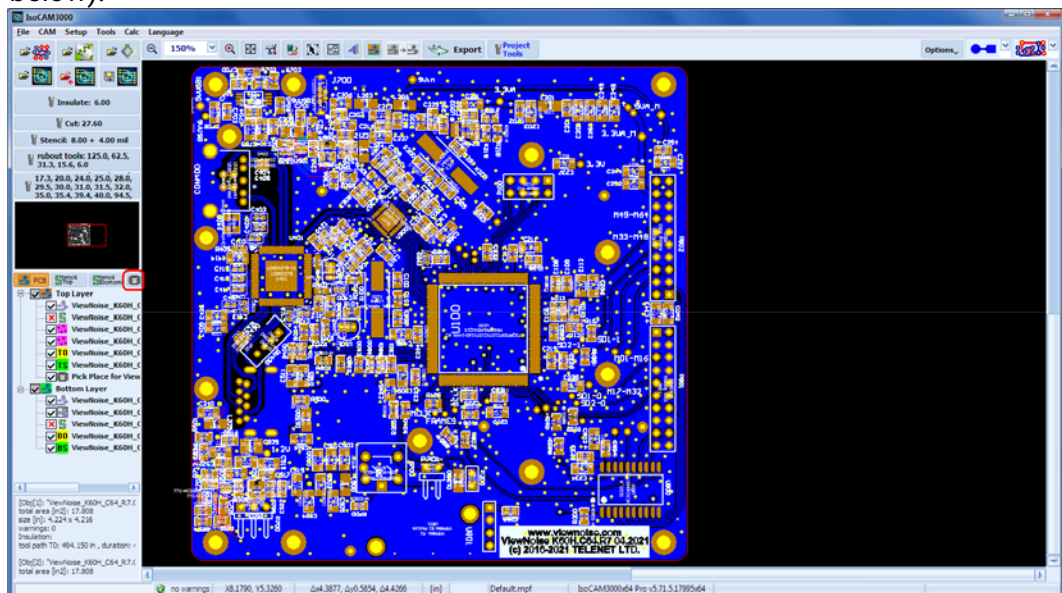
2.34 Pick & Place Helper

IsoCAM3000 can import Pick & Place assembly files in CSV or IPC-D-355 formats (automatically detected by the software).

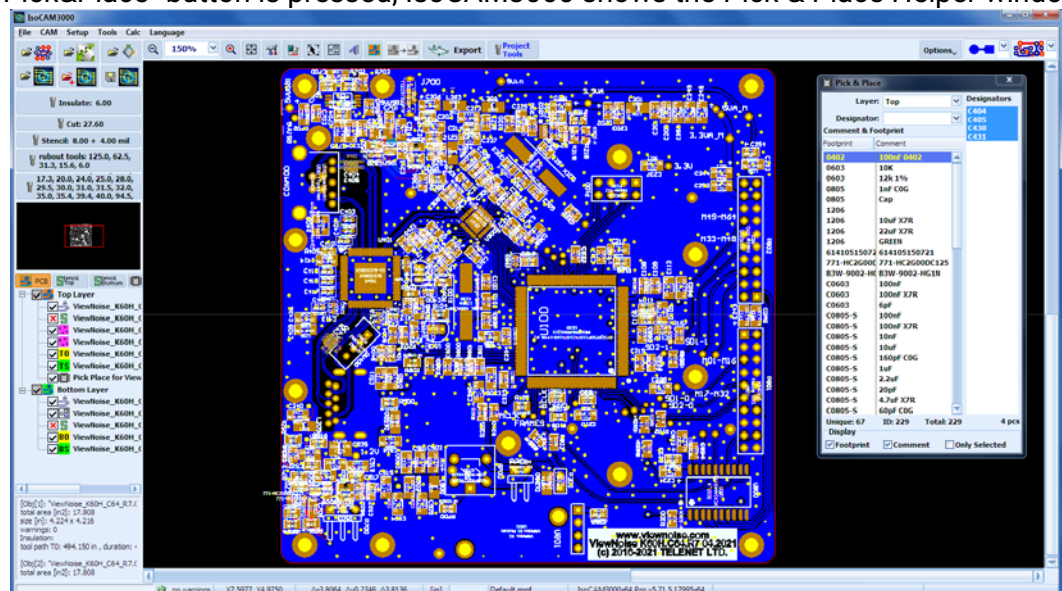
Here follows an example for project import with Pick & Place file:



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, IsoCAM3000 shows the Pick & Place Helper window:



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

(2.34) https://www.youtube.com/watch?v=U0ZC13cZ7_I

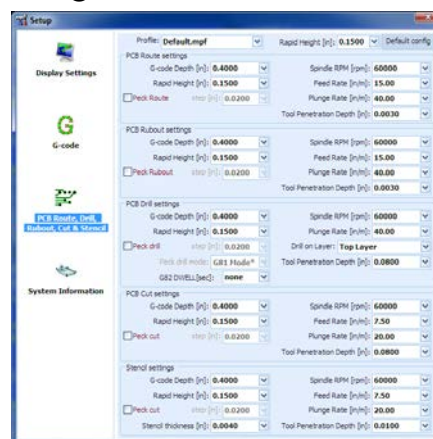


3. Setup

All adjustments needed to run IsoCAM3000 are located in the window below:



3.1 CNC process control settings



Important:

The parameters included in this table are used in creating the G-Code programs.

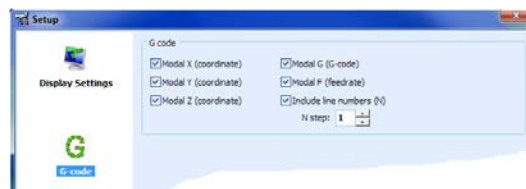
The Profile box is used to select the profiles you create to store your settings. You can create many different profiles for different materials and job conditions. This allows you to do fast and complete settings changes. The current profile is always stored in your .PHJ file, so if you have to repeat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

3.2 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 IsoCAM3000, but they do not affect machine control.



3.3 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%.

IsoCAM3000 has 7 built in skins, you are able to change them in this window.

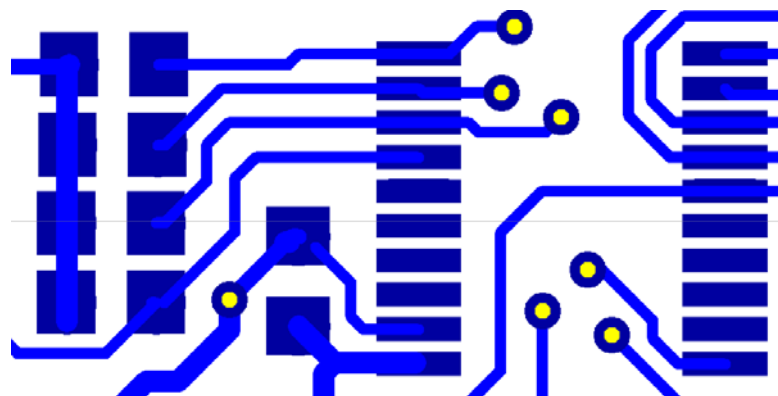
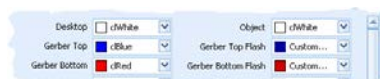


You can change the colors for drawing of the objects on the screen, as well as create different color schemes called profiles. You can select a profile by entering the name of the profile in the profile box or selecting it from the drop-down list.

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.



3.4 Multithreading & IsoCAM300064

IsoCAM3000 does extraordinary volume of calculations during the determination of tool paths in Insulation, Rubout, Remove spikes (IPD), Forced Insulation, ...

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

When a parallel processing is in use, **IsoCAM3000** will need larger size of RAM. This problem does not exist in **IsoCAM300064** - the 64 bit version of **IsoCAM3000**. This version is designed to run under **x64** OS (Vista/Win7/8/8.1) and is capable to address more than 4GB of RAM.

IsoCAM300064 is fully identical to **IsoCAM3000**. The 64 bit version simply is capable of full utilization of the capabilities of the new x64 OSs.

IsoCAM300064 and **IsoCAM3000** can be installed on the same computer, but only one can be run at a time.

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



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

IsoCAM3000 will utilize the maximum number of parallel threads when it is possible or needed.



4. In case of further questions

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

support@bungard.de