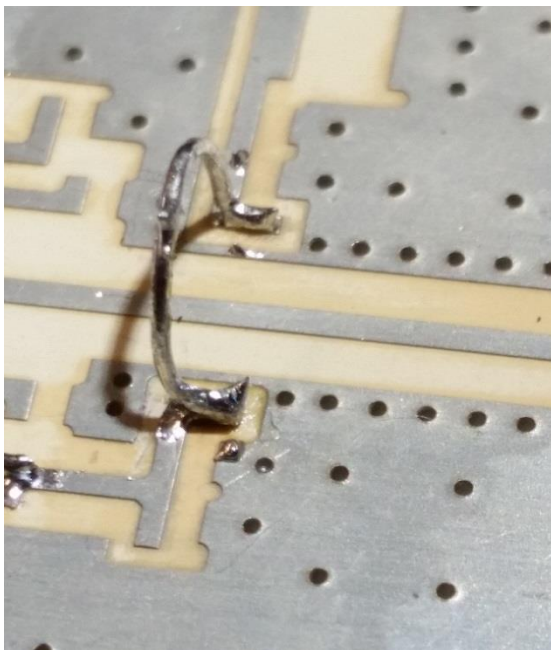


Ultra-Precision Desktop Milling

Accurate CNC

Microwave Applications



Claude Setzer, Ph.D.

© 2017, Engineering Depot, Inc.

Introduction

There are many ways to do things, with each having some advantages and disadvantages. This manual has chosen a few simple tasks to illustrate some unusual capabilities of the AccurateCNC machine, and uses an almost “cookbook” method to allow the reader to get good results and gain both basic and advanced expertise.

Design [CAD] software used was chosen to allow you to experiment with minimal expense and maximum ease of use. [Target 3001 is easy to learn, available free of charge for non-commercial use, and does what is needed.] Even the commercial product is inexpensive, so skills learned will not be lost. When machining the millimeter wave waveguide coupler, AutoCAD is used for simplicity of implementation.

There will be two essential applications illustrated in some detail:

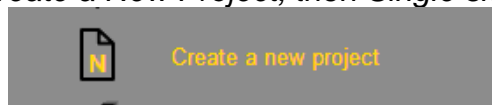
- 1) Making a PCB with a microwave transmission line on it, and some notes on mounting, making vias, and so on.
- 2) Making a waveguide coupler for usable from 140 to 220 GHz.

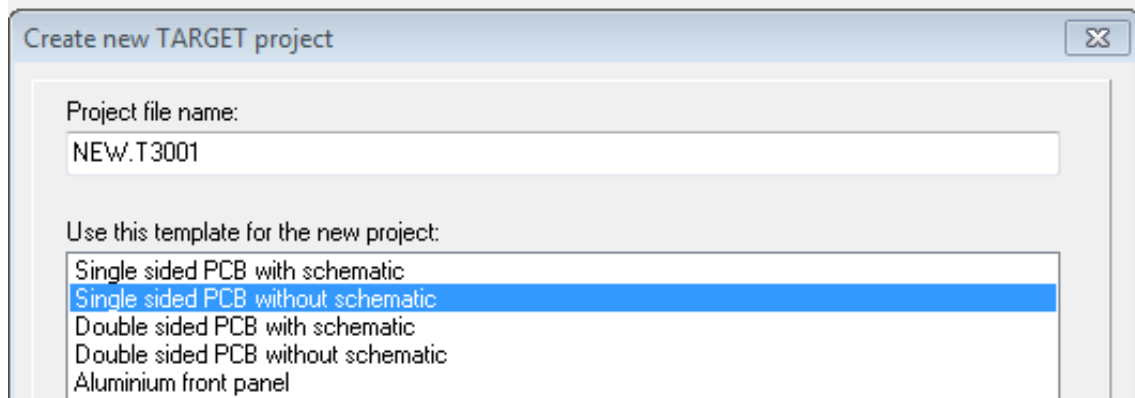
Part A: How to Create Gerber and Drill Files:

The example here is a 1.9 inch long 50 Ω microstrip transmission line. Software used is available free for personal use. Target3001 V17 or 19. [<http://ibfriedrich.com/index.htm>]

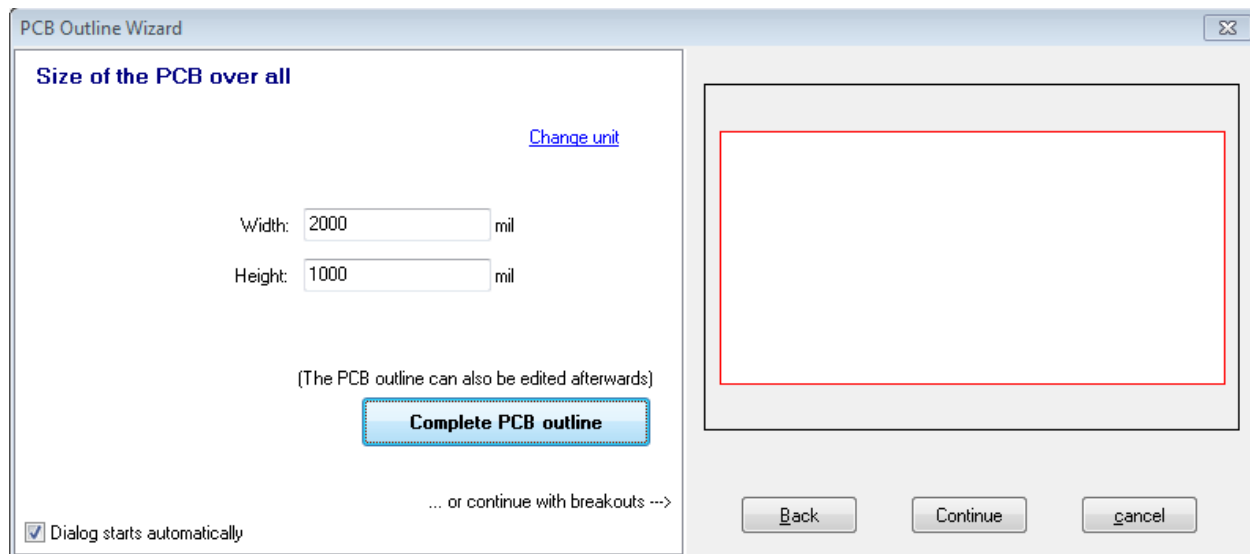
Getting Started with Target 3001

- 1) When generic “splash” screen appears, click anywhere to get *Target* startup options
- 2) Startup: Click on *Create a New Project*, then *Single sided PCB without schematic*

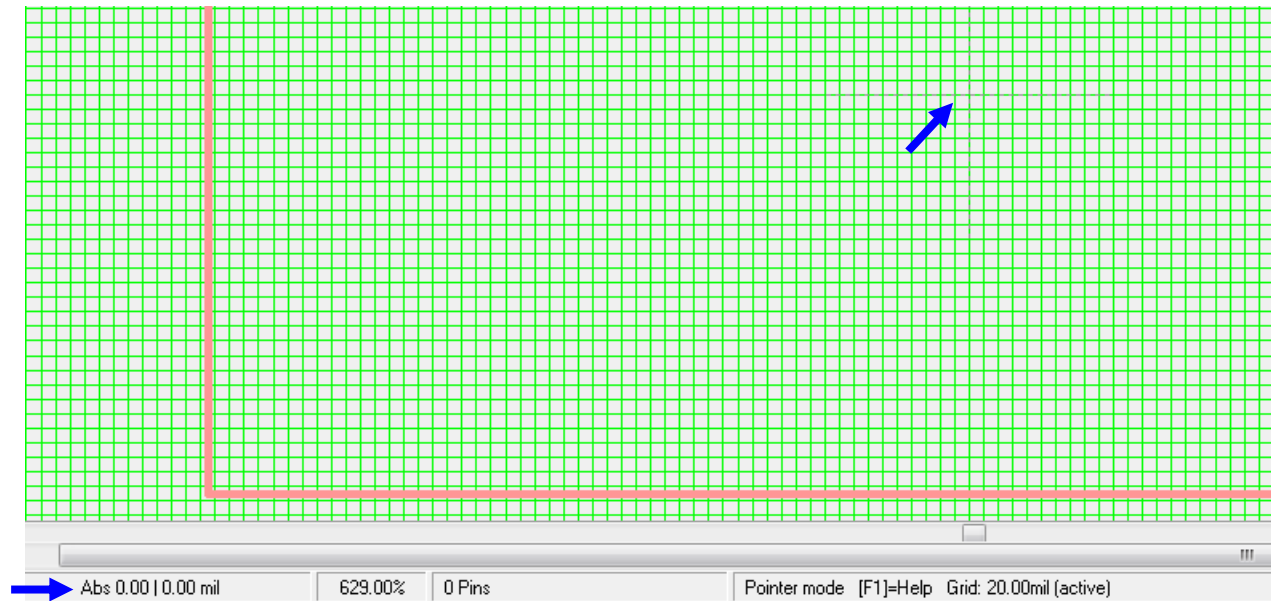




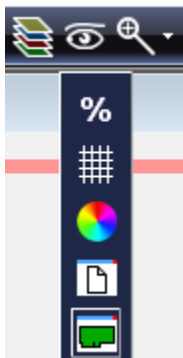
- 3) Menu opens to allow choice of PCB outline (board size). Default units may be mil, but you can click successively on Change unit to get: mm, μm , or Inch units. If you click on **Continue** under the box on right side, options will come up for creating small sections of board that are not cut out => breakouts. Otherwise, click on **Complete PCB Outline** to continue creating your PCB. Here we have a board that is 1.9 in. wide and 0.9 inch high, but add 0.1" extra in Target, then create the real board as a rectangle inside this area so that there will be more control over details and sizes [in the PhCNC software used by AccurateCNC].



- 4) The default board outline has a dotted cross hair (0,0 position) in center of board, which is the absolute reference for *Target*. Wherever the cursor is, you can press the *Home* key to create a *Relative Zero Position* and the scale at bottom of screen will read position with reference to this. To turn off Relative units, click again on *Home* key. You can also change position of 0,0 absolute by dragging board outline to desired part of board, e.g. bottom left corner.

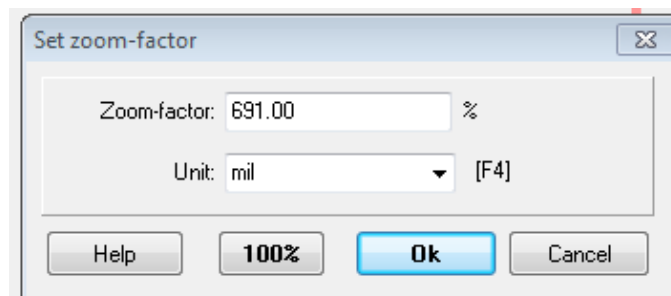


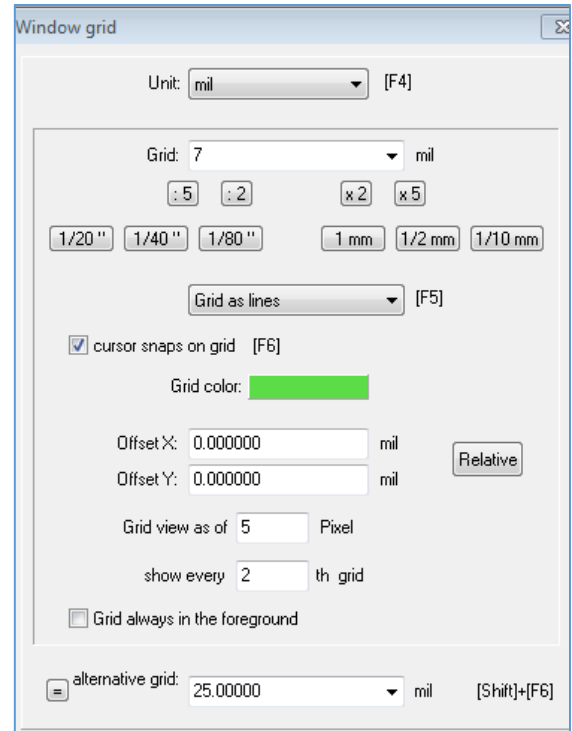
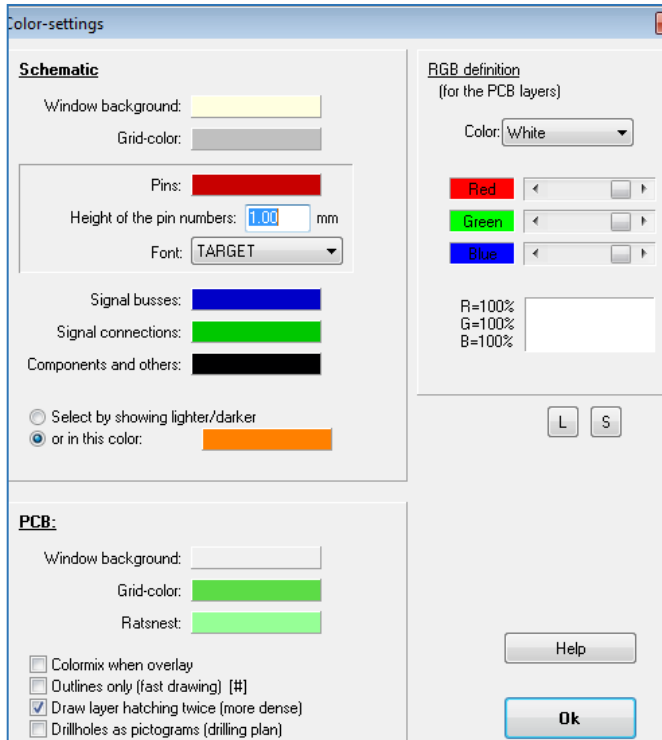
5) To turn on *Drawing Grid*, select Grid under *VIEW* menu at top of window, **OR** click on eye pattern at top of window, then



- a) Click on % sign to specify zoom in percent
- b) Click on grid pattern to setup grids
- c) Click on color wheel to select all *Target* colors
- d) Click Page symbol to create a new schematic window in this project
- e) Click green file to create new PCB window in this project

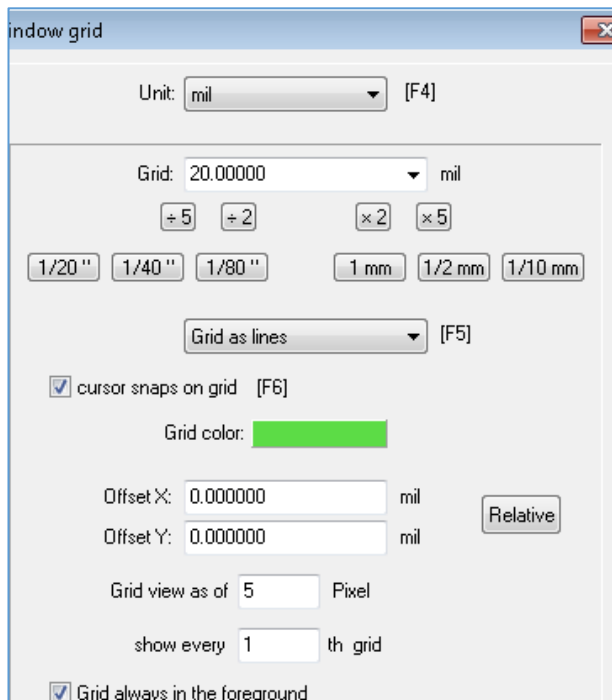
NOTE: You do not have to be overly concerned about details of grid now, as object's exact size and position can be set to what you need via the *Properties* menu. The grid is meant to be a convenient tool for starting position and alignment of shapes, traces, etc.



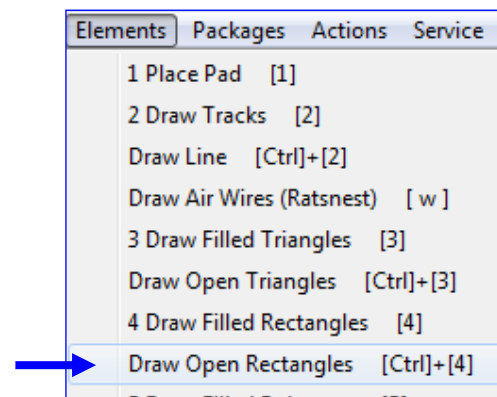


Example: board outline 0.9" by 1.9," circuit trace 43.14 mils wide, with two 60 mil fiducial [alignment] holes. Original board in Target needs to be set larger than needed, e.g. 1 by 2."

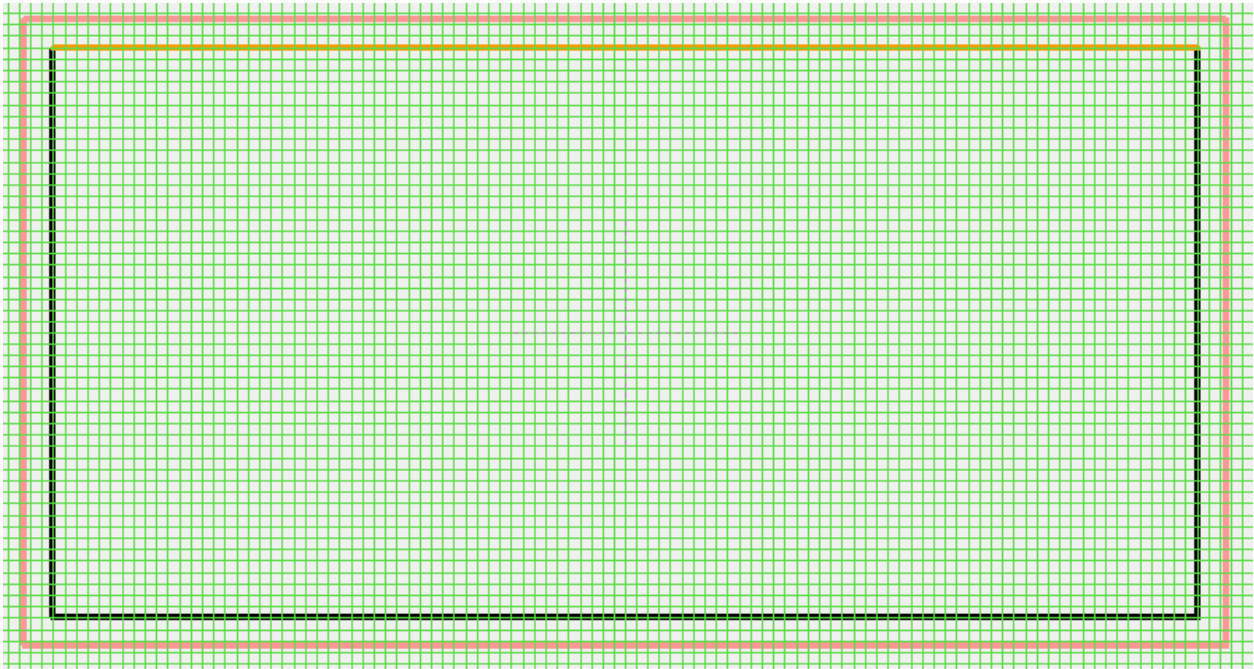
1) Set all grid options as shown:



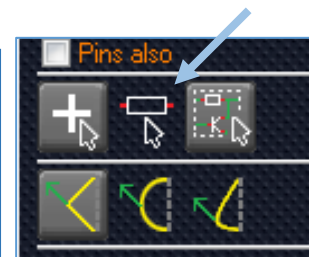
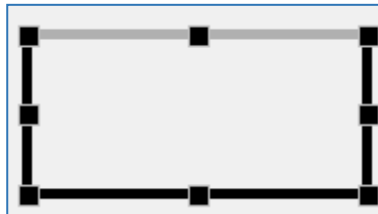
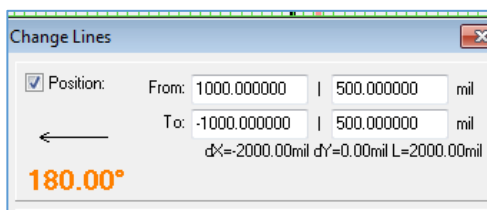
2) Under *Elements* menu at top of screen, select *Draw Open Rectangle*



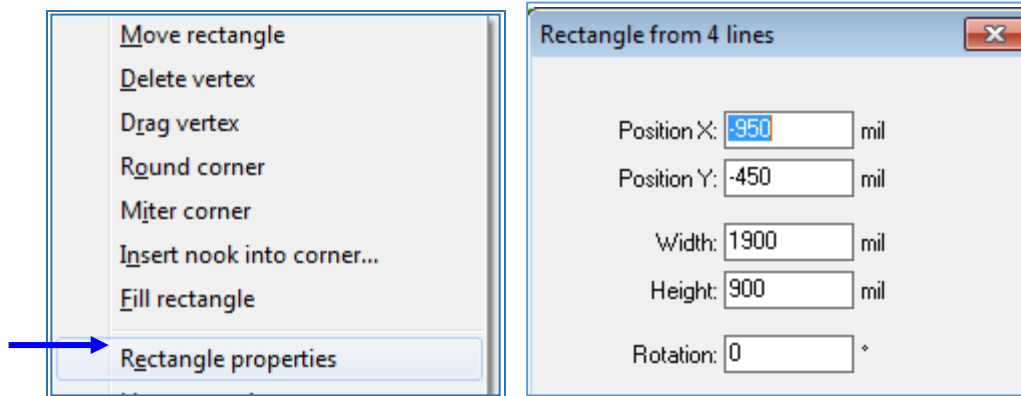
- 3) Use snap grid to create board size inside the Target board size limit set above. Click on top left corner for rectangle position, then hold down left mouse button and drag to give rectangle size of, e.g., 1900 mils by 900 mils.



The exact size and position of inner rectangle is adjusted by double clicking on each line and then typing in details as shown here for top line [Target V 16]. A new feature in Target V 17 allows direct adjustment of rectangle size and position. When open rectangle is created, it will now have multiple handles on it when selected. While handles are showing, click on the rectangle box in tool palette. Then click on box edge again to highlight handles.

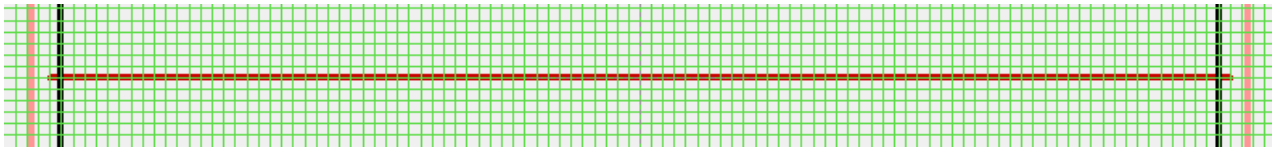


Right click mouse over one of handles and select *Rectangle Properties* to get edit window that allows you to adjust size and position of box.

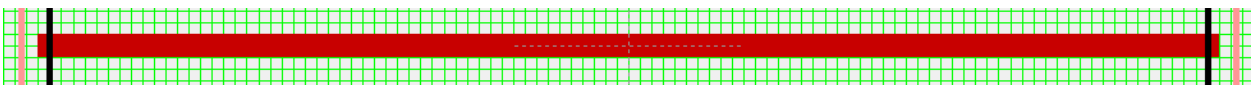
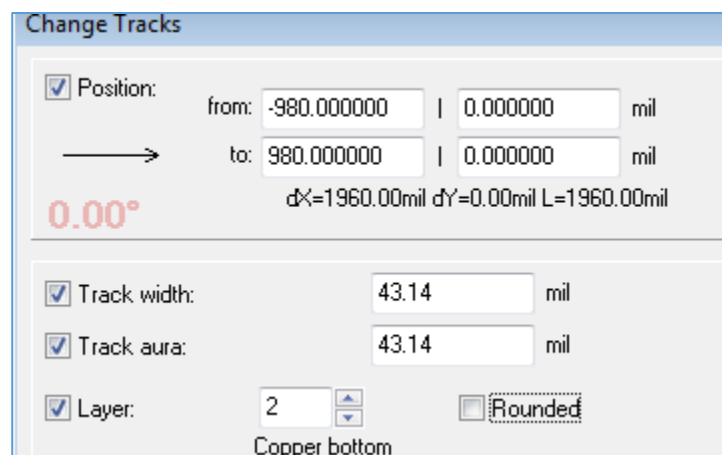


Add RF Transmission Line:

- 1) From *Elements* menu, select 2 Draw Tracks and use mouse to drag across approximate position where line is required. Here we are going to let AccuarateCNC “cut” the transmission line to exact position using the inner rectangle as a board cutout. Thus, the line must extend through the rectangle. Since we are running line to end of board, this is needed to remove the little “horns” that would otherwise appear due to CNC cutter radius.



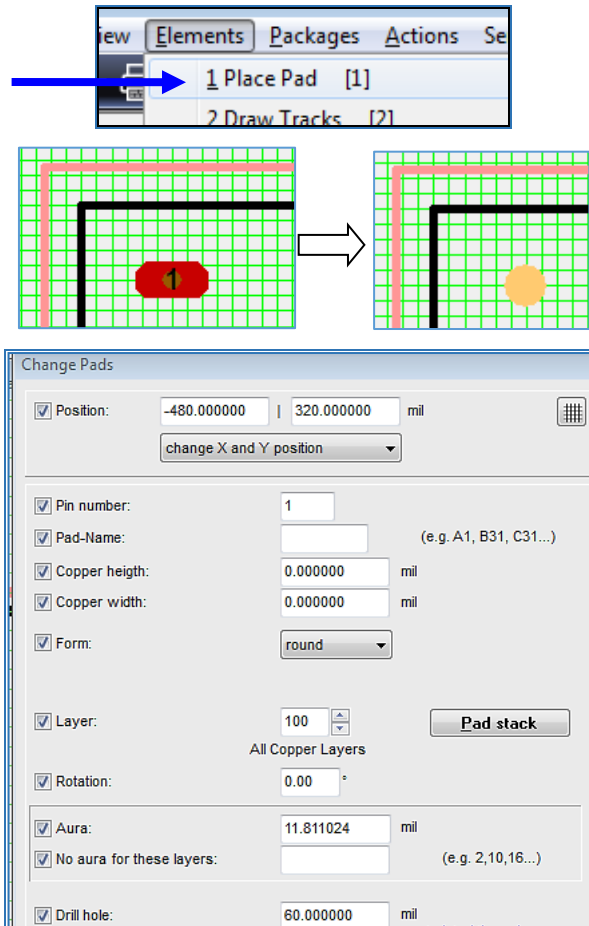
- 2) Double click on created transmission line to bring up *Properties* box. Here position is all right as set, but need to change track width and aura to 43.14 mils to give 50 Ω line width. Also untick the *Rounded* box. New line is shown below.



Add Fiducial Holes:

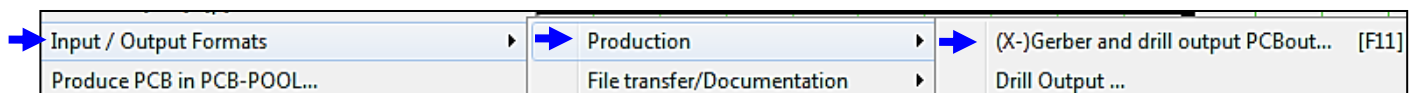
To precisely align board for the CNC machine to cut objects, we will need 2 clean circular holes cut in the board along the X axis. Use a 60 mil stub end mill cutter.

- 1) Under Elements menu, select 1 Place Pad.
- 2) Place a *Pad* near one of upper corners of inner rectangle.
- 3) Select *Pad Properties* by double clicking, then *OK*.
- 4) Set *Copper height* and width to zero as shown.
- 5) Set *Form* to *round*.
- 6) Set Drill hole to 60 mil. => *OK*
- 7) Select, copy, and paste another hole to similar position on other side of board.
- 8) Double click on each hole individually and set position so they are symmetrical.

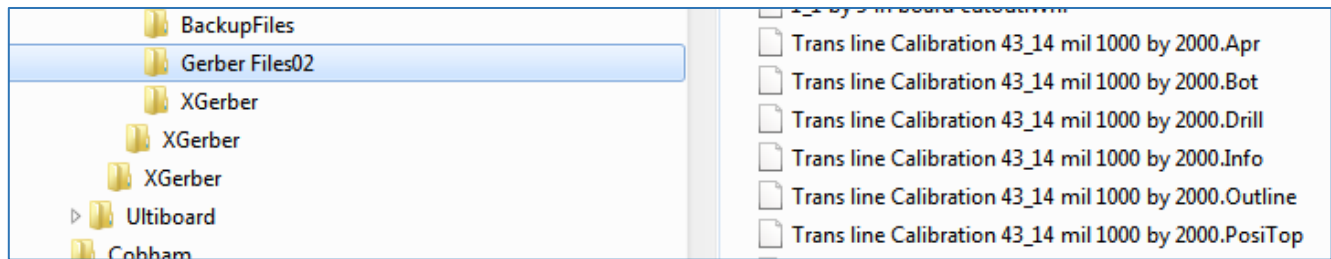


Create Gerber and drill files for CNC:

- 1) Under File menu, select *Save Project as...* then put in desired directory
- 2) Under file menu, select *Input / Output Formats*, then *Production*, then (X-) Gerber and drill output PCBout...



- 3) Use Windows *File Manager* to look in directory and you should see at least the following files (with same file extensions, but different names)



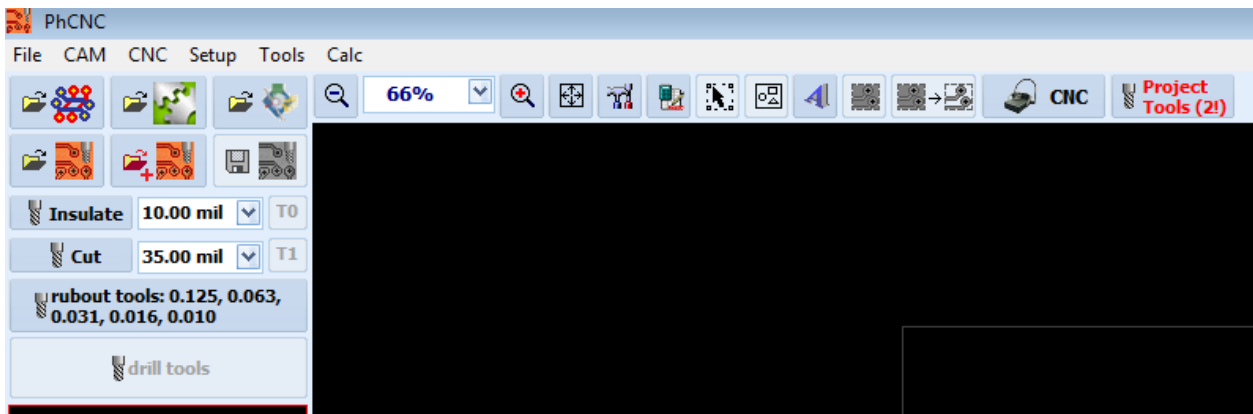
NOTE: If you have Gerber files created in another software package, they may look similar to this. You will probably need to see a file extension of “drl” to have correct format information for CNC control software. [Exelon II format]. For example:

Band 1 Gerber April 15	BRD_OUTLINE.gbr	4/15/2014 4:39 PM	GBR File	1 KB
DATA_SETS	drill.drl	6/5/2014 12:19 PM	DRL File	2 KB
Images	LAY1.gbr	4/15/2014 4:39 PM	GBR File	13 KB
Layout April 10	LAY2.gbr	4/15/2014 4:39 PM	GBR File	19 KB
Layout March 28	MASK-NS.gbr	4/15/2014 4:39 PM	GBR File	10 KB
TEMP	SLK-NS.gbr	4/15/2014 4:39 PM	GBR File	22 KB
DARPA				

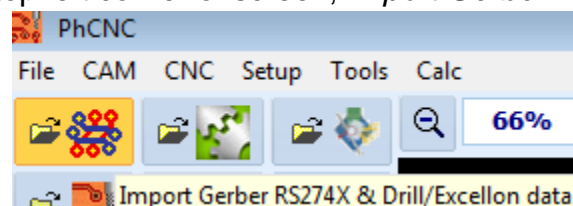
4) Close *Target* software and start *PhCNC* software.

Part B. Using PhCNC and Accurate CNC: Setting up Software to Prepare for Cutting Board

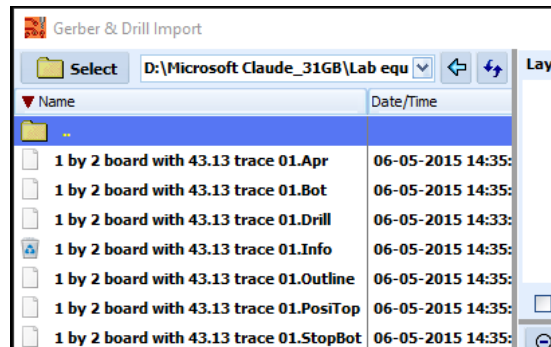
1) Opening screen of PhCNC will look similar to this:



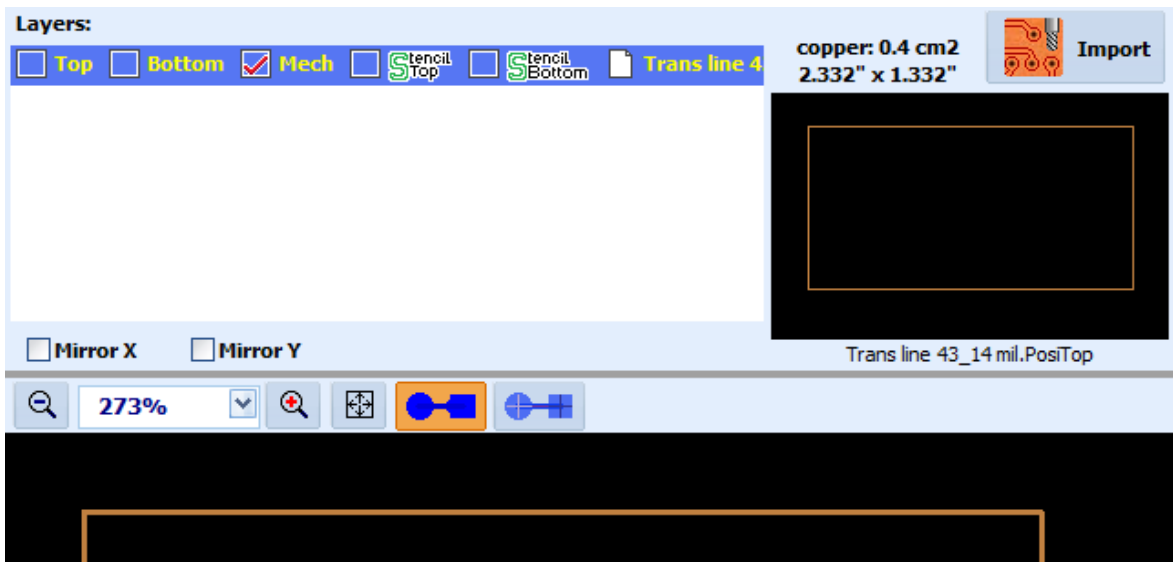
2) Click on icon at top left corner of screen, *Import Gerber RS274X...*



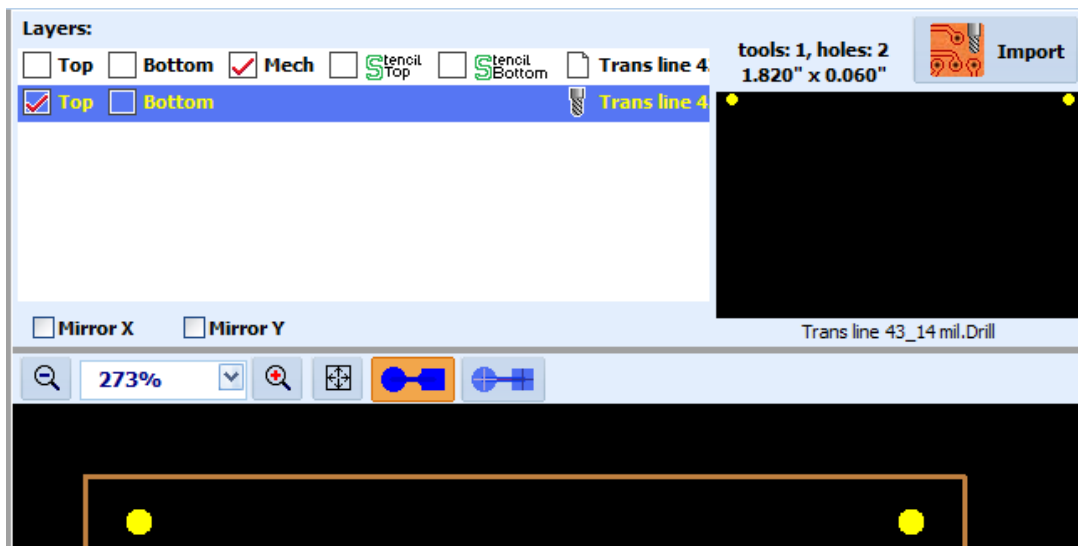
- 3) Click on folder icon [left side] and set directory as above, to see your files



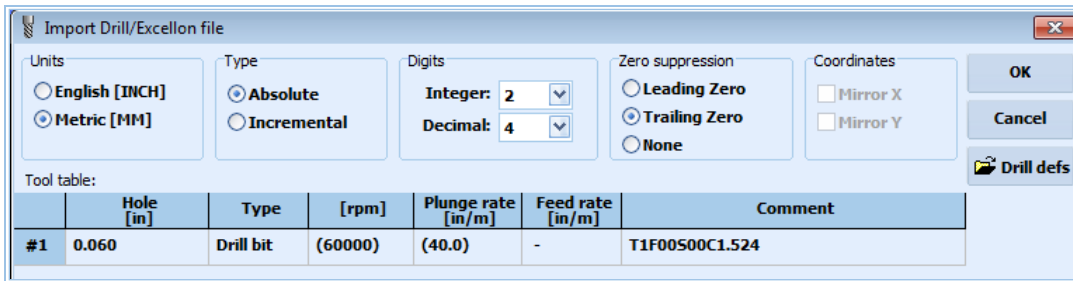
- 4) Click on file with .PosiTop extension to load it first. For this example, the inner rectangle is to be used to cut board outline, so click in *Mech* box to load it as Mechanical Layer and have editable features.



- 5) Click on file with .drill or .drl extension to load it and select *Top* layer box.



As soon as you click on *Top*, the drill bit configuration menu will come up.



Import Drill/Excellon file

Units: ☐ English [INCH] ☒ Metric [MM]

Type: ☒ Absolute ☐ Incremental

Digits: Integer: 2 Decimal: 4

Zero suppression: ☐ Leading Zero ☒ Trailing Zero ☐ None

Coordinates: ☐ Mirror X ☐ Mirror Y

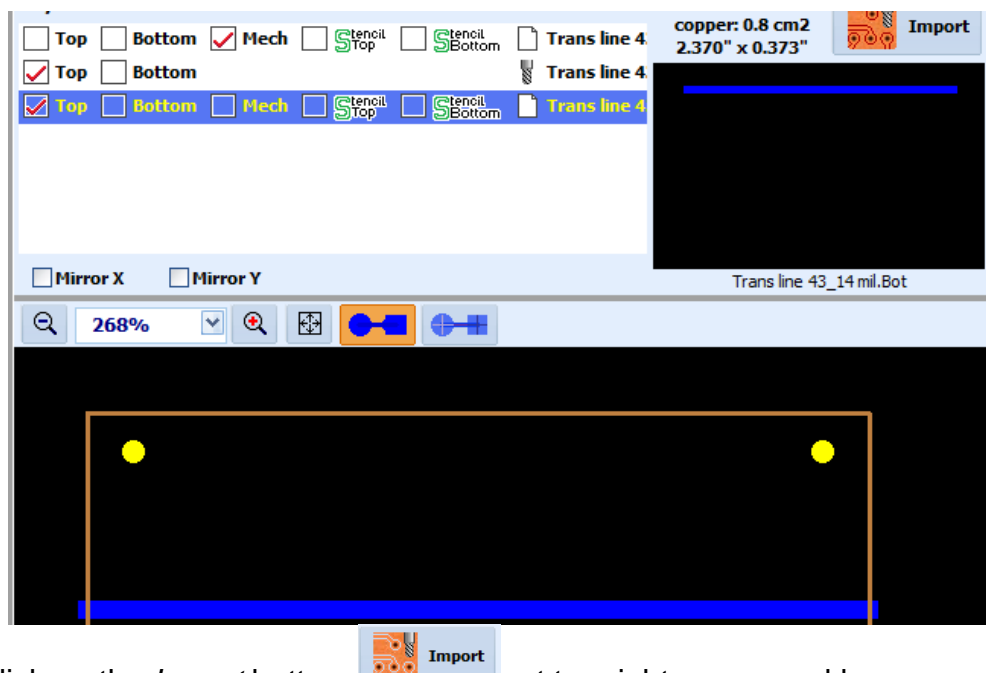
OK Cancel Drill defs


Tool table:

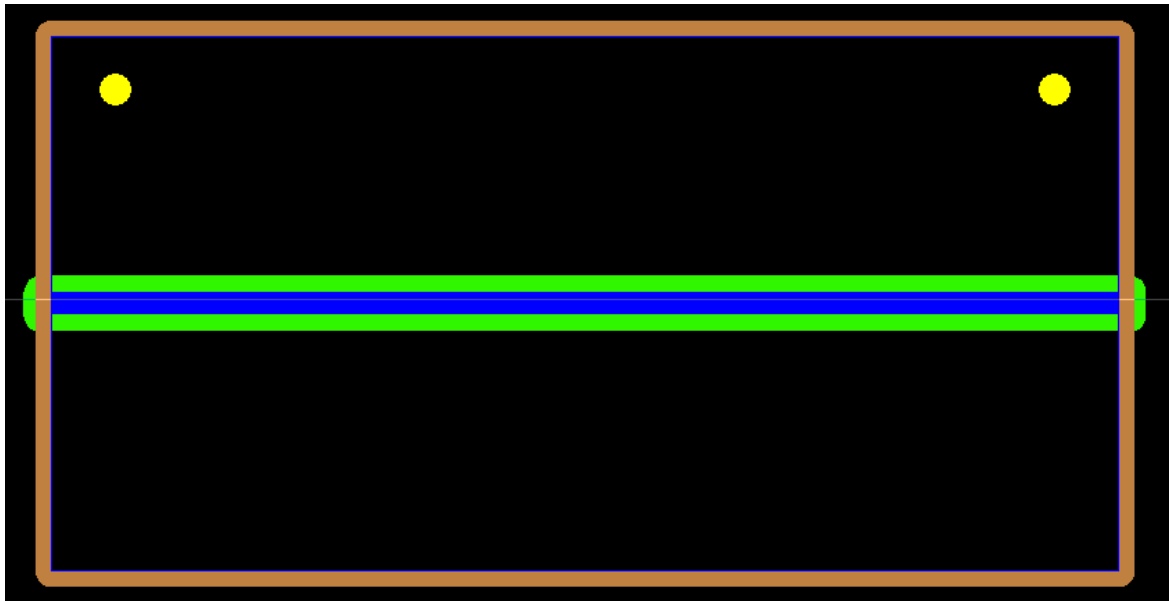
	Hole [in]	Type	[rpm]	Plunge rate [in/m]	Feed rate [in/m]	Comment
#1	0.060	Drill bit	(60000)	(40.0)	-	T1F00S00C1.524

Drill is correctly shown as 60 mils. Click on *OK* button. [set other features later]
Screen will change to show drill layer from *Top* side of board:

- 6) Click on file with extension, *.Bot*, and select *Top* box.



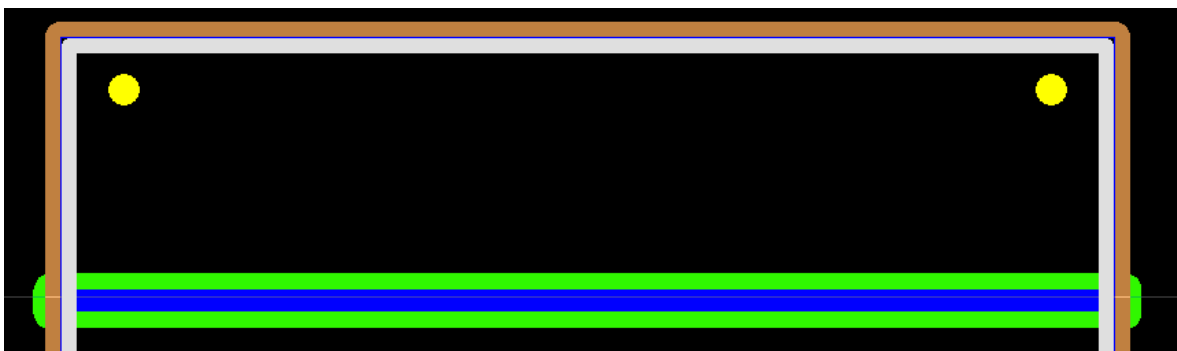
- 7) Click on the *Import* button, , at top right corner and lower screen will now include all 3 files for this project. Screen may be different, depending on settings most recently used. Image will be very small and you will need to click on zoom widow at top to enlarge image. Do it in steps and drag image to center of screen each time or it will go off screen.



8) Click on edit icon at top of screen with triangle, circle and square in it:



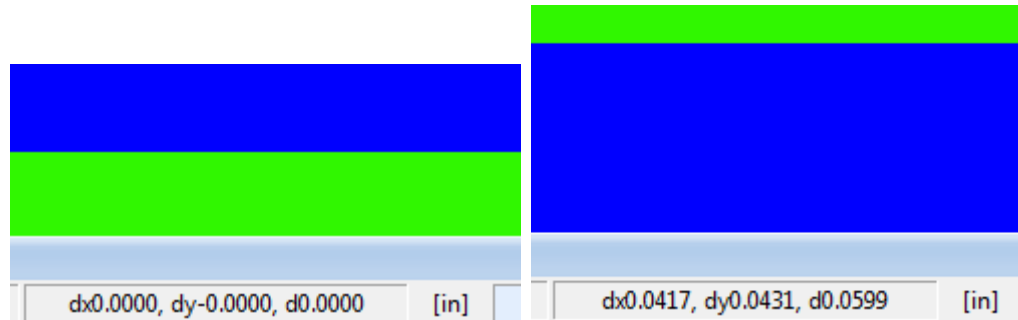
9) Now image of *Mech* layer will change to show options [white and orange line images]. Click on an image to turn it on or off, allowing you to choose which line to cut. Then click Edit icon again to finalize your decision. After accepting choices, only the orange/brown line will remain to be cut.



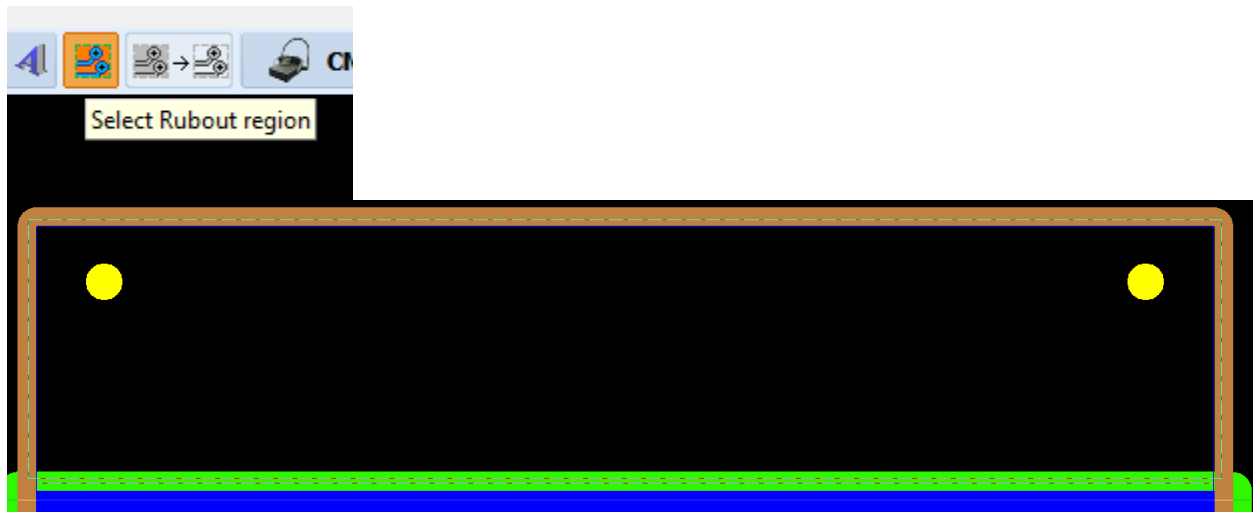
NOTE: The appearance of image on screen will depend heavily on what tools have been selected for cutting. Click on buttons on left top side of screen to set bits to be use for cutting.



- 10) Verify line width: Zoom in to larger magnification [~ 6000] and place cursor carefully on bottom edge of inner blue transmission line while pressing space bar key. Readout position at bottom of screen will indicate dx and $dy = 0$; move cursor to top of blue line and dy will correctly change to 0.0431 or 43.1 mil.

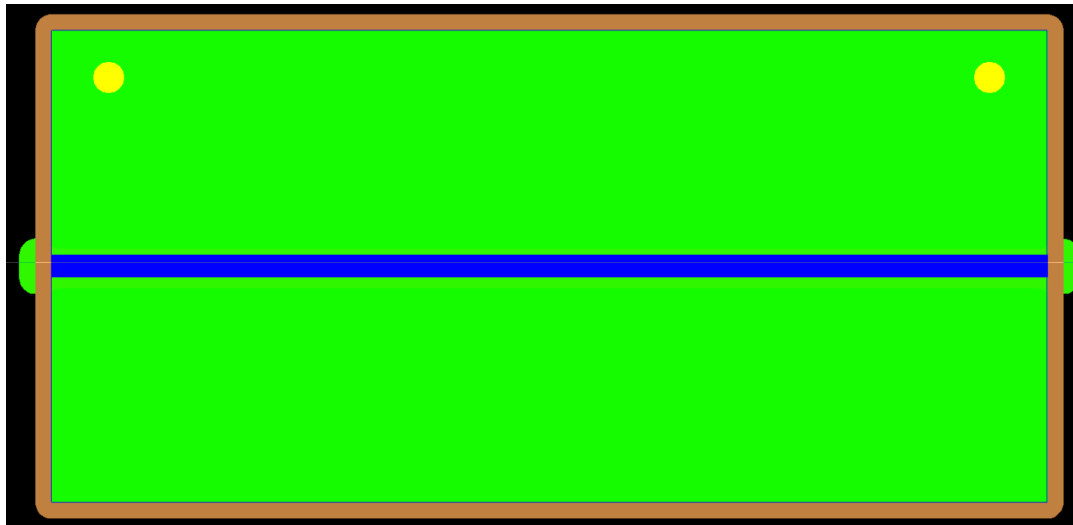


- 11) Using lower magnification, you can measure length of transmission line. [inside PCB outline box]
- 12) Select Rubout region: Do this one half of area at a time => point mouse to upper left corner of area to be rubbed out, then hold mouse down and drag cursor to lower right corner. Release mouse and click on icon just to right of icon used to enter rubout mode.



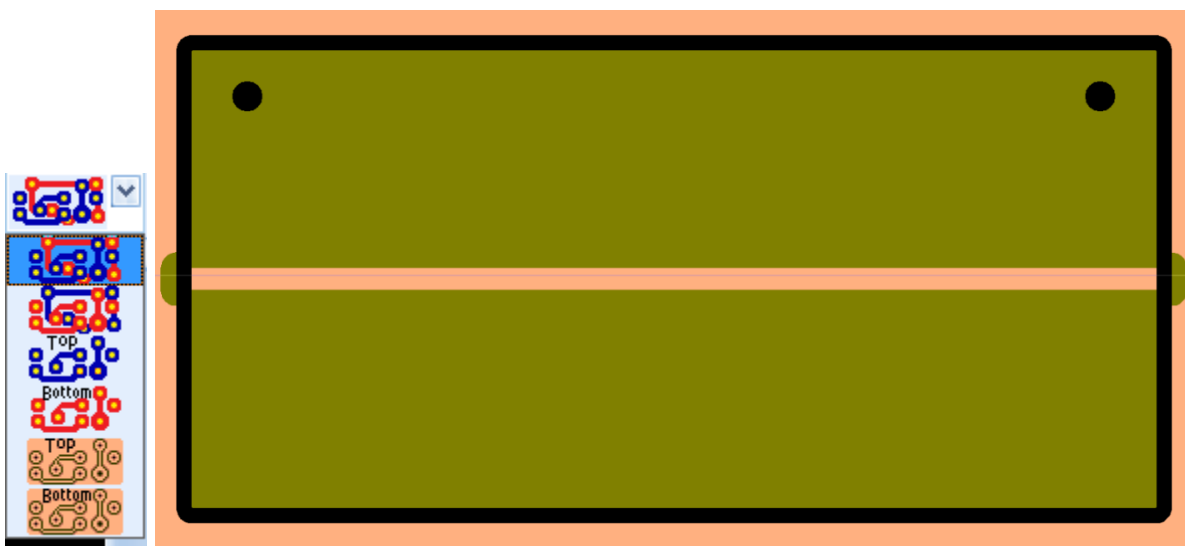
- 13) First section of rubout is shown below. Now repeat process for lower section of box to get full rubout as shown.





14) Click on icon at top right side of screen and choose option second from bottom, labeled TOP. This will allow you to see “copper” left after machining.

15)



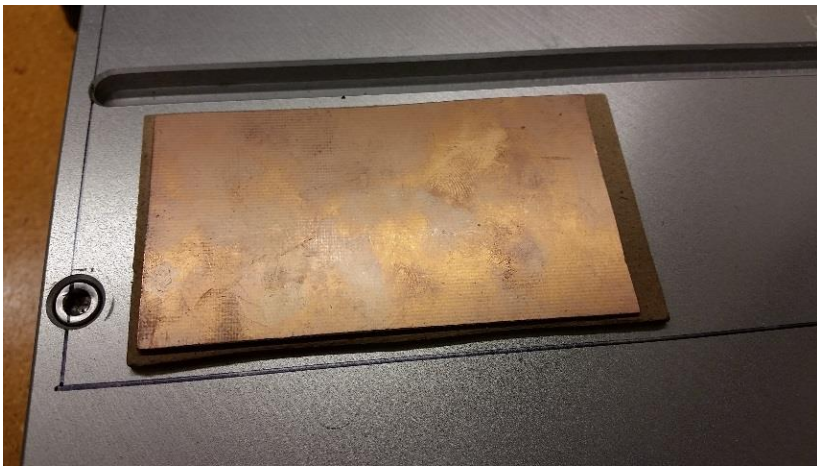
Cutting a Board

1) First, you may want to cut board to general size using a band saw.

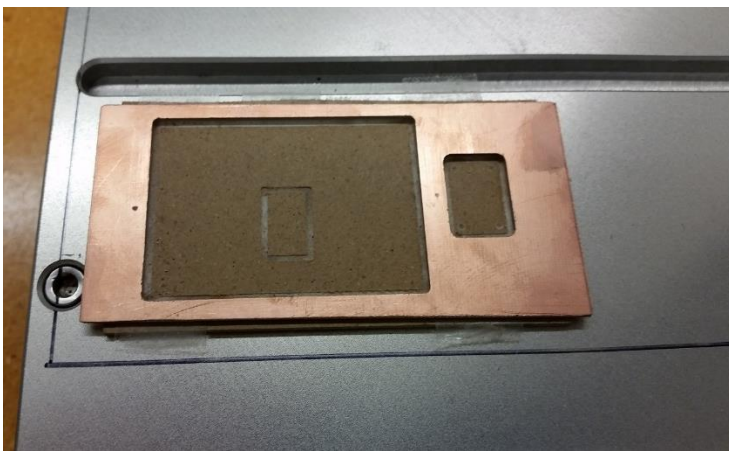
NOTE: Take into consideration where you will put the board during cut. It is wise to mark the visual limits on table with a felt tip pen [see below] so you can put the board where it will be visible in the AccurateCNC microscope camera. At minimum, you must place board where it will be possible to see alignment marks in camera.

2) Always clean bits before use and check under a microscope for sharpness. When cutting very hard material, the bit, even though made of carbide, may dull a little bit after cutting just a few inches. This can reduce the quality of edges of the cut. In most cases this will not be significant, but is sometimes critical.

- 3) It is also important to always clean bottom of boot with Q tip and alcohol before use. Otherwise the boot can drag material across the board and leave scratches or make depth of cut imprecise. It is good idea to use the built in microscope to check quality of cut often. Changes in bit speed, feed, etc. can make big differences in quality of cut, and it will also change as bit become less sharp.
- 4) If your bit is causing significant curf around cut, you may want to consider cutting in multiple passes, and clean board with Scotch Bright between cuts to keep depth of cut consistent.
- 5) Drill fiducial alignment holes, using either .060 or 0.0625 center cut end mill. For doing this, place piece of fiber board on table with regular double-edged tape. Then attach your board [to be machined] to the fiber board, also using double edge tape. [Vacuum chuck can also be used.]



Attach fiber board and PCB to table with double edged tape.



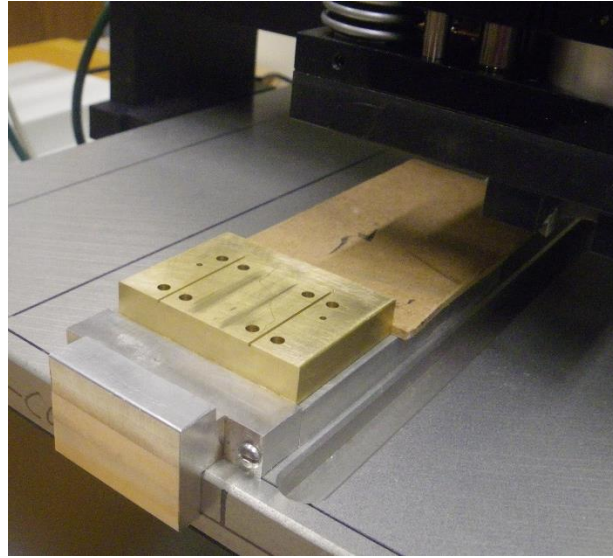
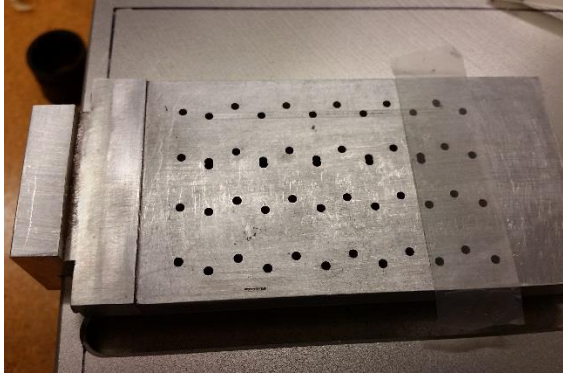
Two sections of this board have been cut out and removed. Note fiducial holes outside part of board to be used. Sometimes it will be necessary to put fiducials inside area used by circuit. Get best accuracy by putting fiducials far apart and on X axis as shown.



Label entire table, showing cut and camera marks.

NOTES:

- a) Depending on your design, you may want to cut out main outline of board just after machining fiducials. Think about advantages/disadvantages.
- b) After board has been cut to approximate size you can easily and quickly do multiple precise operations on a work piece. That means you can quickly position it with respect to your G code by using optical fiducial markings.
- c) If you have a pattern on both sides of board, you will need to drill fiducials all the way through board, otherwise you can drill only part way. If you have the "full surface vacuum attachment and a larger board, you can put it anywhere. This photo shows one version of a separate vacuum hold down that works well for all but the smallest boards and also works for machining soft metal shown here. The bracket on the end serves 2 purposes: first to align hold down jig with table edge, and second, there is a thin lip to press against one side of work piece to improve hold down strength and reference initial alignment.



Use tape, paper, or fiber board to cover unused vacuum holes.

6) Thru-hole [plated] vias:

- a. Cut all needed patterns on both sides and clean up board, then prepare to drill holes for vias. Since you will be pressing conductive silver paste through the holes to make electrical contact between layers, it will be necessary to mask the remaining part of circuit board. This is done with a plastic film having adhesive on one side.
- b. Carefully place your masking film on both sides of board
- c. Use scalpel or razor blade to cut out film around fiducial holes so you can see them clearly for alignment
- d. Mount part back on machine and use fiducial holes to align board with board pattern.
- e. Clean bits and boot of machine
- f. Place a sacrificial part underneath board [for example, 1/8" MDC] and set depth to drill all the way through board, with consideration given to angled part of bit and thickness of masking layer.
- g. Systematically drill all holes.
- h. Remove board and coat one side with silver paste. Use a squeegee to make sure silver is spread evenly and goes into all holes.
- i. Place board on vacuum mount to suck silver paste through holes.
- j. Repeat on other side.
- k. VERY carefully remove mask layer to avoid breaking board
- l. Clean "mistakes" with Q-tip slightly wet with alcohol or acetone
- m. Bake at 160 °C for 30 minutes
- n. Remove and clean board
- o. Electroless plate about 1 minute in tray with Liquid Tin [MG chemicals], available from Mouser Electronics. This coating will enhance solderability and protect board from oxidation.

NOTES:

- 1) Reference tool/bit speed [starting point for your tests]

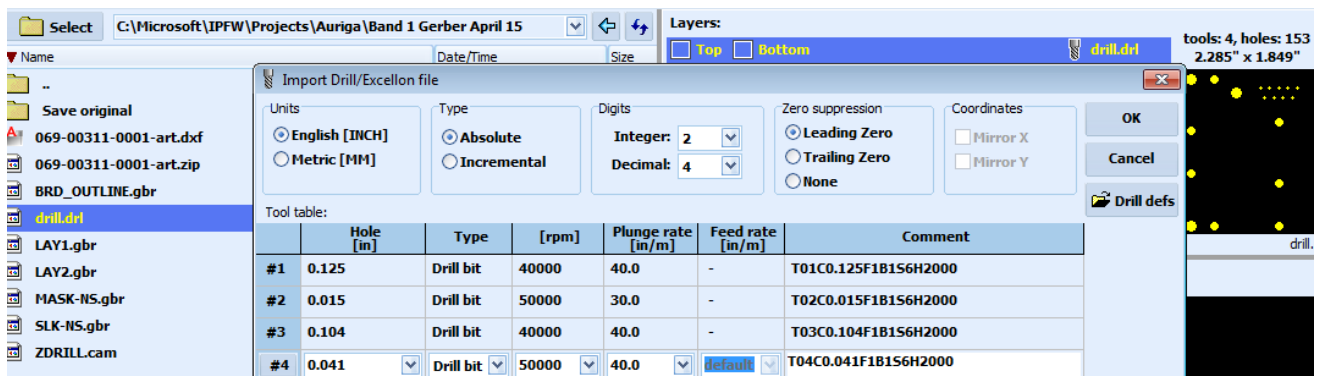
10 mil fishtail carbide end mill	60k + rpm	
20 mil fishtail carbide end mill	50k rpm	
50 mil fishtail carbide end mill	50k rpm	
125 mil (1/8")	40k rpm	
V 45° use 60 k rpm, 15 in/min		

Best cut typically at about 3 mil depth for 1 oz pcb material

Feed 15 in./ min., plunge (down) 40 in/min. but reduce to 20 or 30 for 10 mill bit
15-20 in/min for very smooth hole drilling fiducials and use center cut end mill

- 2) **Import Drill file:** The drill file needs to be Excellon(2) format and can have extension with .txt or .drl. Import under Gerber option. When you click in box for Top layer the following menu will come up with sizes and settings for drill bits. Most likely, plunge rate and rpm will be set to default and may need to be changed as shown. Check to make sure size of board is correct and also the number and size of holes. Drill file is also shown below options table. If it is Excellon2 it will include bit definitions as shown here.

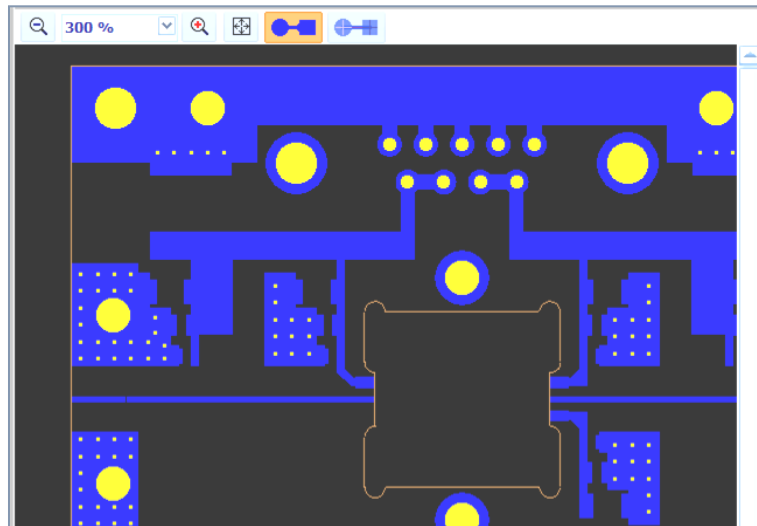
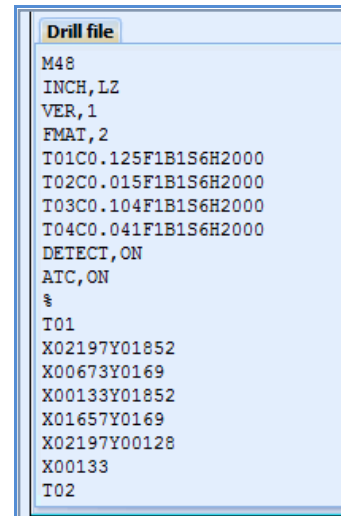
In order for drill file to be aligned properly with top layer detail, it must be imported as part of one "Import" operation=> cannot import top layer first and then later import drill layer.



- 3) Check alignment of drill holes with main pattern to see if drill file has loaded correctly. In order to “**Load**” a file, you must select it on left list and then select a position option box=> *Top, Bottom, Mech...* **Load** both drill file and pattern file, then zoom and compare in Gerber widow. If correct, then click **Import**,



. (Top right corner of screen.)









Orange= cut; Blue = pattern to stay; Yellow=drill hole

- 4) In CAM mode, bottom layer files will show up aligned with Top layer files. When the switch is made to CNC mode for machining, the bottom layer images will be flipped on X-axis and appear to be in “wrong” position. The position is corrected by using fiducial marks. [Of course top and bottom layers are machined separately and require that board be flipped and re-aligned.] The software will allow only one layer to appear at a time in CNC mode, so you have to turn off Top layer in CAM mode and turn on Bottom layer before switching to CNC mode.
- 5) Suggestion: load cut file as a *Mech* layer along with layer 1 (*Top* layer) and Drill (*Top* Layer). Then import these. After importing all *Top* Layers, load *Bottom* layers and import them. If needed, layers can be aligned in CAM mode, and then machined in sequence in CNC mode. Layers can be turned on and off as needed during machining process. It is also possible to zoom in

on a window to machine separate parts of board. It is a good idea to “test machine” a small part of board first, before machining entire board.










Reference material for PhCNC screens:

1) Viewing your layout in PhCNC: [Top in Blue, Bottom in Red]

- a)  - top, bottom, drill & mechanical (natural layers order)
- b)  - bottom, top, drill & mechanical (bottom layer on top)
- c)  - top, drill & mechanical (bottom layer hidden)
- d)  - bottom, drill & mechanical (top layer hidden)
- e) Buttons  &  are for a special visualization mode called “copper view”. Screen shows what should be the final result after executing all machining programs.

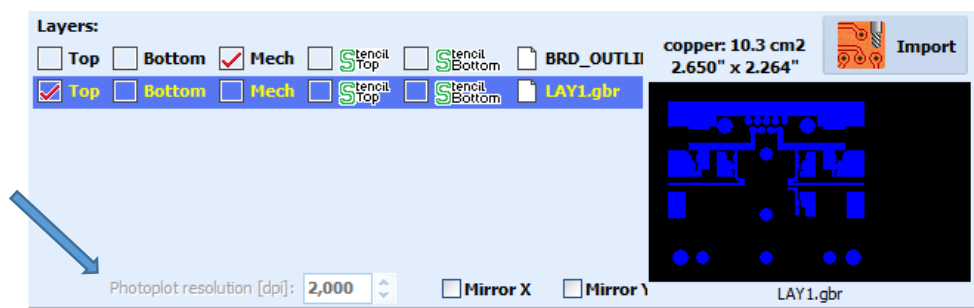
First image below is for mechanical layer, center and thickness are same color, second is a line on top layer, cut on both sides.

- f)  display the tool path center line only 
- g)  display the tool path real thickness 
- h)  display the tool path real thickness & center line 
- i)  display the tool path real thickness & Gerber [blue line] 
- j)  display the tool path real thickness, center line, & Gerber 
- k)  **[new] display tool path center line and Gerber** 
- l)  display the Gerber only 
- m)  display the tool path center line only & Gerber vectors 
- n)  display tool path real thickness & Gerber vectors 

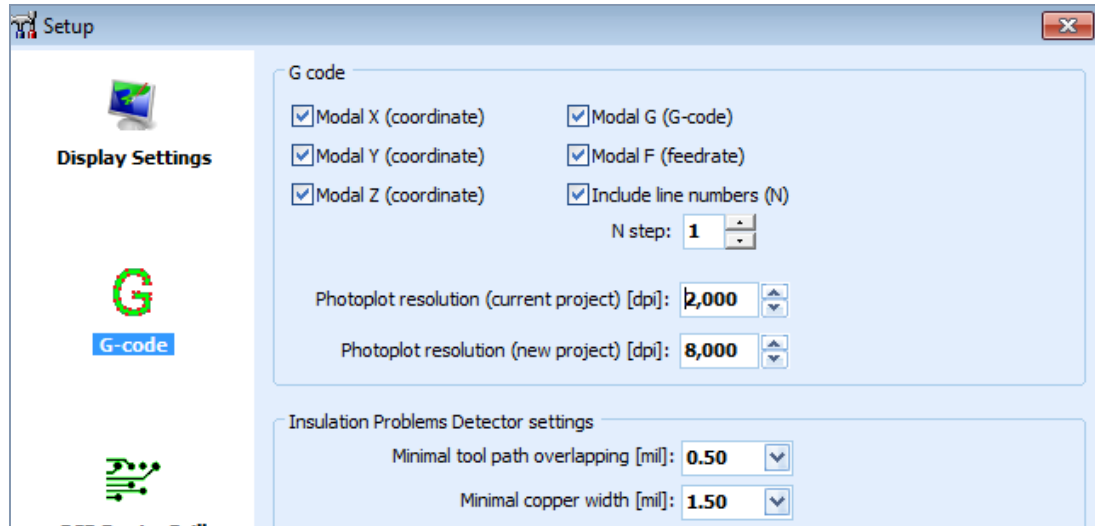
- o)  display tool path real thickness, center line, & Gerber vectors, same
- p)  display the tool path real thickness, Gerber vectors, & Gerber, same
- q)  display real thickness, center line, Gerber vectors, & Gerber 
- r)  **NEW => center line and Gerber vectors & Gerber** 
- s)  display the Gerber vectors and Gerber  

2) Use of PhCNC (including Emulation Mode):

- When head is outside limit switches, then small LC with yellow highlight appears in bottom ribbon of screen
- Insulation:
 - First cut of insulation is with engraving (angle) bit, about 3-5 mill width [if present, otherwise uses smallest bit in holder]
 - Second is about 2 x diameter of first
 - Continue increasing by 2 x for others (will be 1 mil overlap in cut)
- Up to June 2014, PhCNC uses raster basis for CNC vectors, increase Photoplot resolution (dpi) during import to improve resolution. To get 1 μ , will need a about 10,000 dpi.



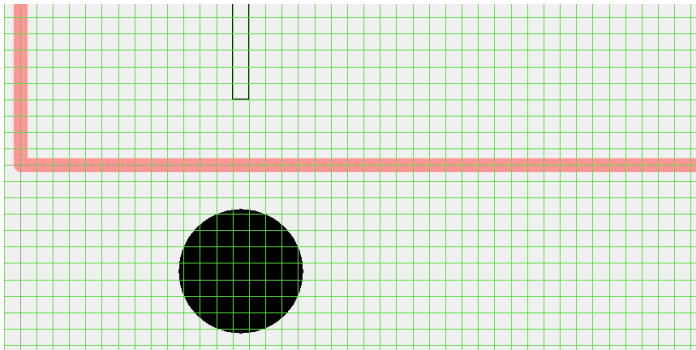
- d) To setup Photoplot resolution, you must set it in defaults before or after starting a project. Shown below, the resolution is 2000 dpi for current files (current project) and 8,000 dpi for next project (new project).



3) Create Fiducial/"centering" mark(s) in Target 3001

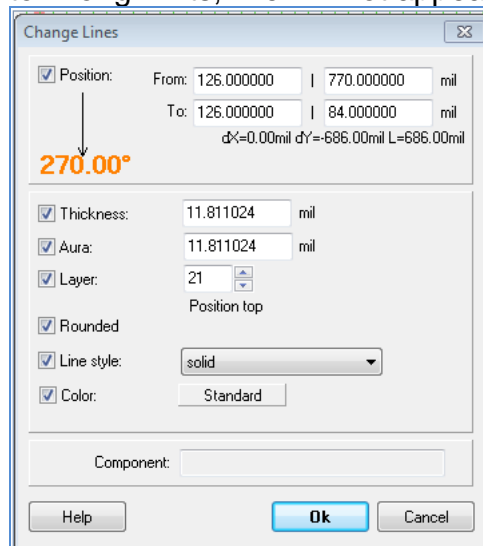
- Open pencil tool menu
- Select circle on 5th line
- Select the placed Fiducial => Properties menu
- Set radius of hole

Preferred location of Fiducials is on x axis [horizontal] and outside of PCB to be made, 60 mil or similar stub end mill is best. If you will need to do more work on board after it has been cut out, then you **MUST** find suitable location for fiducials inside the board.









Here, the smaller line [rectangle] is border of mechanical layer definition of 14 mil line. The orange line surrounding it is edge of PCB to be cut out. The black circle below is a 60 mil fiducial mark.

- 4) Draw box around object and right click it, then select *Properties* to see/edit its properties. A typical default line width is about 11.8 mils, as shown. Line above is set to 3 mils, there are minimal [visible] limits that may change with magnification. If set to wrong limits, line will not appear.



- 5) Board [cut] outline must be imported into PhCNC under Gerber as *.outline* extension and put onto Mech layer.
- 6) CNC mode instructions within PhCNC software:

<p>Program: Drill (Bottom)</p> <p>RUN (GO) Stop</p> <p>MDI:</p> <p>Open Compile</p>	<p> show spindle path (raw data)</p> <p> clear spindle (raw data) buffer</p> <p> show CAM data</p> <p> show processed G code path</p> <p> draw to cursor (G code)</p> <p> draw all (G code)</p> <p>Open G code Program</p> <p>Compile G code</p>
--	--

- 7) Example, flash apertures: The Gerber code was originally used photo flash and flash aperture to define the size of objects exposed or “flashed” by Gerber file:

Aperture Definitions				
D-code	Shape	SizeX	Size Y	
d10	round	0.010		
d11	square	0.030		
d12	rect	0.060	0.020	
d13	thermal	0.050		
d14	oblong	0.060	0.025	

- 8) Tool path and centerline in PHCNC, 600 % 0.175” x 0.800,” outside cut



- 9) Same data with cut on inside and outside

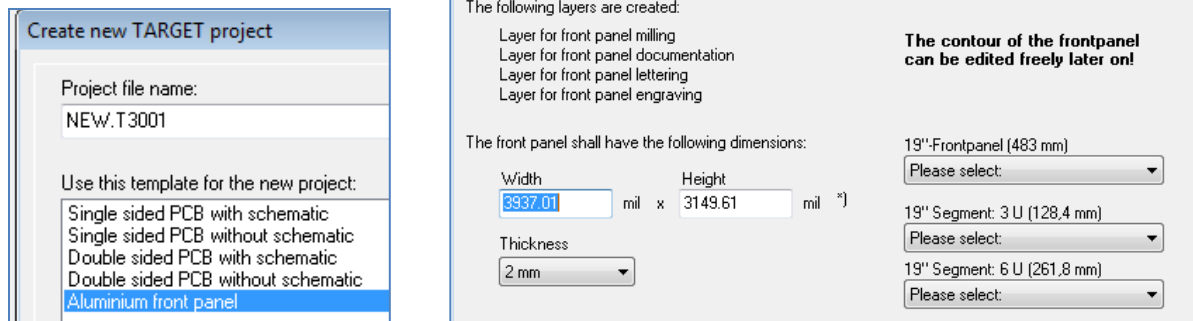


- 10) Rectangle duplicated one with inside other outside cut

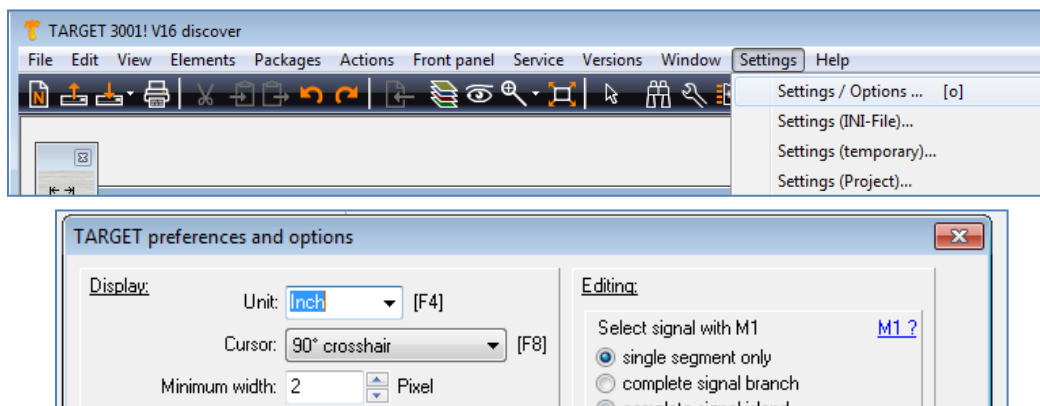


Part C: Aluminum Front Panel Module of Target 3001

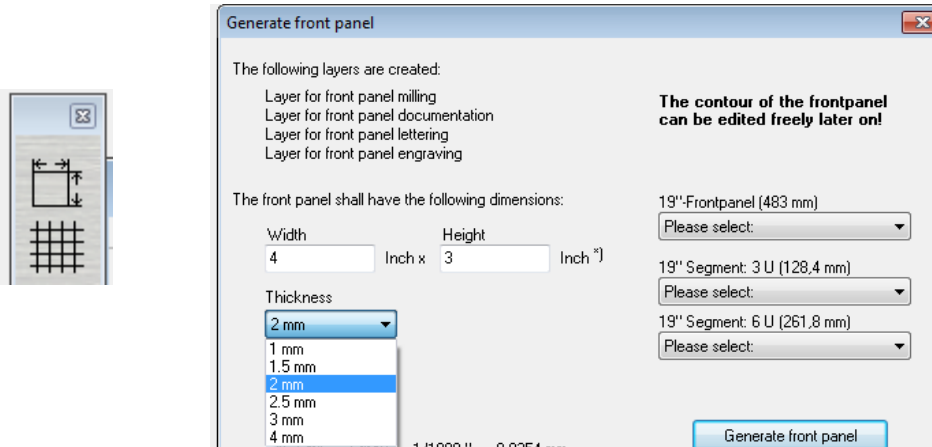
- 1) Startup with new Aluminum Front panel



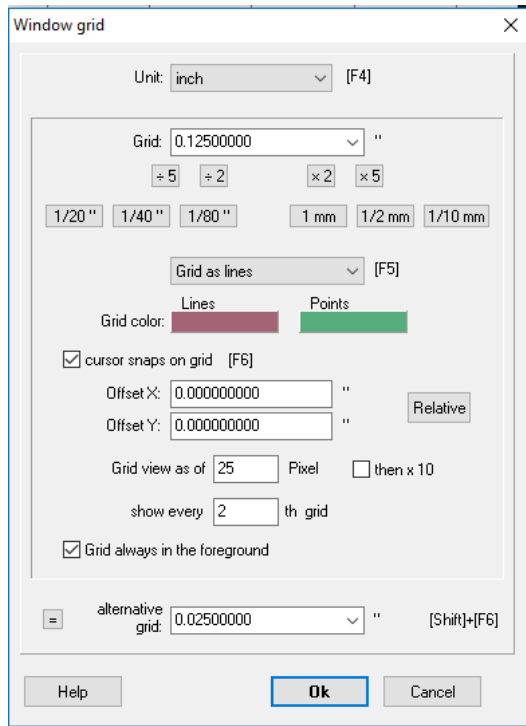
- 2) Default menu will come up asking for panel (metal) dimensions and thickness. Several “standard” options for 19” rack panels are on right side, but can be ignored. If you want to specify values in inches, then cancel this initial menu to get a new worksheet. Click on Settings Tab at top of page and select *Settings/Options...* then set *Unit* to *inch* and click *OK*.



- 3) Click top symbol (square) to bring up front panel size in inches. Thickness has only mm options.



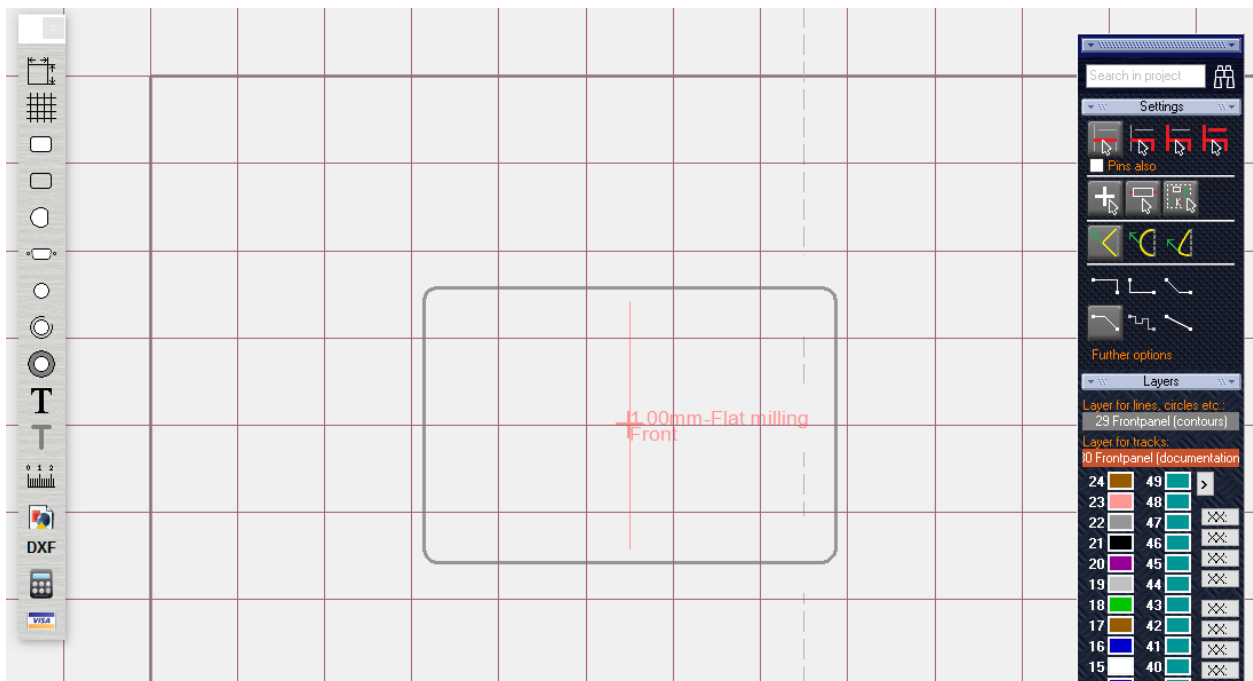
4) Click on cross hatch pattern to set grid spacing(s)



Options for grid on AI front panel:

- 1) Units can be: mil, inch, mm, μm , nm
- 2) Grid: 6.25mil, 12.5, 25, 39.7008, 20, 50
- 3) Grid shows as points, lines, or invisible
- 4) Cursor snap to grid
- 5) Color
- 6) Offset
- 7) Show (grid) every multiple of snap grid
- 8) Alternate grid, shortcut shift/F6 to change grid spacing

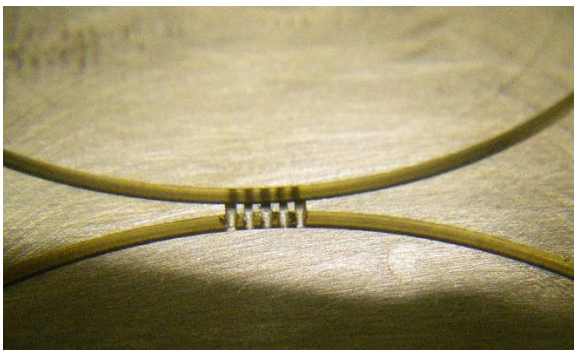
Resultant opening screen in Target3001: [Also see tutorials under Help in Target]:



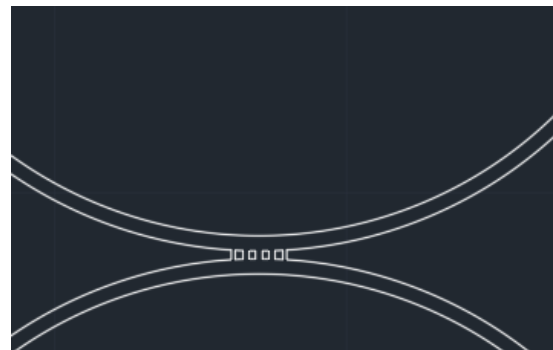
Part D: Tips and Tricks, plus other applications: [more details to be in second edition of this manual]

The AccurateCNC machine can be used for very complex multilayer PCB's, both low frequency and microwave. It can also be used for a wide variety of metal and other material that needs precise machining. Millimeter and sub-millimeter waveguides are extreme examples that traditionally use equipment costing ten times as much and are operated only by uniquely experienced machinists. These other machines can require specialized rooms that need to support very heavy loads and use lots of space.

Some examples of parts made on AccurateCNC:

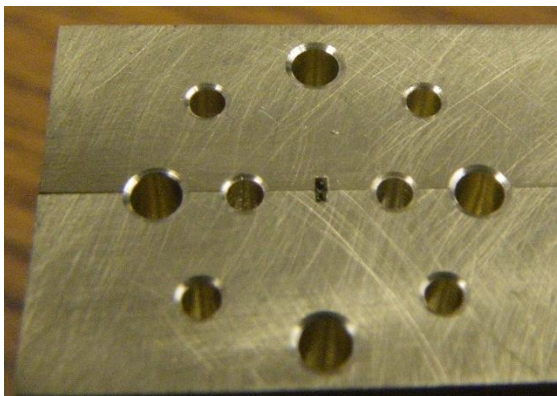


Small channels for 200 GHz coupler are about 6 mils wide and 27 mils deep. Material is free machining brass

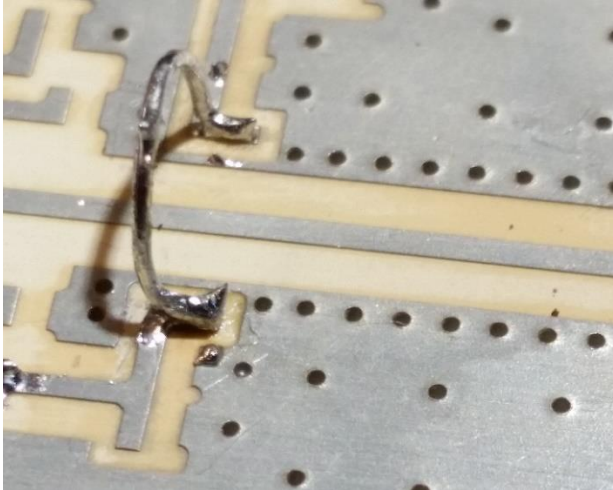


AutoCAD drawing used by AccurateCNC to cut pattern.

Arcs shown are just a simple section of a circle in AutoCAD. When translated to AccurateCNC, the arcs were cut first with a “larger” bit of about 20 mil diameter, then microscopic sections were cut very slowly using a 5 mil bit. Best results are achieved if the small sections have as much mass as possible while being cut, to reduce vibration. Runout on both bit and spindle also influence quality of cut. With great care, bits as small as 4 microns in diameter can be used.



Due to the great difficulty machining very small sizes with high aspect ratios, millimeter-wave waveguide devices are usually machined in “split block” mode: two halves are made separately then bolted together. The waveguide above is shown “end on” here, along with holes for mounting waveguide fixture.



For high frequencies and very wide bandwidths, coplanar waveguide often gives vastly superior performance in comparison to simple microstrip. In order to get that performance, the RF via “fence” ground plane plays a critical role. The circuit to the left has good performance to above 22 GHz. The highest frequency of use determines the size, spacing, and position of vias shown.

Even a small board can have hundreds or even thousands of via holes, so an automated method of drawing, drilling, and metalizing the vias is essential. The late 2017 version of Target3001 [V19] includes a specific tool for automated creation of RF via fences.

Best bandwidth/high frequency performance of coplanar waveguide depends directly on several factors as well indirect dependence on several others. Since “ideal” performance depends critically on quality of ground plane conductor, which is created by vias between the top and bottom layers of the board, these vias are the critical factor when all other factors are held constant. Assuming that vias are adequately plated through with good quality conduction material the two remaining factors are the distance between outer edges of 2 vias and the spacing between rows of vias. Some authors have recommended the spacing between 2 vias be about 1/10 of the wavelength of the transamination line at the highest frequency of interest. Although it was done with a single board type and transmission line, perhaps the most extensive experimental determination of the spacing was done by Bill Rosas and he found the dimension to be a little bit less than $\frac{1}{4}$ wavelength, but also found that the spacing between rows was important and had considerable influence on the high frequency end. It would be expected that the size of via holes might modify this data somewhat, at least indirectly, as it would change the shape of cavity between vias. My research has also found that size and spacing of vias can substantially effect the mechanical strength of board. If difficult to obtain adequate ground wall with one row of vias on either side of transmission line, then a second [offset] row behind the first one will help. Other physical dimensions for transmission line can also influence performance, and this might be compensated by decreasing the spacing as much as practical, but the 1/10 figure is likely to be overkill and unnecessarily degrade other properties of system.

References:

Bill Rosas, *Optimizing Test Boards for 50 GHz End Launch Connectors*, Southwest Microwave, Tempe, AZ., 2007

Martin Dresler, Oscar Solano Verigy, and Heidi Barnes, *High Density RF Loadboard Design*, Verigy, Boeblingen, Germany and , Cupertino, USA, 2009

Harald Friedrich, Various Manuals and personal communications regarding Target 3001 software, I.B. Friedrich, Germany, 2010 to 2017

Chris Ovtcharov and David Delagrange, AccurateCNC operations manuals and many personal conversations. Accuratecnc, Fort Wayne, IN 2012 to 2017.

Claude Setzer, *Microwave Amplifier Design*, First Edition, Engineering Depot, Inc., Fort Wayne, IN, 1989 to 20012