T|TECH

Quick Circuit Prototyping Systems

ISOPRO SOFTWARE USER'S MANUAL

T | TECH
Quick Circuit Prototyping Systems

Developed and Manufactured by: T|Tech, Inc.
Version 4.8 Date: 2009

Published by: T-Tech, Inc.
510 Guthridge Court
Norcross, GA 30092

Phone: 800.370.1530
Phone: 770.455.0676
Fax: 770.455.0970
Email: info@t-tech.com
Web: www.t-tech.com

REGISTRATION SHEET

Thank you for purchasing your IsoPro Software Manual. Please take a few moments to fill out and return this page to T-Tech, Inc. This will ensure that your warranty will be effective. In addition, by registering as a Quick Circuit end user, T-Tech will be able to supply you with future system updates.

User Name ______________________________

Company Name ______________________________

Address ______________________________

____

____

Phone Number ________________ Fax Number ________________

E-Mail address ______________________________

Quick Plate Serial Number ______________________________

Mail to:
T-Tech, Inc.
510 Guthridge Court
Norcross, GA 30092

Or Fax to: (770) 455-0970

Table of Contents

ISOPRO SOFTWARE DESCRIPTION

ISOPRO programs the Quick Circuit system to drill, mill and route your circuit board design. The number of computers on which ISOPRO can be installed depends on the license key you purchased. Only the computer connected to the controller will actually be initialized to run the milling table.

The sample files used in this tutorial are on the installation diskettes on the CD. Your CAD files are translated into Gerber plot files for the CAM process. A general understanding of Computer Aided Manufacturing (CAM) and Gerber files in particular is needed to use the IsoPro software. Refer to the Appendix A: Gerber and CAM essentials, and photoplotting principles for further detailed explanations.

ISOPRO SOFTWARE INSTALLATION

The ISOPRO software is copy-protected with a hardware protection key that resides on the computer's parallel or USB port. This software allows users to import the Gerber and Excellon drill files and calculates the needed isolation or mill paths from the circuit board data.

To install the IsoPro software:

Step 1: Insert the IsoPro CD

Step 2: Follow the instructions in the Setup program.

Due to Windows security and variation in operating environments you may have to install for everyone in multi-user environments. This is the "anyone who uses the machine" option. In some cases computers with multiple Iso-Pro users will have to re-register the machine for the individual user. Please contact support for further help on this issue.

ISOPRO SOFTWARE TUTORIAL

Overview

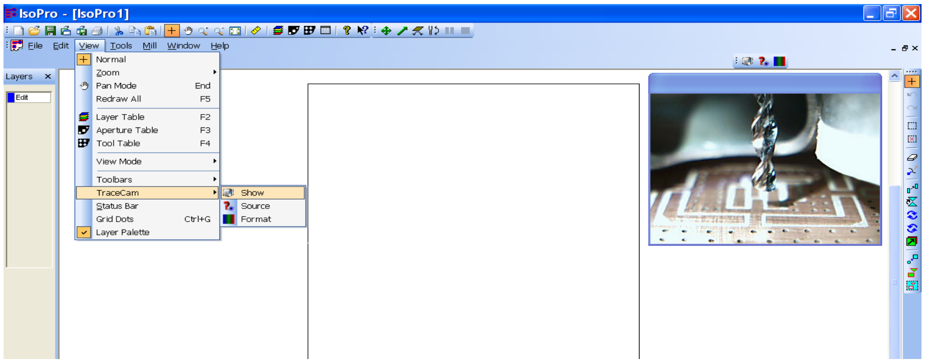
This tutorial provides a training procedure for using T-Tech's IsoPro software. The sample case involves a typical design with a component side, a solder side and a drill file. This tutorial describes how to:

- Load your CAD data (importing Gerber of DXF files.)
- Go to the layer list and ensure the bottom layer is identified as "solder".
- Register the component and solder sides
  (for this tutorial the component and solder layers are purposefully input as Gerber files that do <u>not</u> register with each other)
- Correctly size the tools required by editing the Tool Table
- Perform a clearance test
- Perform a 0.25 mm (0.010") and a 0.78 mm (0.031") isolation
  (for use as a mill path to outline all of the pads and traces on the board)
- Perform a 0.78 mm (0.031") rubout that can also be used as a mill path to remove all excess copper
- Create a board outline

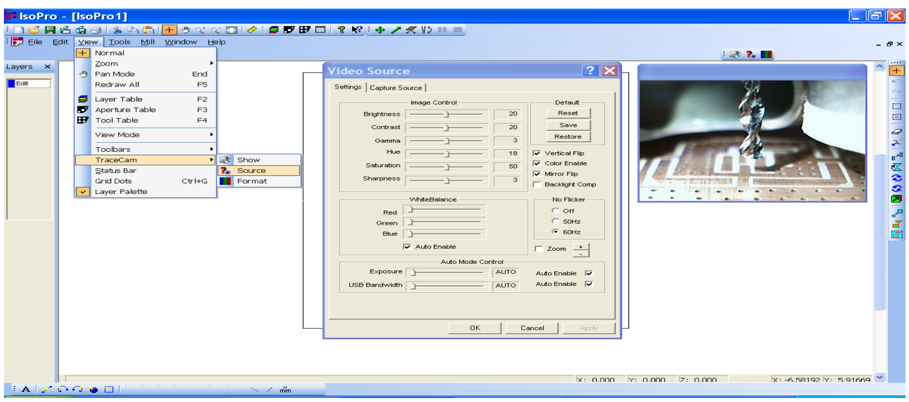
The hardware protection device [dongle] must be installed on the computer's USB port in order to fully use IsoPro.

Trace Cam Information

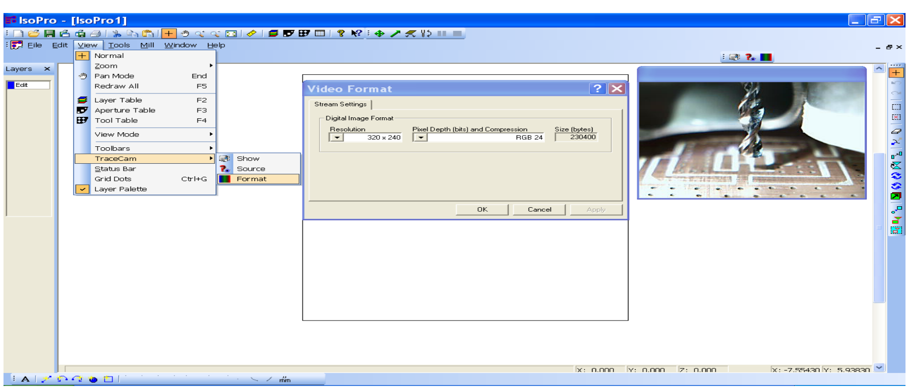
If you purchased a TraceCam with your IsoPro software, please refer to the specifications below to determine set-up and resolution and lighting control:



The source tab provides options to control the lighting and brightness controls of the camera and enables the user to change these video options.

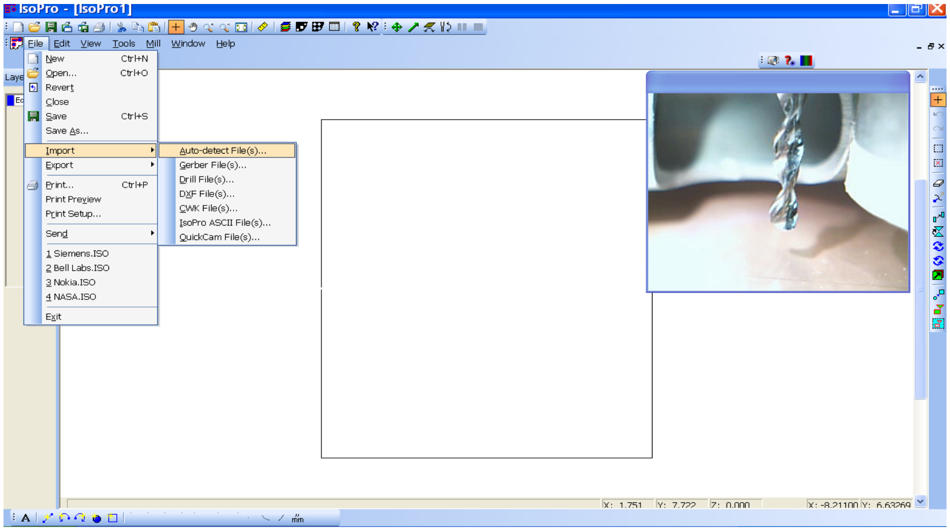


The format tab allows the user to control the resolution settings of the camera, T-Tech recommends using the 320 x 240 resolution, however, the camera can be set at 640 x 480.



Importing Gerber Data Files

Step 1: To import your CAD design files, select File > Import from the tool bar. We recommend that you use the Auto-detect File(s) feature to load your files. Click on Auto-detect File(s)...

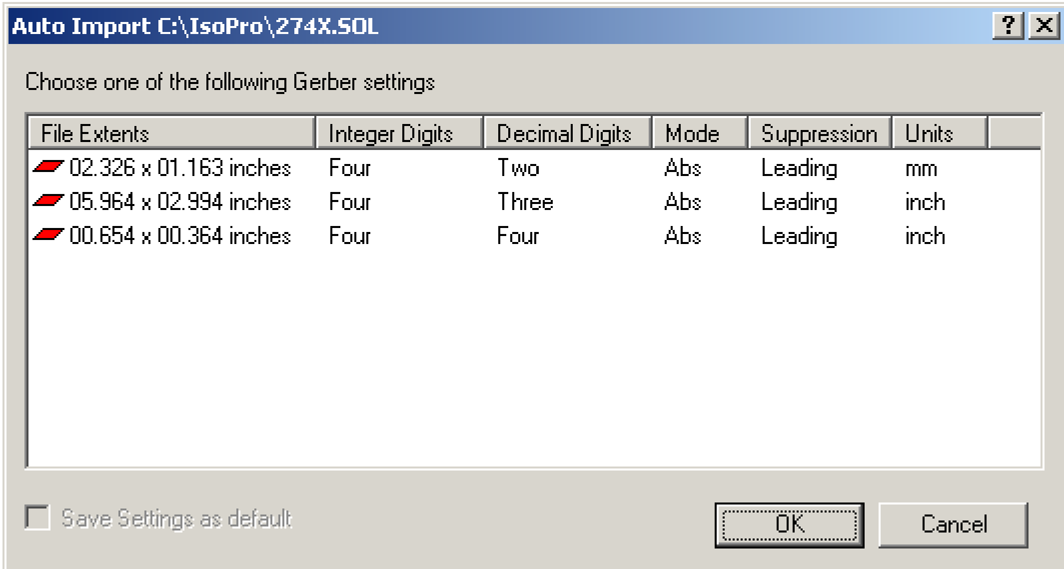


Step 2: Select the folder that contains your files (in this case C:\IsoPro) Select the following files:

274X.GBR	component file
274X.SOL	solder file
274X.NCD	drill file

To import all three files at the same time, hold down the CTRL key and click on each file.

Step 3: Click Open. The Auto Import window lists various parameters; including board dimensions (file extents), whole digits, precision digits and zero suppression methods.



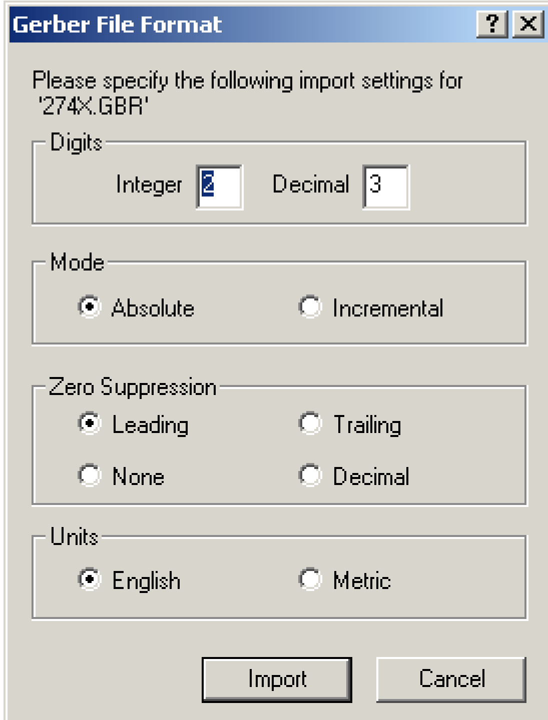
change screen capture to show the 6 x 3 board that we will be using

You can identify the correct parameters by noting the file extents, which are approximately the same as the board size.

Step 4: Double-click on the file extents that represent the approximate size of your board. For our purposes, select a board size of 5.9 x 2.99 inches using 4.3 absolute mode and leading zero suppression.

When using your own CAD files, rather than the samples provided with this tutorial, select the file extents closest to the size of your circuit board data. Please note that file extents are approximate only. The Auto-detect File(s) feature does a quick estimate rather than an exact calculation of the board size.

You may also bring in your file by using File > Open on the toolbar and specifying the import settings.



IsoPro also has an optional DXF import feature. If your IsoPro software is licensed for DXF import, click on File > Import > DXF File(s)... to import DXF data.

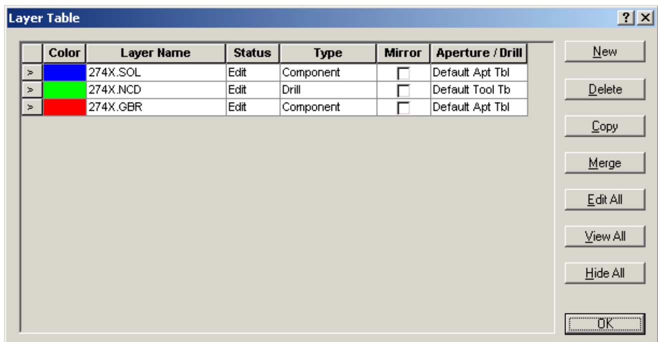
A single DXF file often contains all the layers of a circuit board. IsoPro automatically imports each embedded layer separately. Use the Layer List (explained hereafter) to identify your layers as 'Component', 'Solder', or 'Drill' as appropriate.

Layers containing unnecessary information, such as unwanted text, should simply be set to 'Hide'.

Working with Layers

IsoPro imports each file into a separate layer represented by a different color. Layers enable you to separate the component, solder, and drill information so that you can edit one without affecting the others.

The Layer List icon looks like a stack of four sheets of colored paper. When clicked, it brings up the Layer Table. From here, you can change the layer color, name, status, and type. You can also specify whether the layer is mirrored and define the aperture table to use.

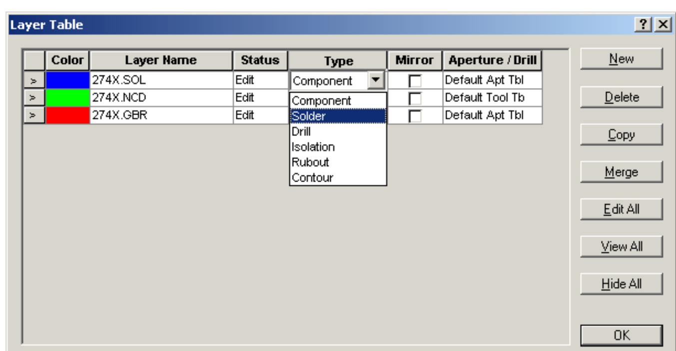


Layer Palette in the left margin of the screen displays the same colors as the ones shown in the Layer Table. This is an easy way to identify data on your screen. Changing a layer color in the Layer Table also changes it in the Layer Palette.

Set the following layer types for the tutorial files imported earlier:

set 274X.gbr to Component

set 274X.sol to Solder



Notice that 274X.ncd is already identified as Drill and that the layer type defined as Solder is automatically marked as 'Mirror'.

Mirroring here refers only to the files that are output for use with the Quick Circuit machine. On the screen in IsoPro, all layers are displayed as viewed from the component side of the board. When a layer is mirrored, it means that all future work on the particular layer will also be mirrored automatically.

There are three possible status modes. They are:

View - Allows you to see the layer, but not edit the data. This is helpful when using a layer as a logical reference.

Edit - Allows you to modify, select, delete, mirror, and edit t he data on the layer.

Hide - Allows you to hide the layer to prevent confusion while working on other layers.

While in the Layer Table, you can also add a new layer, copy an existing layer, or delete unnecessary layers.

Layer Registration

Purpose: To align the layers so that they line up (register) with one another.

For this tutorial, we purposefully created a component and solder side that did not line up with each other.

Inspect your CAD files to determine if the solder side is mirrored. If your solder side is not mirrored, the holes on each layer will NOT line up.

In the following procedure, you will first hide your drill layer since you will not be working with it for now. Then you will set the status for the component layer to View and verify that the status of the solder side layer is set to Edit.

Use the following steps to register each layer:

Step 1: Set the 274.ncd layer to Hide. It is already registered so there is no need to display it at this time.

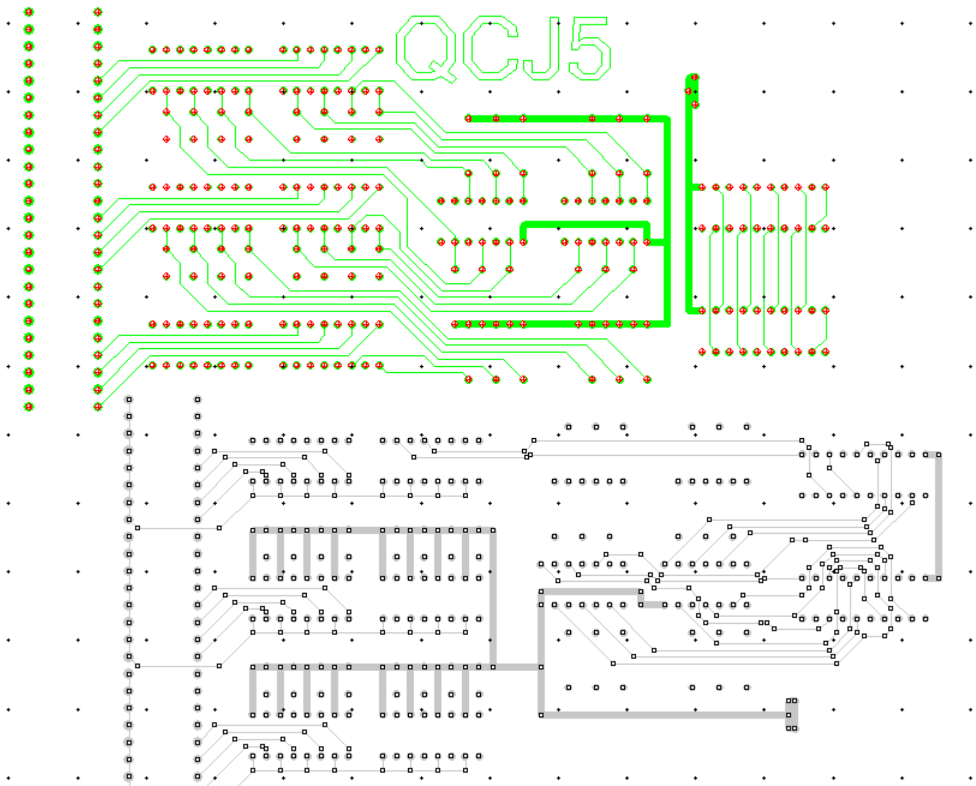
Step 2: We will be moving the solder data to register with the component data so set the component layer 274x.gbr to View so we do not change it.

Step 3: Set the 274x.sol layer to Edit and make sure the Mirror check box is selected. The only data that can now be edited is the solder layer.

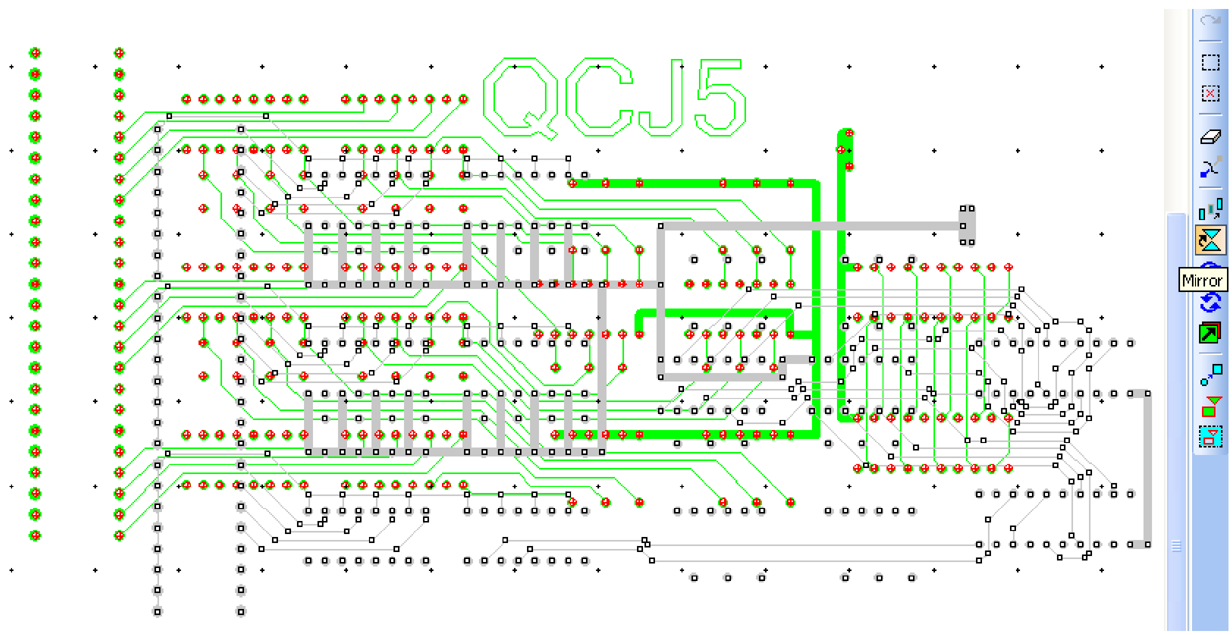
Note: You can also change the status of the layers (i.e. Edit, View, Hide) by left clicking on that layer in the Layer Palette.

Step 4: Close the Layer Table.

Step 5: Use your mouse to click and drag a box around the entire solder file (you can also use Edit > Select All or press Ctrl+A).



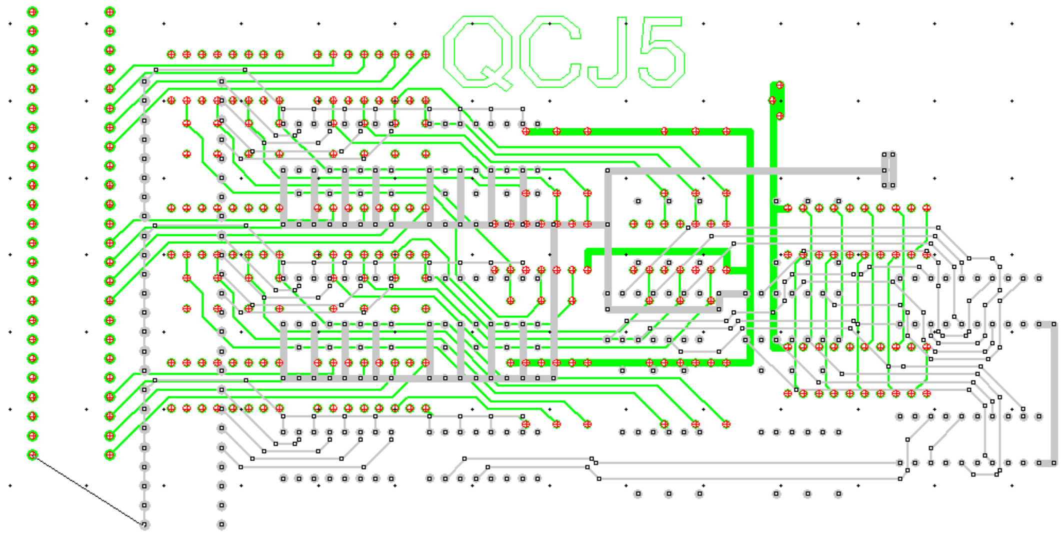
Step 6: The next step is to mirror the board. Click on the Mirror icon.



Step 7: Click on the Layer Registration icon to select the Register tool.

Step 8: Visually identify a pad on the solder layer and the component layer that should be registered with each other. Move the cursor over the pad on the solder layer and click the left mouse button.

While holding the button down, drag the cursor over the matching component pad. You will notice that the line snaps to the center of the pads. Once the line snaps to the correct pad on the component layer, release the mouse button. The data is then offset to the new position.



If you make a mistake, you can use the Undo function to undo the previous action and Redo to redo a previous action. IsoPro has unlimited Undo and Redo functionality.

Step 9: Once you have the layers registered, you can use the Layer List or Layer Palette to set the status of each layer to Edit.

In this tutorial the drill layer was already registered with the component side. However, if the drill file was not registered you would repeat this procedure for the drill layer.

Saving Your Work

At this point you should save your work. IsoPro files are saved as *.iso files. Select File > Save As in the menu bar and name the file "Tutorial Step1.iso". This will allow you to come back to this step in the tutorial if you wish to practice.

A *.iso file includes all work done in IsoPro, including layers, aperture lists and tool tables. Work saved in a *.iso file may be restored at any time by selecting File > Open in the menu bar.

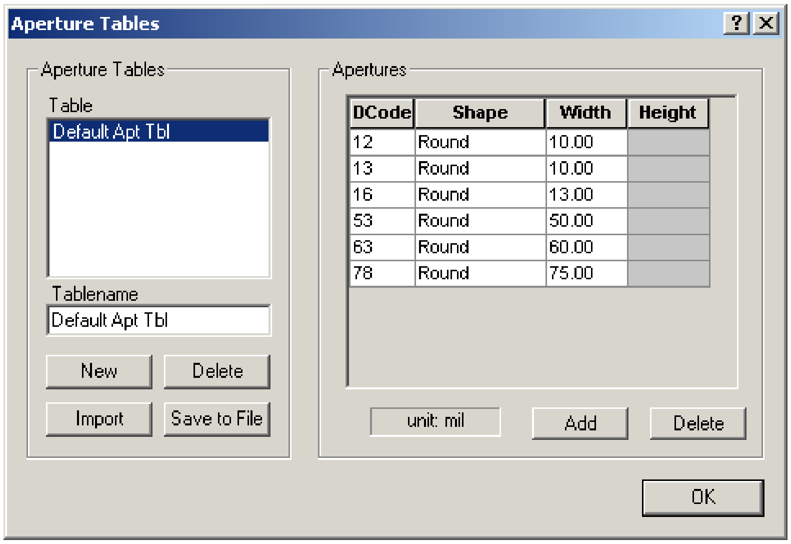
Verifying the Aperture List

Purpose: To verify the aperture shapes and sizes used to draw your circuit board's pads and traces.

To view the aperture list, click the Aperture List icon on your tool bar. You should verify that your aperture list is correct for your circuit board.

In this tutorial, the imported files use the RS274-x standard. This means that all the apertures were imported directly without intervention from the user. T-TECH STRONGLY RECOMMENDS USING RS274-X FORMAT. Most CAD packages released since 1995 support this format.

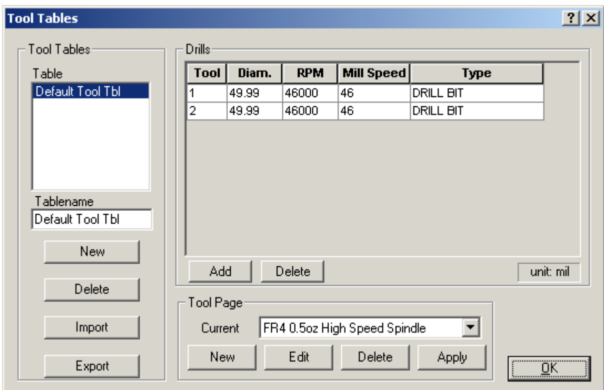
If your files do not use RS274-X, and you do not have an aperture list loaded, you will notice that your aperture dimensions default to 9.99 and 49.99. This is a very distinctive size and will probably never be used on a circuit board. If you see these values in your list, manually edit the aperture widths to match the output of your CAD package. Most CAD packages, when not using RS274-X, output the aperture list as a separate report file.

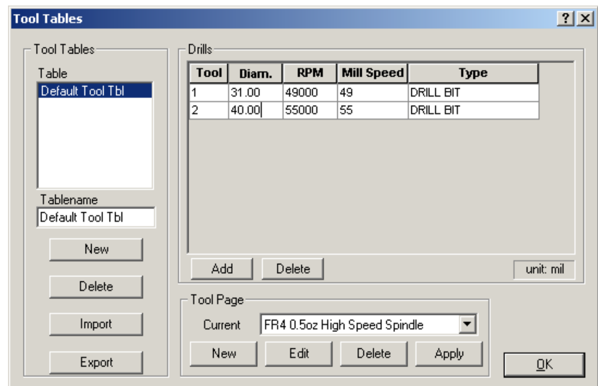


Editing the Tool Table

Purpose: To verify the tool sizes used to make your circuit board.

Click on the Tool Table icon on the tool bar. The tool table usually imports into IsoPro automatically, but on occasion it may not. In that case you must edit the data manually.





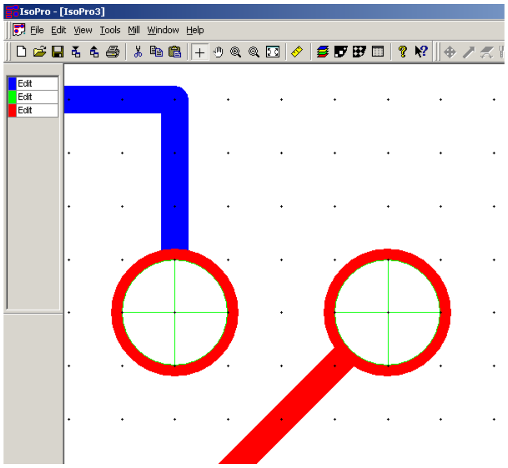
change these screen captures to show current tool table information

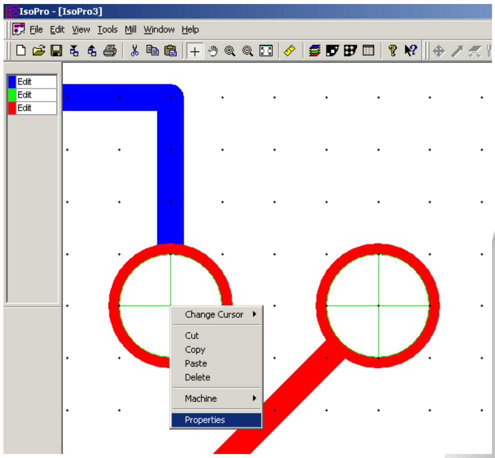
If your tool sizes default to 49.99 mils or 1.27 mm, this is a flag that indicates these are the default values. For this tutorial, simply set Tool 1 to 0.78 mm (31.00 mils) and Tool 2 to 1 mm (40.00 mils). Notice how IsoPro automatically selects the appropriate RPM and Mill Speed for the selected tool.

However, for your own applications, you should always match the sizes of the holes in your board to those that your CAD package outputs. These sizes can be determined by either looking at a report file that was output by your CAD package or looking at the header of the drill file itself. If they are in the header of the drill file, you can view it by opening the file using a simple text editor.

Changing a Board Entity

At times, it may be necessary to manually change the size of an entity in IsoPro. Notice how the default entity size of 49.99 mils in this tutorial makes the drill hole oversized. To change the entity, follow the steps below.



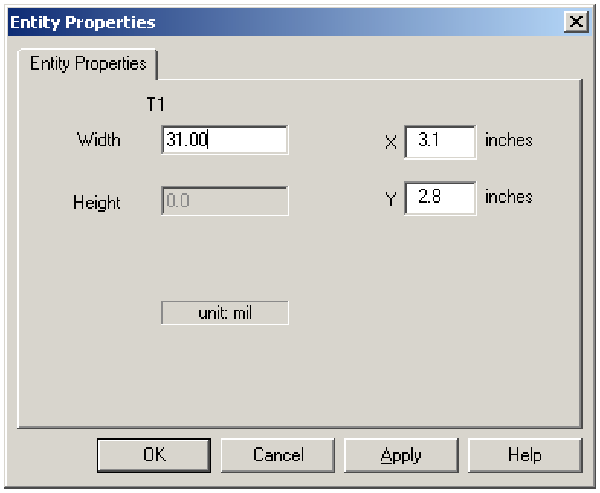


Step 1: Place the mouse over the entity to be changed and right click. This will reveal a drop down box. Select Properties. If using the same file, make the drill layer visible.

Step 2: The Entity Properties dialog box appears.

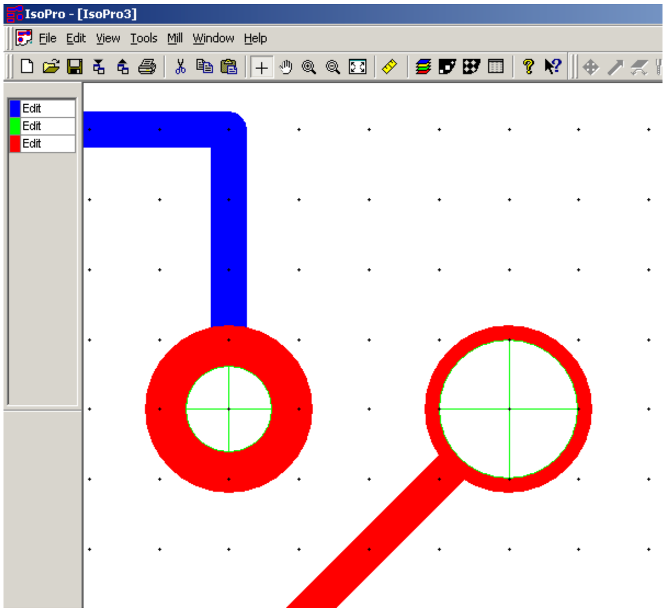
'D' codes refer to the entities such as pads and traces, 'T' codes refer to the tool sizes.

In this example we are going to change the 'T' code (tool size). When a drill hole is centered on a pad, the entity [pad] will be shown first, and the drill size follows. Select OK until you reach the tool size window you want.



In some cases there will be two entities buried on top of each other. In this case, IsoPro will iterate through both entities. Changing the properties of an entity in this manner will not be necessary if the files from your CAD package are correct. (However, if you want to do some last minute editing without going back to your CAD package you can change the size or shape of a single pad or trace using this method. No other pads or traces will be affected.)

Step 3: Change the diameter of this hole from 49.99 mils to 0.78 mm (31.00 mils) as shown above, and then select OK. The resulting change in the drill hole is shown below.



Step 4: Repeat these steps for each entity you wish to change.

Always remember to change your CAD file accordingly. Otherwise, your prototype board will be correct, but your production boards may not.

Isolating the Layers

Purpose: To create an outline around the pads and traces of your design so that a circuit board prototyping system can produce your board.

Now that you have verified that your aperture list and files are correct, you need to isolate the component and solder layers.

Step 1: Set the status of the drill and solder layers to Hide.

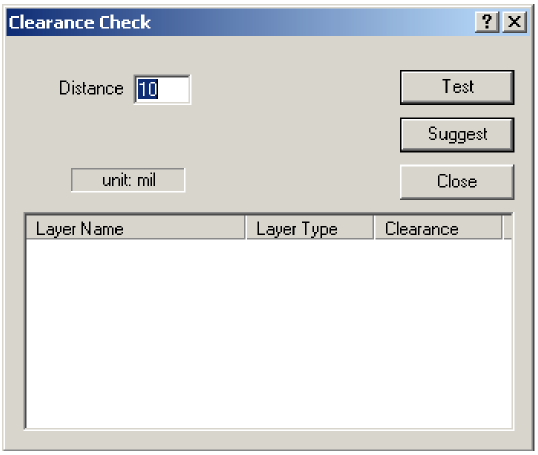
This is not a required step. However, it can help prevent mistakes and makes your first few board designs run a little smoother.

Another way is to set the solder and drill layers to View. The important thing is to make sure that only the layer you wish to have in action is in Edit mode.

Step 2: Determine the minimum clearances for pad-to-pad, pad-to-trace, and trace-to-trace distances.

The initial isolation cannot exceed the minimum clearance. You can determine the minimum clearances in your circuit in IsoPro manually using the measure function or by using the Clearance Check function. Alternatively, you may already know this from your CAD software.

a) To determine the minimum clearance, select Tools > Clearance Check in the menu bar.



b) At the Clearance Check screen, click on Test. IsoPro will automatically determine the minimum clearance on that layer. Areas on the circuit that fail the clearance test will be highlighted. Make sure that the highlighted areas are de-selected before continuing.

IsoPro calculated that the minimum clearance for the component layer at 0.34 mm (13.5 mils).

For the purposes of this tutorial, we will do the initial isolation with a 0.25 mm (10 mil) tool, followed by a 0.78 mm (31 mil) tool. It is necessary to perform an isolation with a tool size equal to or less than the minimum clearance so that all of the nets in your circuit are properly isolated.

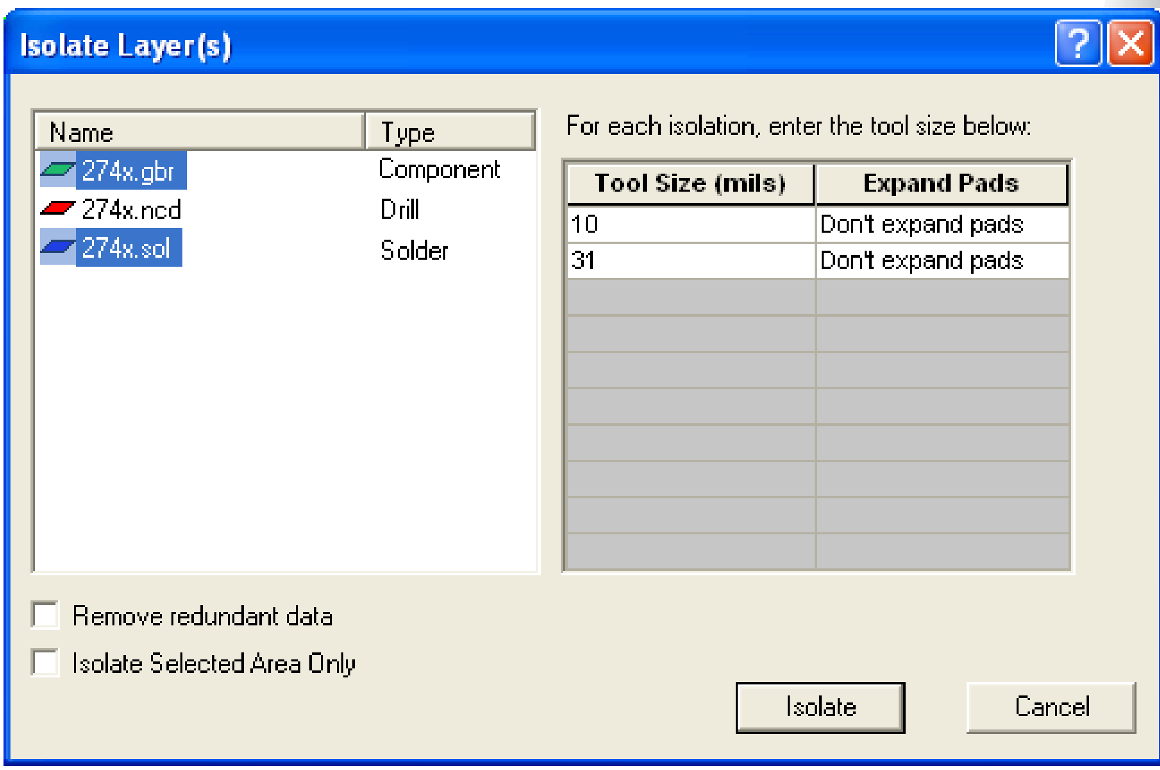
Choosing a diameter that is too large will result in the merging of nets on your board. To solder more easily, a larger second pass, in this case 0.78 mm (31 mils or 0.031"), is recommended.

This is typically all that is required for a digital board but please note that IsoPro can perform up to three different size isolations simultaneously.

Step 3: Click on Tools > Isolate in the menu bar.

Enter 0.25 mm (0.010") for Pass 1 and 0.78 mm (0.031") for Pass 2. Set Pass 3 to zero which indicates that this pass is not used.

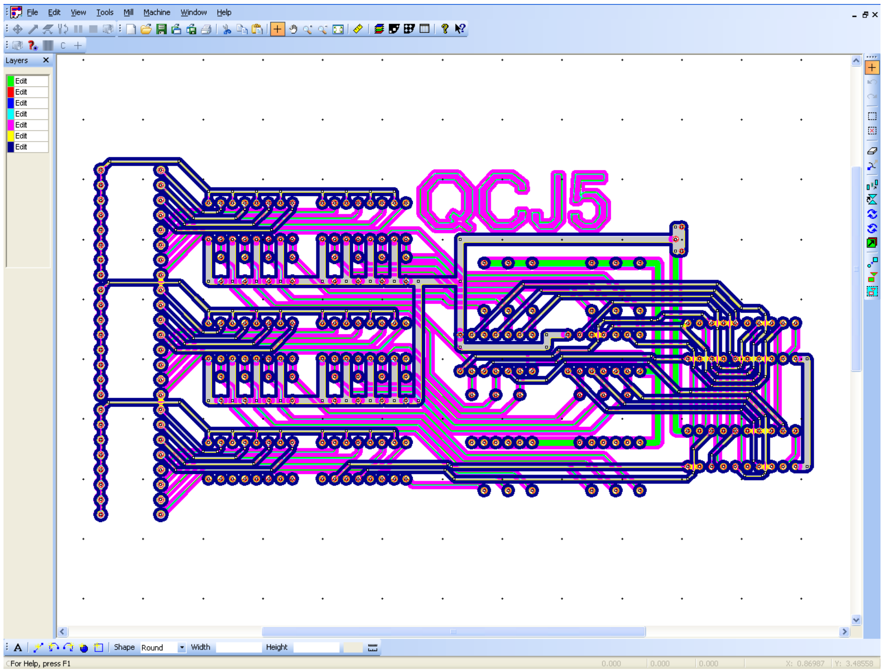
Note: Tool sizes can be entered in inches or metric mode [mm]. Metric mode can be selected from Edit > Preferences in the menu bar.



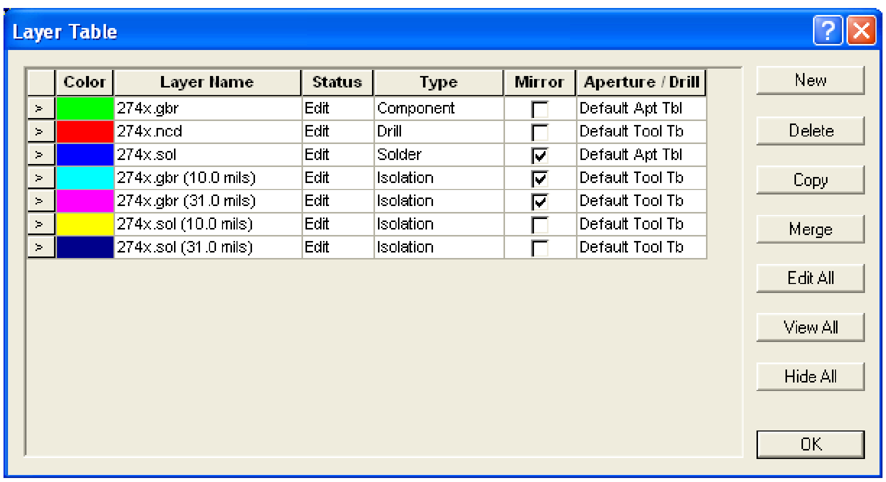
Step 4: Select the component and solder layer by holding down the CRTL key while selecting both layers.

Step 5: Click on the Isolate button.

You will see different color representations (or shades of gray if you did not print this manual in color) of the defined mill paths for both the component and solder layers once the isolation process is completed.



Notice that four additional layers are shown on the Layer Palette. Click on the Layer icon to see the definitions for these new layers.



	Color	Layer Name	Status	Type	Mirror	Aperture / Drill
>		274x.gbr	Edit	Component	☐	Default Apt Tbl
>		274x.ncd	Edit	Drill	☐	Default Tool Tb
>		274x.sol	Edit	Solder	☑	Default Apt Tbl
>		274x.gbr (10.0 mils)	Edit	Isolation	☑	Default Tool Tb
>		274x.gbr (31.0 mils)	Edit	Isolation	☑	Default Tool Tb
>		274x.sol (10.0 mils)	Edit	Isolation	☐	Default Tool Tb
>		274x.sol (31.0 mils)	Edit	Isolation	☐	Default Tool Tb

When IsoPro creates the isolation layers, it automatically lists them by tool size. In addition, it defines all isolation layers for the solder layer as mirrored. (This occurs only if you selected Solder as the layer type.)

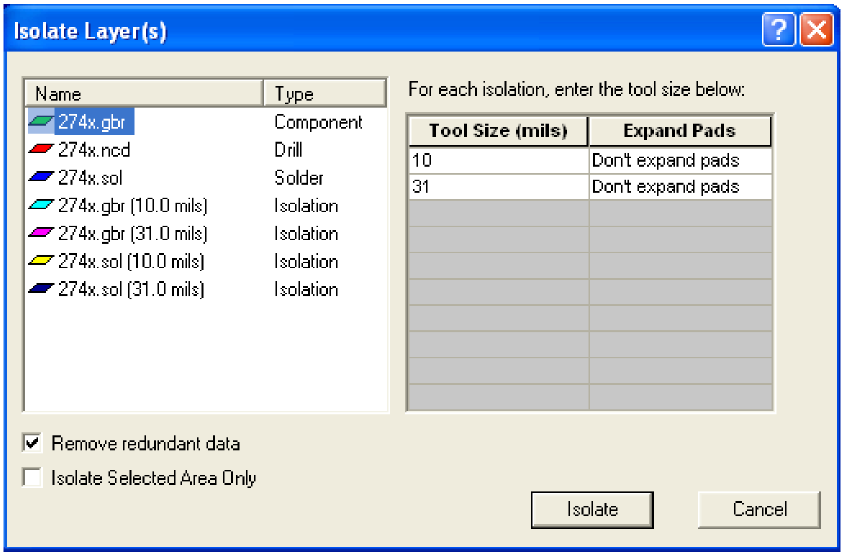
IsoPro also provides several special options for use during the isolation routine. These are Remove Redundant, Force Isolation and Expand Pads.

Remove Redundant Function

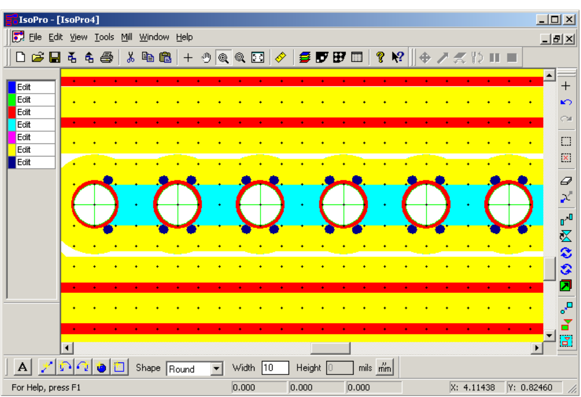
Remove Redundant locates and deletes those sections of the smaller tool isolations, which are completely overlapped by larger isolation paths. On some types of circuit boards, the use of Remove

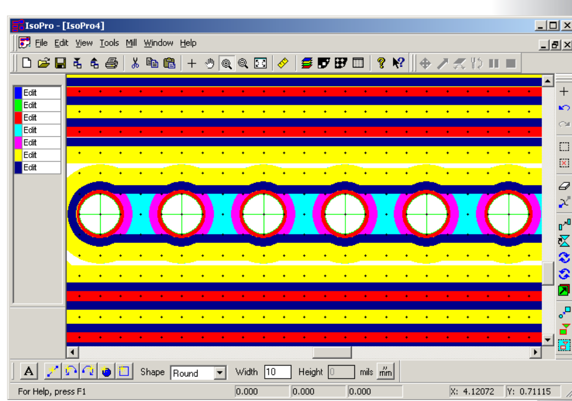
Remove Redundant Function

Remove Redundant locates and deletes those sections of the smaller tool islations, which are completely overlapped by larger isolation paths. On some types of cicuit boards, the use of Remove Redundant can offer significant savings in machine time and tool wear. The Remove Redundant option is enabled and disabled in the the Isolation dialog box.



In the illustration above, we are isolating only the component Layer by setting the solder layer to Hide and the drill layer to View. We are making two isolations and have enabled the Remove Redundant feature by checking the applicable box in the Isolation dialog.

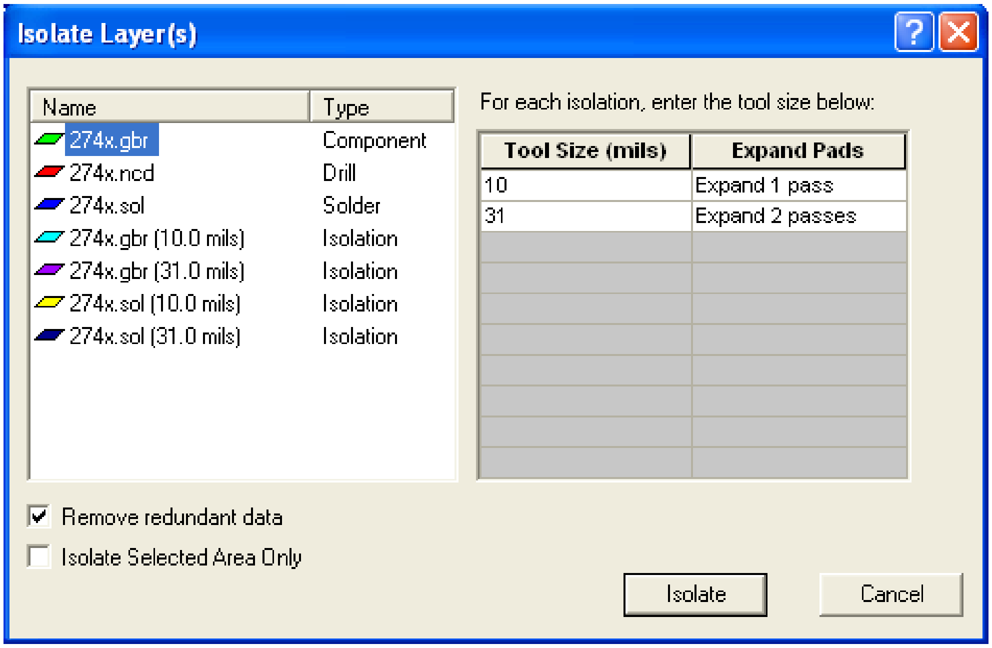




Enabling the Remove Redundant option prevents isolations from occurring (as shown in the figure on the left). The red (or darker gray) represents the 0.25 mm (10 mil) isolation path. It only appears in those areas where the subsequent isolation, in this case the 0.78 mm (31 mil) represented by the light green (or lighter shade of gray), cannot effectively remove the required copper. Without the Remove Redundant feature, your isolation paths appear as shown in the figure on the right.

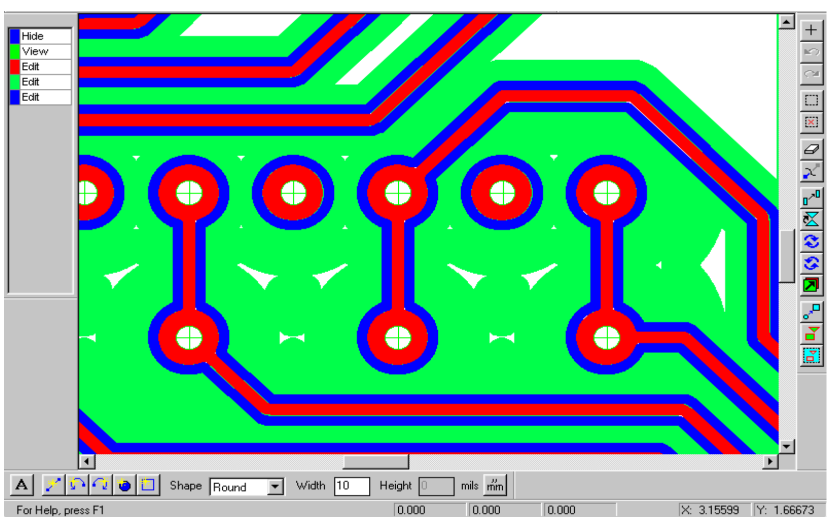
Expand Pads Function

Expand Pads is used to increase the number of isolation passes around a pad without increasing the number of passes around associated traces. This feature is useful in widening the area around pads to provide additional space for soldering.



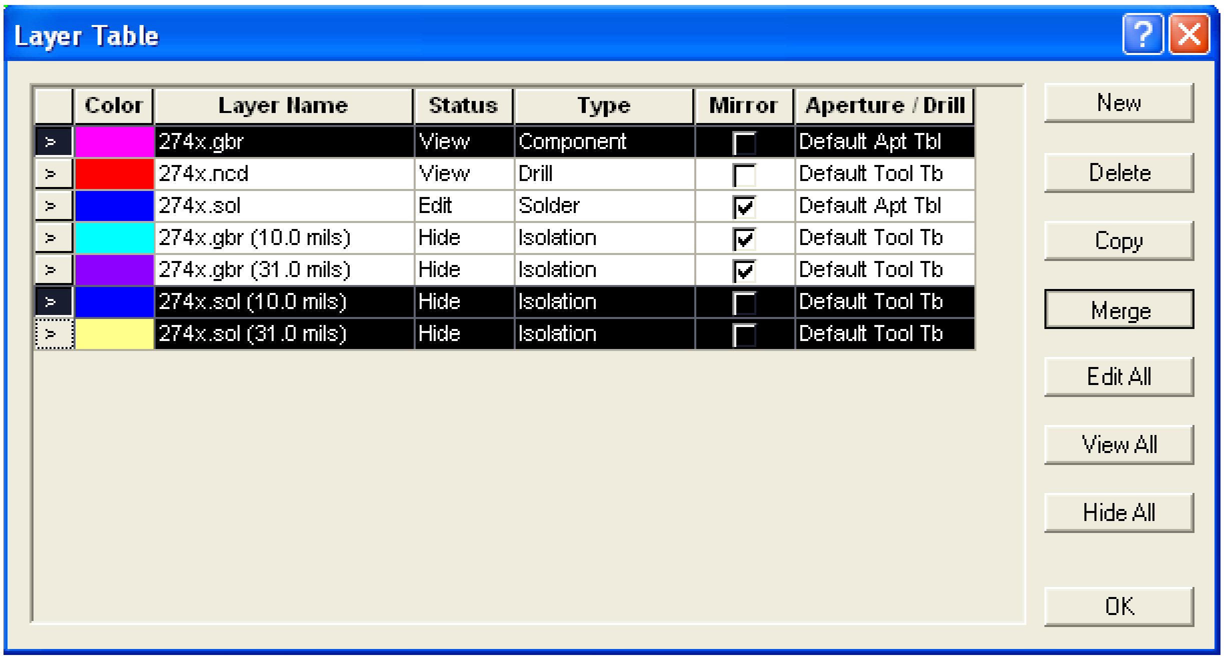
As shown above, isolate only the component layer by setting the solder layer to Hide and the drill layer to View. As before, we want to make two isolations and enable the Expand Pads feature on the second pass since it is the larger one. When Expand Pads is selected, you must also specify the number of additional passes you wish to make with that tool.

Looking at the illustration below, we can see that the second isolation paths around the pads are twice as wide as those around the traces.



Merging Layers

Purpose: To combine all like layers for seamless milling also allowing for the Auto Tool Change function to be used. Without merging layers the machine will stop milling after each layer and ask for a new tool.



Inspecting the Isolations

It is a good idea to zoom in and inspect the isolation to make sure it was done properly. You should see a clean outline around each electrical net.

Use the zoom-in cursor on the tool bar to drag a window around an area of interest.

Use the pan-hand feature to move around and inspect the data.

If the isolation path is not completely defined around each entity on your board, or if the isolation path is too large as a result of using the Force Isolation feature, the board will not be electrically correct. Use a smaller tool size and repeat the Layer Isolation procedure as needed.

Rubbing Out the Base Copper

The Rubout feature allows you to remove an area of unwanted copper in a single function. Ordinarily, you would not perform a rubout on a digital board; however, if you have a set of fingers (for an edge connector) or an SMT component that requires a rubout, you can take off copper from a particular part of the board, several specific areas of the board or the entire board. The Rubout feature can be useful for many RF/MW applications.

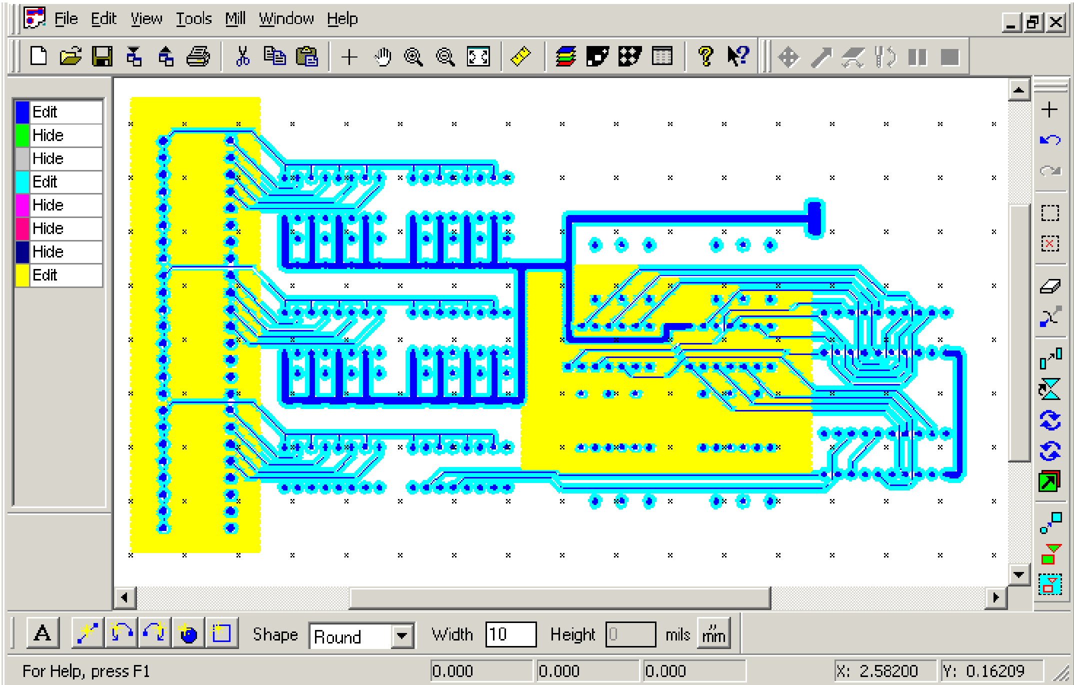
You can also modify the rubout area by using the "Ball & Stick" mode under View in the menu bar, then selecting unwanted portions of the rubbed out area and deleting them. The "Ball and Stick" view mode allows you to see the exact path of the rubout tool.

Rubout only works one layer at a time. If you want to perform a rubout on both the component and solder layers, you must repeat this procedure for each side.

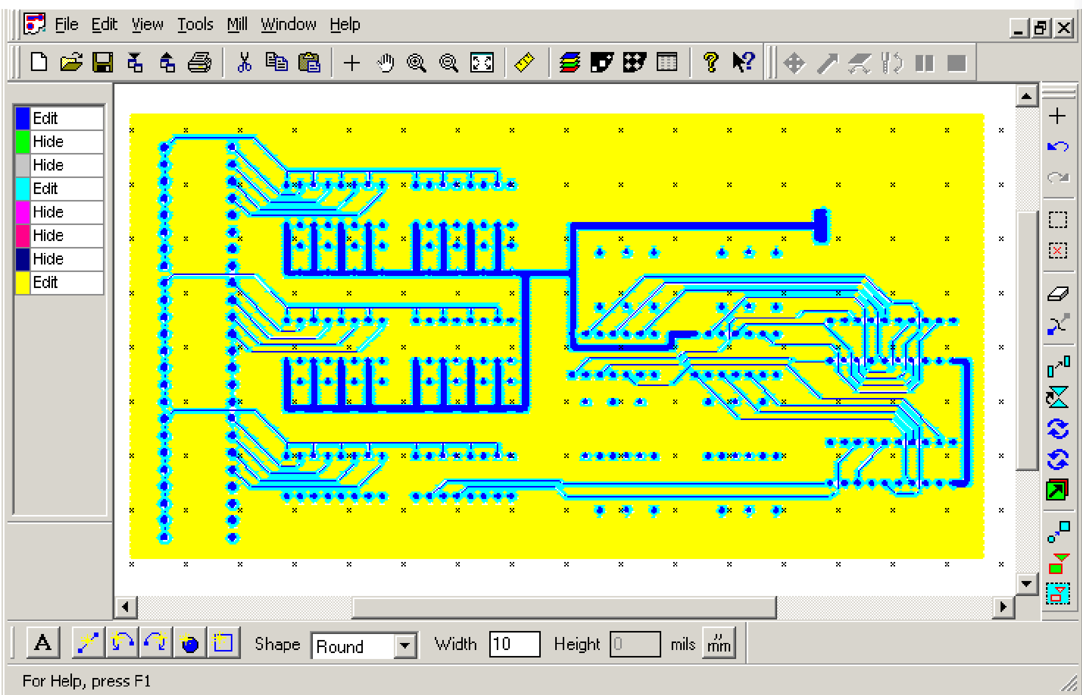
Step 1: Set the isolation layers where you want to create a rub out to Edit .The Rubout function will automatically select the largest size isolation layer available in Edit mode and base the Rubout pattern on this tool size.

Step 2: Click the Rubout icon at the bottom of the right vertical tool bar.

Step 3: Drag a box around the area(s), where you want to rub out the base copper. Release the mouse to activate the Rubout.



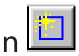
Yellow (or light gray) areas show the effect of a partial Rubout as shown above, or of a full Rubout as shown below.



Creating the Board Outline

Purpose: To create an outline of your board.

You can import the board outline from your CAD package or create it in IsoPro. Using IsoPro, there are two methods for creating the board outline:

- Click on the Create New Rectangle icon on the lower left of the screen , and drag and draw a rectangle for the board outline
- or lay out individual lines around your circuit if a simple rectangle is not appropriate

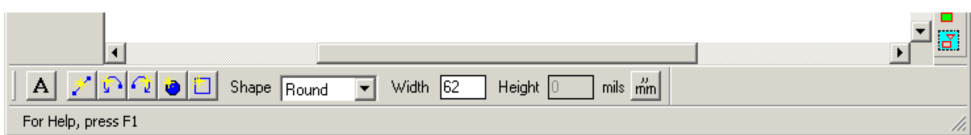
In either case, start by creating a new layer to put information about the board outline.

Step 1: Click on the Layer icon and select New.

Step 2: Set the component layer to View and all the other layers to Hide except for the new layer which should be in Edit mode.

Step 3: Name the new layer Board Outline, then close the dialog window.

Step 4: The most common routing tool is the 1.575 mm (62 mil) diameter router. At the bottom of the screen, define the tool shape as Round and its Width as 62 mils (1.575 mm if you are in metric mode).



If you are using a different tool size to create your board outline, enter the appropriate information in the Shape and Width boxes.

Step 5: For the purpose of this tutorial, we want to use a simple rectangle as the outline for our board.

Click on the box shaped icon and bring your cursor to the upper left area of your circuit. Click and drag a box around your circuit. When done, release the mouse.

You can resize the outline as needed, until you are satisfied with the result.

Alternately, if you have your board outline described on a Gerber layer:

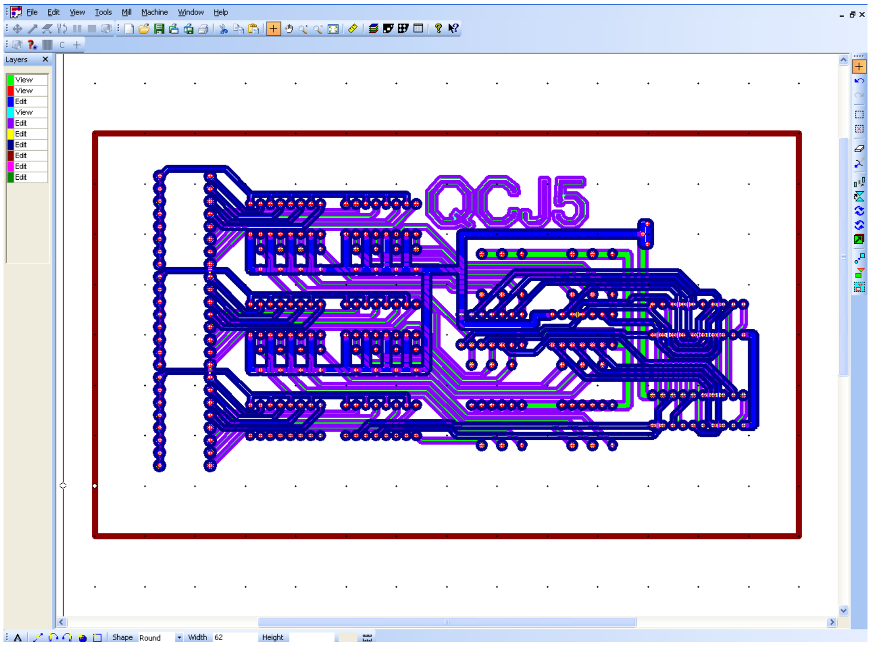
Step 6: Set the layer that contains your outline data to Edit and other layers to View or Hide.

Step 7: Delete all traces, pads and text that are not part of the outline.

Step 8: Select all of the traces that form the outline.

Step 9: From the Edit menu, select Convert to Polygon.

Step 10: Now, isolate this layer with a .062 mil tool (or the appropriate diameter for your board)



We recommend that you create the board outline before you perform the Rubout function. This way the board outline will serve as a boundary for creating a full rub out if needed.

Creating Text

Purpose: To create text on a board.

Creating text in IsoPro is very easy. IsoPro offers a wide variety of fonts and sizes. The first step is to create a new layer for the text to reside on. Follow the steps below:

Step 1: Click on the Layer icon and select New.

Step 2: Set all the layers to View or Hide except for the new layer which should be in Edit mode.

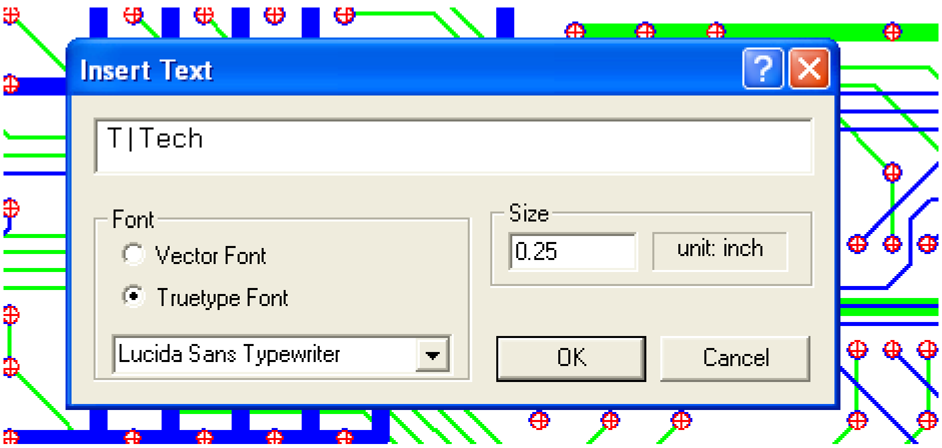
Step 3: Name the new layer Text.

Step 4: At the bottom of the screen, click on the icon containing the letter A.

Step 3: Name the new layer Text

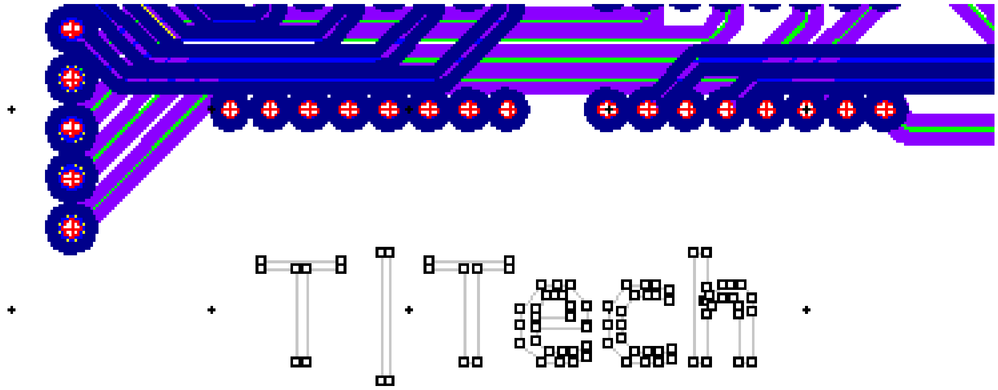
Step 4: At the bottom of the screen, click on the icon containing the letter A.

Step 5: Place your cursor where you want to locate your text. A dialog box will appear



Step 6: Type in "T-Tech" and select the text font and size as needed. When finished, select OK. The text image appears on the board.

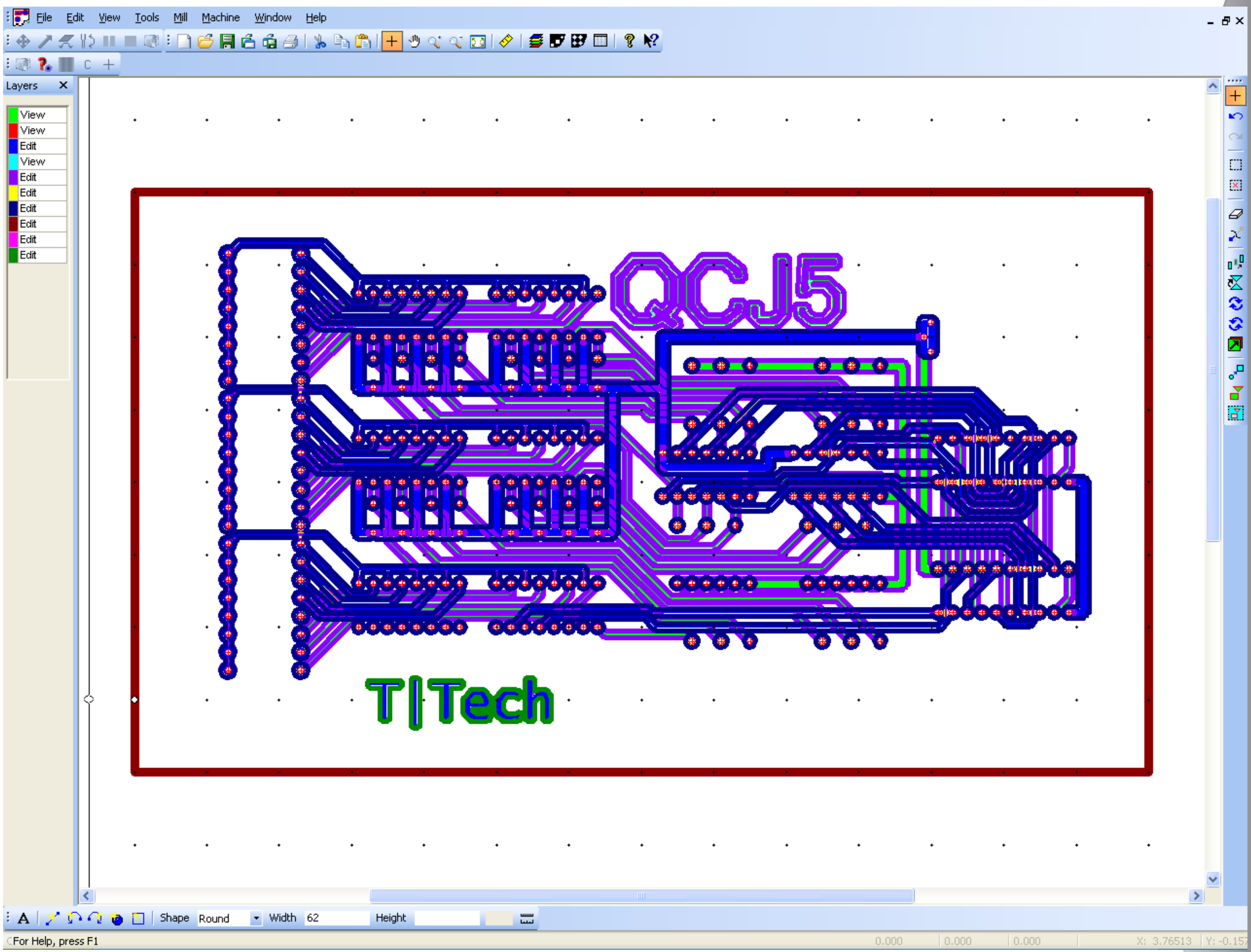
Vector fonts are recommended for milling. These are fonts that are made of only center line data as opposed to TrueType fonts which are made of outlines.



True Type (TTF) fonts should be converted to a polygon to avoid milling an outline instead of a full letter.

You are now done with your design that will be used by the Quick Circuit prototyping machine. Save this file as “Tutorial complete.iso”. If you design on a computer that does not have a Quick Circuit attached, save your .iso file to a disk or copy the file to the computer where your Quick Circuit is located.

Keep a copy of this file in a safe place. If you ever notice Quick Circuit created poor quality boards, having a known “good” file will help you discover if the problem is hardware or software related. See Diagnostics on page 67.



"Mill" Drop Down Menu

The Mill drop down menu is probably the most used function in the menu bar. A brief description of each item associated with this menu is given below.

Initialize / Re-Initialize – Prepares IsoPro to run Quick Circuit and zeroes the machine.

Setup > Adjust Pin Position – Adjusts the Y-position of the pin to improve hole-to-pad registration on double-sided boards. This is a fine adjustment used with the Set Pin Position feature.

Setup > Material – Sets chip load and surface speed.

Setup > Set Pin Position – Sets pin position with respect to the current head position. Use the Adjust Pin Position feature to fine-tune the head position.

Run Layer – Allows user to select a layer to be machined.

Mill Selected – Machines the selected entities on a layer.

Pause – Temporarily stops operations to give the user control of the machine.

Stop – Stops operations on the milling table.

Tool Change – Brings the head assembly to the front of machine to allow for tool change.

Material Change – Sends the head assembly to the back of machine to allow for material change. The Material Change position is determined by the size of the board.

Home – Returns the head assembly to Home position. (Home is specified under Preferences > Machine Settings in the menu bar.)

Head Down/Up – Raises or lowers the head assembly.

Jog – Places the machine under user control.

Move – Allows an absolute or relative move of the head assembly.

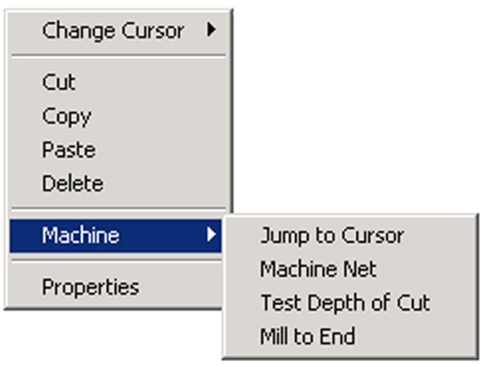
Spindle – Turns the spindle on or off. A checkmark next to this item indicates that the spindle is on.

Park – Returns the head assembly to the Park position. (The Park position is set under Preferences > Machine Settings in the menu bar. The head assembly travels to the designated Park position after completing a drill/mill routine.)

Zero Machine – Zeroes the milling table.

Right-Click Menu

Right clicking in the main window and selecting the "Machine" cursor menu gives you the following display.



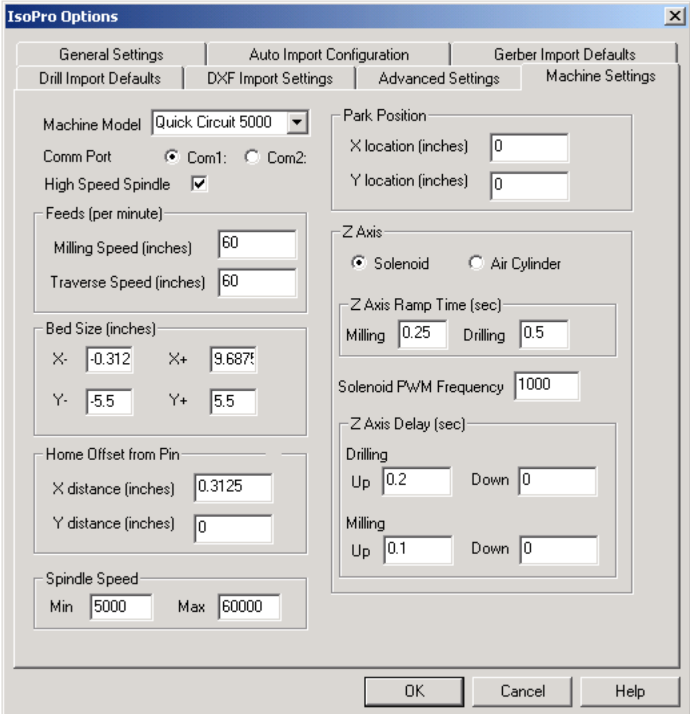
Jump to Cursor – Moves the head assembly to the current X,Y position of the cursor.

Machine Net – After right clicking on an entity, the machine mills all entities contained in connected the net.

Test Depth of Cut – Allows you to test the depth of cut of a pointed milling tool. This feature is not available if the machine has not been initialized.

Mill to End – If you need to stop machining on a board before the entire layer is finished, this feature allows to specify a new starting point and select Mill to End to finish machining the layer entities.

MANUAL CONFIGURATION OF ISOPRO MACHINE DRIVE



When you install IsoPro using the CD/disks included with your machine, your machine configuration is set automatically. However, if you need to set or change some of these parameters, you can do so by selecting Edit > Preferences > Machine Settings tabs.

Machine Model – Allows you to select your machine type. Select Quick Circuit/HF.

Comm Port – Allows you to specify which communications port the controller is connected to.

High Speed Spindle – Allows you to indicate whether or not you are using a high-speed spindle. This should be checked for the Quick Circuit/HF.

Feeds (inches per minute)

> Milling Speed – Indicates the tool speed with the head assembly down during a manual move.
>
> Traverse Speed – Indicates the speed at which a tool moves with the head assembly up.

Bed Size (inches)

X- – Indicates the usable bed space in front of the pinning hole. This area is used to prepare mill paths before machining.

X+ – Indicates the usable bed space behind the pinning hole.

Y- – Indicates the usable bed space to the right of the pinning hole.

Y+ – Indicates the usable bed space to the left of the pinning hole.

Home Offset from Pin (inches)
The Home position is usually set at X = 0, Y = 0.

X distance – Determines the location of the Home position on the X-axis as an offset value from pinning position.

Y distance – Determines the location of the Home position on the Y-axis as an offset value from pinning position.

Spindle Speed (RPM)

Min – Indicates the minimum actual speed to be associated with the minimum software speed. This value should be different depending on type of spindle.

Max – Indicates the maximum actual speed to be associated with the maximum software speed. This value should be different depending on type of spindle.

Park Position (inches)
The Park position should outside the edges of the board. After each layer is machined, the head assembly moves to the Park position.

X location – Determines the location of the Park position on the X-axis as offset from the Home position.

Y location – Determines the location of the Park position on the Y-axis as an offset from the Home position

Z-Axis

The Z-Axis is controled by a stepper motor and adjusts Z+- steps as called for by the controller.

Z-Axis Ramp Time (sec)

Milling – Indicates the time it takes to ramp the solenoid power from zero to full power while milling. This feature is not used on the Quick Circuit/HF model.

Drilling – Indicates the time it takes to ramp the solenoid power from zero to full power while drilling. This feature is not used on the Quick Circuit/HF model.

Solenoid PWM Frequency – Base frequency that is used by the pulse width modulation of the solenoid. There should not be any reason for you to change this value. This feature is not used on the Quick Circuit/HF model.

Z-Axis Delay (sec)

Separate Delay for Drilling and Milling operations.

Up – Indicates the amount of time to wait after a head up command before continuing with next command. This feature is not used on the Quick Circuit/HF model.

Down – Indicates the amount of time to wait after a head down command before continuing with next command. This feature is not used on the Quick Circuit/HF model.

Z-Axis Ramp Time, Solenoid PWM Frequency and Z-Axis Delay should not be changed from their defaults unless you are experiencing problems with tools not penetrating your board material or dwelling too long in the board material.

Additional Features

There are many additional features built into the IsoPro program. You should take the time to become thoroughly familiar with all of the tasks, icons and commands. Please reference IsoPro's extensive on line Help menu for additional information concerning IsoPro's features.

APPENDIX A

The following is reprinted with permission from an article in the November 1992 edition of Printed Circuit Design Magazine:

Gerber and CAM Essentials

The inside view

by Andy Wise

Today, virtually all PCB designs are reduced to a series of Gerber plot files as the first step in the PCB manufacturing process. These Gerber files constitute the database that drives the CAM process. Historically, designers have been unaware of the exact nature of Gerber files and were therefore unable to verify the correctness of a Gerber file or open a dialog with the CAM people about the problems that commonly occur. Furthermore, without having the proper tools, designers have had little or no control of their design once it has been committed to a Gerber database.

What is Gerber?

Gerber is the de facto standard photoplotting command language. It is supported by virtually all modern photoplotting equipment in use today. The command structure and format of a Gerber file (the name Gerber borrowed from the popular photoplotter maker Gerber Scientific Instruments Co.) is actually a subset of the EIA RS-274-D standard for numerically controlled machines. Each Gerber file contains commands and data that instruct the photoplotter on where to expose the film when generating PCB artwork.

Gerber Command Structure

While typical Gerber files can be megabytes in size, there are only a few essential commands in the Gerber language. The most prevalent are the “G” and “D” command codes, commonly referred to as G-codes and D-codes. The G-codes are called preparatory codes and are used to set the state of the plotter. The D-codes serve a dual role. They act as aperture select commands, or, along with XY coordinates, they specify whether the photoplotter “flashes” an aperture or “draws” a line.

The terms “flash” and “draw” relate to the older vector photoplotters that exposed the film through an aperture (Figure 1) by either momentarily energizing a light source while stationary (flash) or by maintaining there is not a one-to-one relationship between aperture position numbers and D-codes! In other words, you can't simply subtract nine from a D-code to find its aperture position. The reason behind the madness stems from the use of aperture "wheels" on the old vector photoplotters.

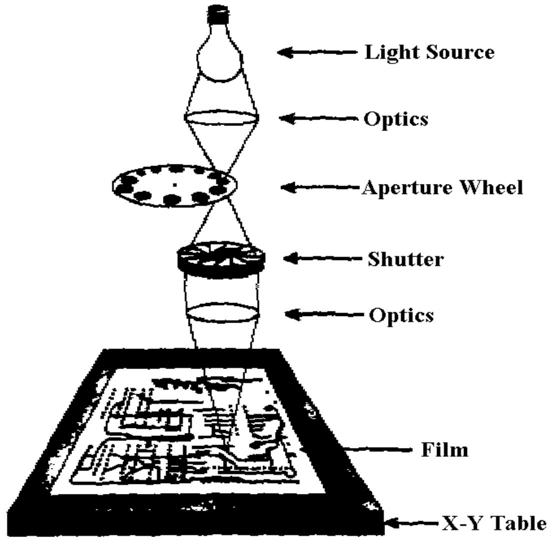


Figure 1 Photoplotter operation

These wheels typically contained 24 different apertures and for reasons unknown to this author, they were numbered as shown in Table 1. Fortunately, the right CAM software will automatically handle the ordering of D-codes within an aperture list. Furthermore, a major benefit of modern laser photoplotters is the absence of an aperture wheel. There is no 24-aperture limit nor a need for a list mapping aperture positions to D-codes! Although an aperture list is still required, it simply needs to describe the size and shape of each D-code used.

Table 1
Aperture Wheel/D-Code Mapping

Aperture	D-Code	Aperture	D-Code
1	D10	13	D20
2	D11	14	D21
3	D12	15	D22
4	D13	16	D23
5	D14	17	D24
6	D15	18	D25
7	D16	19	D26
8	D17	20	D27
9	D18	21	D28
10	D19	22	D29
11	D70	23	D72
12	D71	24	D73

An Incomplete Language

A major shortcoming of the Gerber command language has been the lack of a method to describe the size and shape of apertures within a Gerber file. Thus, a separate file in a nonspecific format is typically used to convey this information to the CAM operator. Because of this, much time and money has been wasted on bad film. Bad boards have been built if the errors are not detected in time, due to errors in converting aperture lists. Fortunately, the correct CAM software can recognize and automatically convert most aperture lists in use today.

A Variable Format Language

Although Gerber only has a handful of basic commands, the format of these commands can vary, adding confusion to the deciphering of Gerber data. Gerber files may be generated using imperial or metric units of measure. The precision of the data coordinates may be specified in different ways. For example, the value 01255 could be interpreted as 1.255 if two places are assumed before the decimal point (2.3 format) or 12.55 if three places are assumed (3.2 format).

Coordinate data may also be specified in either absolute or incremental formats. In absolute format, all coordinates are relative to the photoplotter origin or 0,0 point. On the other hand, incremental format specifies each coordinate value relative to the previous coordinate. Coordinate data may also be specified with either leading or trailing zeros suppressed. Thus 1255 could be interpreted as 1.255 or 12.550 assuming 2.3 precision. Again, the right CAM software should be able to automatically detect the format of a Gerber database (Figure 2).

```
G54D41*
G01X00200Y00600D03*
G54D20*
```

```
G01X00500Y00500D03*
G01X00500Y00600D03*
G01X00200Y00600D02*
G54D12*
G01X00350Y00600D01*
G01X00350Y00500D01*
G01X00500Y00500D01*
G01X00200Y00500D02*
G01X00275Y00500D01*
G01X00275Y00400D01*
G01X00500Y00400D01*
G01X00200Y00300D02*
G01X00500Y00300D01*
M02*
```

Figure 2 Sample Gerber file and plot.

What's Missing?

When a CAD system outputs the Gerber database, almost all of the electrical intelligence is lost. That is, all reference designators, pin numbers and, hence, netlist information is missing. The most serious consequence of this is the one-way nature of Gerber. There is no way to automatically back-annotate a design from changes made to the Gerber database. This isn't to say that you can't do anything useful with Gerber other than photoplotting. Quite the contrary!

Aperture Wheel Setup for Vector Plotters

The setup of an aperture wheel is an exacting and time consuming process since each aperture in the wheel must be hand-mounted and aligned. In order to avoid repeated setup costs, designers have the photoplotting vendor keep a wheel on file and are forced to always use that same set of apertures. This has obvious drawbacks, both in terms of design flexibility and the ease of migration to other vendors.

Raster (Laser) Plotters

Aperture Lists

Increasingly, vector photoplotters are being replaced by the laser photoplotter, which emulates the older style machine in a raster (bit-map) fashion. While use of the term "aperture" to describe a pad or trace shape persists, the term "aperture wheel" is now being replaced by "aperture list", which implies the greater flexibility now available to the designer.

There are three principle advantages with aperture lists on raster plotters:

Aperture shapes can be easily generated in software, thus eliminating the need to design a physical wheel.

More apertures can be defined on a list.

Allowable apertures sizes are typically (but not always) greater than those imposed by the physical dimensions of an aperture wheel.

Flash and Draw Apertures

No distinction need be made between Flash and Draw aperture types since the light source intensity is constant.

Speed Advantage of Laser Plotters

Laser plotters operate much quicker than vector machines. A complex plot that required hours on a vector machine can usually be performed in ten minutes or less on a laser photoplotter. This decreases turnaround time and in many markets has driven photoplotting costs down.

Talking to Photoplotters

The de facto standard for photoplotter data is the Gerber format, more properly known as RS-274D. The term Gerber refers to the Gerber Scientific Instrument company, a pioneer and leader in photoplotter manufacturing.

RS-274D is a variation on traditional Numerical Control (NC) machine tool languages. It differs from traditional NC formats (i.e. drill data), as far as its use of tool selection codes but is otherwise compatible.

RS-274D data is organized in "blocks". A block consists of a combination of codes:

Tool selection
Setup
Movement
And, an End Of Block (EOB) character, which only follows a combination of the above codes.

An EOB character is usually an asterisk ('*') or dollar ('$'), optionally followed by a carriage return and line feed.

An RS-274D code consists of a letter D,G,M,X,Y,I or J followed by a numerical value. These codes designate the following:

* - End of Block (end of command)

D - Select aperture, or set aperture use mode

X - Move to X value

Y - Move to Y value

G - Various setup codes

M - Various control codes

I - Relative X location for arc center

J - Relative Y location for arc center

D Codes

D codes have multiple purposes. The first is to control the state of the light being on or off. Valid codes for light state are D01, D02, and D03.

D01 - Light on for next move.

D02 - Light off for next move.

D03 - Flash (Light On, Light Off) after move (effect is limited to block in which appears, ie non-modal). You can also think of a D03 as D02, D01, D02 series of commands linked together.

D codes with values of 10 or greater represent the aperture's position on the list or wheel. It is very important to understand that there is no universal "D10" or "D30". Unlike the D01 , D02, and D03 counterparts which have a fixed meaning (draw, move, flash), D10 and higher values have aperture shapes and dimensions assigned to them by each individual user. Hence, one job's D10 could be a 10 mil Round, when another job's D10 could be a 45 mil Square.

There are two distinct ways to number an aperture list. The traditional 24 aperture system started with D10 - D19, jumping suddenly to D70 -

The X & Y values in the Gerber file determine where the aperture shape and dimension will be positioned and drawn. X & Y values are used as coordinate pairs to determine where the light will be exposed, using the D codes shapes (i.e. D10) and light exposure status (i.e. D01, D02, D03) for drawing lines and arcs, as well as moving between drawing entities.

Here are a few examples of using X & Y codes with D codes.

D10* { Select aperture D10}

X1000Y1000D02* {The D02 tells us that the light will be off, and we move to coordinate position X1000 and Y1000}

X2000Y3000D01* {The D01 tells us that we will draw (light on) to coordinate position X2000 and Y3000}

X5500Y100D03* {The D03 tells us to move to coordinate position X5500 and Y100 with the light off, then flash (turn the light on and off)}

G Codes

G codes are used to configure the photoplotter. Commonly implemented codes include:

G01 - Future X,Y commands are straightline moves

G02 - Future X,Y commands are clockwise arcs

G03 - Future X,Y commands are counterclockwise arcs

G04 - Ignore the rest of this block (used for Comments)

G54 - Prepare to change apertures

G74 - Future arcs are quadrant arcs

G75 - Future arcs are Full 360 arcs

G90 - Absolute data

G91 - Incremental data

Typically for laser photoplotters, G54 codes are rarely necessary. Older vector plotter controllers may require this preparatory G codes for changing apertures (i.e. G54D10*).

A common situation where G codes are mandatory for all machines is when the data is switching from vectors to arcs and vice versa. When switching from drawing vectors (G01) to drawing arc (G02, G03), the controller must be informed of the change of mode.

Another important case for G codes is when determining if the arc is a quadrant (G74) or Full 360 (G75). Quadrant arcs never cross quadrant boundaries, because the center coordinate offsets (I,J Codes) are always unsigned (even if they are negative!).

Therefore, it requires at least four G74 arcs to draw one complete circle.

Center coordinate offsets for 360 arcs (G75) can be positive or negative, allowing for a single command to draw a complete circle.

In either case, the center coordinates are given relative to the start point of the arc. The most dramatic difference between Quadrant and Full 360 arcs is that a Quadrant arc with identical start and end points has a sweep of 0 degrees, whereas a similar Full 360 arc is a full circle.

The G90 code tells the machine controller that all data following is absolute data. Hence, if following X & Y data follows, the controller will move to the absolute value given by the X & Y value.

G91 tells the machine controller that all data following is incremental data. The machine will move the data by the amount of the X & Y value, rather than to the absolute coordinate point.

Example:

X1000Y1000D02*

X3000Y3000D01*

In absolute mode (G90), the machine will first move to coordinate point X1000 and Y1000 with the light off, then draw a line to coordinate point X3000 and Y3000 with the light on.

In incremental mode (G91) the machine will first move to coordinate point X1000 and Y1000 with the light off, then draw a line to coordinate point X4000 and Y4000 with the light on. This was done by adding X1000 + X3000 = X4000 and Y1000 + Y3000 = Y4000.

NOTES

NOTES