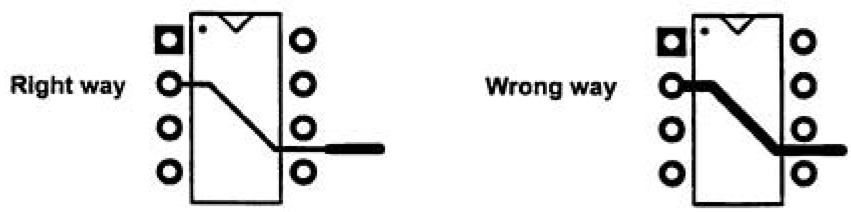
# *PCB Design Tips for PSPICE*

## 1. TRACES

**A.**In the ***Combo bar*** for the trace size pull down on the drop box and select ***trace_25*** . This is for a trace width of ***25 mils***. This size will ensure that there is enough copper trace for a proper electrical conduction. If a trace must go between two pads select ***trace_12.*** Trace sizes smaller than ***trace_12 (12 mils)*** are not recommended.

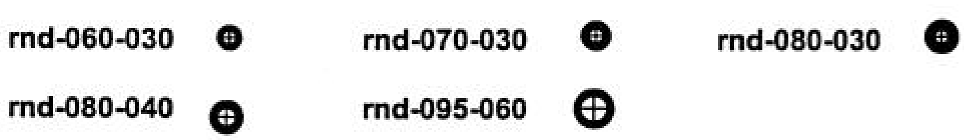

If you need to run a trace between two pads, such as an IC, use ***12 mils*** **between** the pads and ***25 mils*** for the rest of the trace route.




## 2. PADS or VIAS

**A.** In the ***Combo bar*** for pad size pull down on the drop box and select the proper pad size. Examples are below of the right pad sizes to use. These sizes will ensure that there is plenty of copper pad to solder the component too. The first number, ***.070*** stands for the ***diameter of the pad***, 70 mils. The second number, ***.030*** is the ***diameter of the drill size***, 30 mils.

**Note: There are only 4 bit sizes available for drilling –**
**30 mil 40 mil 60 mil 125 mil bits**



**B.** Some of the pads in the **component footprints** of Pspice are too small to solder to, but these pads can be changed with the ***Edit Attributes***  button in the ***Tool bar***. Highlight the component, then click the ***Edit Attributes*** button and look for the attribute name - ***Via Padstack***. This attribute can be changed to a size that is usable for soldering.

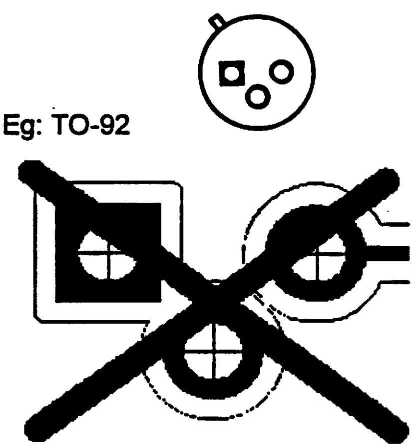
**RC05**
**rnd-054-034**
Pad size is too small for soldering

**RC07**
**rnd-060-030**
Usable pad size for soldering

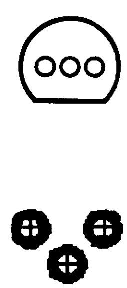
## 3. Placement of PADS and TRACES

**A.** Do not use **round** and **square pads** together on the same component, such as **transistor footprints**. Instead, use **3 round pads** together Do not place the pads and traces too closely together. The QuickCam Machine needs space to mill your design and ensure proper isolation of the components. Allow at least **12 mils** between **pads** and **traces**.

Eg: TO-92



**The WRONG way**



**The RIGHT way**

# Working Files Needed for MicroSim PCB & ISOPRO

## MicroSim PCB Files:

**COMPONENT (Red) is the TOP layer**
**SOLDER (Blue) is the BOTTOM layer**
**DRILL (Gray) is the layer for the drill sizes**

These are the only files needed to make a working PCB in IsoPro. Each one has to be individually saved and exported as a separate layer and then individually imported, into IsoPro, as a separate layer. Read step 4 of the handout **Milling a Circuit Board**, to learn how this is done. Combining them all into one DXF file will not work.

## IsoPro Files:

**BLUE is the COMPONENT layer**
**GREEN is the DRILL layer**

In IsoPro, once you have REGISTERED and MIRRORED all layers the next step is to ISOLATE the COMPONENT layer. This step will create an outline around the pads and traces of your design so that the pads and traces will be isolated from the rest of the copper on the board. Please read pages 17-23 provided from the QUICK CIRCUIT User's Manual. These pages will take you step-by-step through this process.

## Board Outline File

A board outline file in MicroSim is not needed. This file can be created in the IsoPro program or the QuickCam program.

Submit your drawing consisting of 3 files for a doubled-sided PCB (COMPONENT, SOLDER, and DRILL) or 2 files for a single-sided PCB (COMPONENT and DRILL) to the **Lab Technician** in **Room #309**, on a floppy disk or email them to the Lab Technician:

email address:______________________________

You should sumit your design 3 or more weeks before your project is due. The later you submit your design the less time you will have to make revisions on the printed circuit board. The time for milling PCB's is during working school hours: 8 am – 5 pm. The Lab Tech. does not mill circuit boards on weeknights or weekends.

# MILLING A CIRCUIT BOARD

Read the QUICK CIRCUIT User's Manual. Everything you need to know, to properly cut a PCB and use this machine, is in the manual. If you don't read it you could very possibly **DAMAGE** the OUICK CIRCUIT machine and cost the ECE dept. hundreds of dollars, or more, in repairs.

Steps to milling a board:

1. Load your drawing into the MicroSim PCBoard program

2. Check the Pads, Traces, and Drill holes to see if they are the proper size – refer to the **PCB DESIGN TIPS** sheet

3. Go to **FILE** then drop to **EXPORT** and click on **DXF**

4. A list of Board Layers will be displayed and highlighted in black. Go and click on the **CLEAR ALL** button. Next move the cursor arrow to the layer selection and click on the word **COMPONENT**. This will be the **ONLY** layer highlighted in black. Next click on the **OK** button. It will ask you for a file name and where you would like it to be saved. **SUGESTION:** *Whatever name you select at the end add the word COMP so you can remember that this is the component file and not get it confused with the drill file or other milling files.*

5. Next go back to step 3 then repeat step 4. But this time highlight the **DRILL** layer and save it. **NOTE:** *For a double sided PCB you will have 3 files; the COMPONENT file, SOLDER file, and the DRILL file. For a single side PCB you will have 2: the COMPONENT and the DRILL file.*

6. Close the MicroSim program and then open up the IsoPro program.

7. Under **FILE** drop to **IMPORT** and select **DXF FILES.** Find the 2 files you saved, **COMP** and **DRILL**, and open up the **COMPONENT** file. As it coverts the file it will ask you for the **DXF IMPORT SETTINGS**, just click the **OK** button.

8. Your drawing is now converted to the IsoPro program. There should be 2 layers; a **BLUE** layer with the pads and traces and a **GREEN** layer with the drill holes.

9. **Now go to PAGE 17 of the QUICK CIRCUIT manual.** Starting with the paragraph about the layer list these pages will give you step-by-step instructions on how to **MIRROR** the layer, **REGISTER** the layer, **ISOLATE** the layer, and finally **EXPORT** the layer as a **QUICKCAM** file. The QUICKCAM files are the final files the machine reads in order to mill a circuit board.

**PLEASE READ THE QUICK CIRCUIT MANUAL AND FOLLOW IT STEP BY STEP AND YOU WILL BE ABLE TO MILL A CIRCUIT BOARD PROPERLY**

# ISOPRO TUTORIAL

IsoPro imports each file into a separate *layer*. After loading the three layers, each one is 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.

**Layer List** 

Your *layer list icon* looks like a stack of four sheets of paper or four planes on the upper toolbar. You can use the layer list icon to control data on a layer by layer basis. You can change the layers name, type, aperture table, color, and status. The status allow you to control the viewing and editing of layers. There are three possible status modes. They are:

1. **View -** Allows you to see the layer, but you cannot edit the data. It is helpful for using a layer as a logical reference.
2. **Edit -** Allows you to modify, select, delete, mirror, and edit the data on the layer.
3. **Hide -** Allows you to hide the layer which prevents confusion while working on other layers.

You can also set the layer's type from this dialog. Go ahead and set the type for '274X.GBR' to '*Component*' and set the type for '274X.SOL' to '*Solder*'. You will notice that the drill file is already identified as '*Drill*'.

## 2.5 LAYER REGISTRATION

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

The tutorial files provide you with a component side layer and a solder side layer that do not register over the top of each other. To correct this data so that you can logically view the two layers as they display on your computer monitor we must first mirror the 'Solder' layer.

**Important:** Inspect your CAD files to determine if the solder side is mirrored. If your solder side is mirrored, the holes on each layer will <u>NOT</u> line up.

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

Use the following steps to register each layer:

**Step 1:** 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 2:** Set the 274X.NCD layer to **Hide.** It is already registered, so there is no need to view it.

**Step 3:** Set the 274X.SOL layer to **Edit** and close the layer list. The only data that can now be edited is the solder layer.

**Step 4:** Zoom out and click on the select icon. Then, use the mouse to *drag* a box around the entire solder file (You can also use the **Select All** command in the **Edit** menu). The selected data turns gray. The component side does not turn gray because it is still in *View* mode and cannot be selected.

**Step 5:** The next step is to *mirror* the board. Click on the Mirror icon.

**Step 6:** Click on the **Deselect** icon and use the mouse to *drag* a box around the entire file or use **De-Select All** from the **Edit** menu.

**Step 7:** Next, use the **Layer Registration** icon to select the register tool. The first step is to 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 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 the previous undo action.

**Step 8:** Once you have the layers registered, you can use the Layer Table to set the status of each layer to *Edit*.

**Note:** In this tutorial the drill layer was already registered with the component side. However, if the drill file was not registered you would follow all the same steps for the drill layer that you did for the solder layer.

**Save the File**

At this point you should save your work. Under the **File** menu, select the **Save As** command. Name the file "TUTORIAL STEP1.ISO". This will allow you to come back to this step in the tutorial easily if you wish to practice

IsoPro files are saved as *.iso files. A *.iso file includes all work done in IsoPro, including all layers, aperture lists and drill racks. Work saved in a *.iso file may be restored at any time by selecting the **Open** command under the **File** menu.

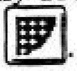
## 2.6 VERIFY THE APERTURE LIST

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

You should verify that your Aperture List is correct for you circuit board. The aperture list can be viewed by clicking the aperture list icon. For this tutorial your files use the RS274-X standard, so all of the apertures were imported directly without intervention from the user. T-Tech strongly recommends using RS274-X. Most CAD packages released within the last three years support this format. If your files are not RS274-X, and you do not have an aperture list loaded, you will notice that your 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 this value, please double-check it and manually edit the values to match those output from your CAD package. Most CAD packages, when not using RS274-X, output the aperture list as a separate report file. Converters are available for some CAD packages but it is also easy to enter the list manually.

## 2.7 EDIT THE DRILL RACK

**Purpose:** To verify the drill sizes used for your circuit board's drill holes.

The drill rack usually does not import into *IsoPro* automatically, so you must edit it. Click on the Drill Rack icon . Your drill sizes default to 24.99. Again, the '.99' is a flag that indicates these are the default values. For this tutorial, simply set both tool sizes to 25.00 mils, but 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 output by your CAD package or sometimes they are in the header of the drill file itself. If the latter is the case then you can view it by opening the file in *Notepad* or a similar editor.

To verify the size of a particular drill hole you can check the properties of the drill hole. To check the properties, place the cursor over the drill hole and click the right mouse button then select properties. This dialog will show you the diameter and shape of drills, pads, and traces and their center or starting/ending points.

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. If you want to do some final editing without going back to the CAD package you can change the size or shape of an individual pad or trace.

## 2.8 ISOLATE 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.

**Step 1:** Now that you have verified that your aperture list and files are correct, you need to isolate the component and solder layers.

**Step 2:** Set the status of the drill and solder layers to *Hide* (this is not a required step, although it can help prevent mistakes and makes your first few board designs run a little smoother).

**Step 3:** Click on **Tools** and then **Isolate**. You will now perform two isolations. Enter the two diameters above under "Pass 1" and "Pass 2". Enter a zero diameter for "Pass 3"; this indicates that this pass is not used. Then select the component and solder layer by holding down the CTRL key while selecting both layers. The tool size can be entered in inches or mm. Metric mode can be selected from the **Preferences** selection under the **Edit** menu.

**Step 4:** Click on the **Isolate** button.

Now would be a good time to save your files again, save it as TUTORIAL STEP2.ISO.

# ISOPRO TUTORIAL

IsoPro also provides a **Remove Redundant** option during mulit-pass Isolations. This feature locates and deletes those sections of the smaller tool isolations which are completely overlapped by larger isolations. On some types of circuit boards, the use of Remove Redundant can offer significant savings in machine time and tool usage. The remove redundant option is enabled and disabled via a small check box in the Isolation dialog. Please note, the speed of the Remove Redundant feature is proportional to the square of the size of board (It is fast on medium size boards, but can be very slow on larger boards).

**Inspect 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 to drag a window around an area of interest then use the pan-hand to move around and inspect the data.

**Clearances and Solderability**

The first pass for this design was 10 mils (.010"), but the diameter you choose for your design should be less than any pad-to-pad, pad-to-trace, or trace-to-trace clearance. This is so that the tool can isolate all of the nets. Choosing a diameter that is too large will result in the merging of nets on your board. For easier solderability, a larger second pass, in this case 31 mils (.031"), is recommended. This is typically all that is required for a digital board. RF/Microwave and analog customers might want to rub out or remove all unwanted copper for a section of the board or for the entire board.

**Rub Out the Base Copper**

Ordinarily, you would not perform a rub out on a digital board; however, if you have a set of fingers (edge connector) or something else (SMT component) that requires a rub out, you can rub out any area of your board or the entire board. To do this, use the following steps:

Step 1: Rubout only works on one side at a time. If you want to perform rubout on both the component and solder sides you must repeat this procedure for each side. For the component side case, set the isolation layers for the component side to edit and set the isolation layers for the solder side to hide. (Conversely for the solder side case, set the isolation layers for the component side to hide and set the isolation layers for the solder side to edit.) Zoom-In to the area that you want to rubout. Rubout will automatically select the largest isolation layer in edit mode.

Step 2: Click on the Rubout icon. Now drag a window that describes the area in which you wish to remove all of the excess copper. Repeat this step as necessary.

To prevent unwanted copper slivers during machining, the rubout tool paths overlap by a default percentage of 20%. This percentage may be increased or decrease by clicking on the **Preferences** selection under the **Edit** menu and then editing the "Percent Overlap for Rubout" field located on the **Advanced Settings** page.

## 2.9 EXPORT YOUR FILES

You are now done with your design except for writing the files that will be used by the prototyping machine to disk.

**Step 1:** Go to the layer list and set every layer that you want to export to *Edit* or *View*. After doing this, inspect the drill file, the component isolation, and the component rub out. You want to export the isolation layers, the rubout layers, and the drill layer(s). You do not need to export the Gerber files, because they are your original CAD files.

**Step 2:** Click on **File** and then **Export**. You will notice that each time you performed an isolation or a rubout, *IsoPro* created a new layer with the original layer name. The difference being that the new layers have their type set to "Isolation" or "Rubout" and the tool diameter has been appended to the end of the layer name. You want to export a QuickCAM file for each of the following layers:

| | |
|---|---|
| 274X.GBR (.010") | Isolation |
| 274X.GBR (.031") | Isolation |
| 274X.GBR (.031") | Rubout |
| 274X.SOL (.010") | Isolation |
| 274X.SOL (.031") | Isolation |
| 274X.SOL (.031") | Rubout |
| 274X.NCD | Drill |

**Note:** Since QuickCAM (the machine driver for Quick Circuit) is not written for Windows 95 or Windows NT you should make sure that the file names you choose are no longer than eight characters. The ".PLT" is appended automatically.

For files to be used by Quick Circuit they must be loaded into the QuickCAM software. All of the above files should be initially loaded into QuickCAM as Isolator files. These files use T-Tech's modified HPGL format and may contain mill, drill, or route data. QuickCAM can only load one file at a time. Before beginning to mill or drill your circuit board, it is recommended that each file be viewed in QuickCAM. This allows you to inspect all of your files before you begin to produce your circuit board.

**Save Your Work**

This completes the procedure to prototype your circuit board. Remember to always save your work before exiting IsoPro. This enables you to return to IsoPro for editing.