

# ELECTRONIC CIRCUIT AND PCB DESIGN TIPS FROM AN OLD PRO

DM Fraser

Updated Wednesday, May 27, 2020

# Contents

Introduction. ....	<a href="#">4</a>
Procedures for board design. ....	<a href="#">5</a>
Engineering practices for printed circuit layout. ....	<a href="#">6</a>
Connectivity Check. ....	<a href="#">6</a>
PCB Decals. ....	<a href="#">8</a>
Grounding. ....	<a href="#">8</a>
Star Ground. ....	<a href="#">8</a>
Multiple Ground System. ....	<a href="#">9</a>
Zero Ohm Resistors. ....	<a href="#">11</a>
Single Ground System. ....	<a href="#">12</a>
Chassis Ground. ....	<a href="#">15</a>
Switching Regulators. ....	<a href="#">16</a>
Mitters. ....	<a href="#">16</a>
Routing. ....	<a href="#">18</a>
Autorouting . ....	<a href="#">18</a>
Keepouts. ....	<a href="#">18</a>
DC power input connector wiring. ....	<a href="#">19</a>
OLD PRO TRICK. ....	<a href="#">22</a>
OLD PRO TRICK. ....	<a href="#">24</a>
Bypass Capacitors on ICs. ....	<a href="#">25</a>
Power supply filter capacitor wiring. ....	<a href="#">27</a>
Trace Width. ....	<a href="#">27</a>
Trace Clearance. ....	<a href="#">28</a>
.....	<a href="#">28</a>
Teardrops. ....	<a href="#">28</a>
Trace Naming. ....	<a href="#">29</a>
Routing Op-Amps. ....	<a href="#">29</a>
Clock Lines. ....	<a href="#">30</a>

Component Spacing. ....	31
Libraries. ....	32
Double check that all pins are correct in your PCB decal. SOT23, LED decals and 8 Pin IC decals have sometimes had issues due to some bad decals in the libraries supplied by Mentor Graphics. If a part is imported from a different library this has been an issue in the past. ....	36
. ....	37
Text. ....	37

**The following are not in any particular order yet**

## **Introduction**

The design tips in this document are mainly oriented towards design and manufacture of audio equipment. These techniques may be used on any sort of low frequency equipment where noise pickup may be an issue.

How the heck did I become an old pro I still remember too well my days taking electronics in high school then going to the local university and technical institute. I only graduated in 1972. Seems like just a year or two ago. The government says I am old I guess having started to give me my pensions. In a career of nearly 50 years I have worked on many things in Canada as well as 10 years in the USA. I have been fortunate that I have been able to devote most of my energies professionally in the field I am most passionate about. Audio. Both consumer and professional. For most of my career I actually worked at my hobby and got paid for it.

Though I did stray outside audio now and then. My one patent is for the control system a diesel fuel fired heater for field kitchens. I also worked on marine autopilots and written software for automated background music systems. Then there was the 10 year detour in charge of the maintenance of the Audio/Visual equipment and Classroom computers of a large school district with over 100 school spread out over 250+ square miles.

When I moved to Orange County, California in 2001 I designed signal processing, amplification and distribution products for high end professional loudspeaker systems where products I designed went all over the world and into some very high profile installations. Where I worked with state of the art Class-D amplification, Ethernet based audio distribution with Cobranet, applies high end professional grade audio A/D and D/A devices and digital signal processors.

I mastered PADS Logic and Power PCB and designed boards up to 6 layers. Boards that had to be super quiet and yet very small while having large digital chips on them and very sensitive A/D and D/A converters.

In 2010 I move back to Vancouver due to family issues and went to work designing high quality sonically superior electronic devices for musicians. Much of it for the touring market. It is likely that when you go to any major music show in the world that there is a high chance that there are some electronics I designed on the stage. Then few years I was given the opportunity to bring back to production two very highly regarded lines of audio gear that has disappeared. One line solid state and the other vacuum tube based.

I was the very last generation that had their basic electronics education with vacuum tubes when I was in high school.

I still have tons of projects I want to build. I have a web site full of ideas and things. You may have even found this document there.

# Procedures for board design

These procedures are guidelines for the process we sort of follow where I have been working. At any stage there may be back and forth communications required between the originator, the Engineer and Design as well as marketing and others. There may even be loopbacks to a previous point in the process at times.

Revising a board for the next version will be a subset of these procedures.

1. Basic exterior design of product will come from the Product Originator, usually the Tech Team in marketing.
2. Engineer to create block diagram of the product functionality and save that to a new project directory in the Engineering\Products folder on the server.
3. Block diagram to go back to originator of product to see if this is what they intended.
4. Back and forth occurs until a consensus on the block diagram is achieved. The feature set may change during this process.
5. Engineer to create the schematic for the product. This includes getting the attributes right for all the parts so the BOM can be created from the schematic. Reuse of already proven sub circuits for common functions from previous corporate products is strongly encouraged. The wheel is not to be reinvented.
6. Engineer to create the PCB outline. This is generally done by taking the PCB layout of an existing product in the same package and modifying it as required.

Schematic and PCB is sent to Design<sup>1</sup> for adjustment of external component placement and 3D modelling of the PCB. Design may move some internal parts without regard to any electrical considerations in order to get the external part placement to export to Solidworks. Design will also add drawing numbers to the schematic and PCB and add those to the file name to bring them to company standard.

7.

Design will work Product Originator on the final placement of external parts. Substantial movement may occur from the original positioning by the Engineer. There may even be addition and deletion of features and the Engineer may have to revise the schematic.

8. Once external parts are placed, the layout goes back to the Engineer. Engineer should make all external parts are "Glued" in place.
9. The Engineer shall then place internal parts where they make most sense electrically and route the PCB. This includes copper pours and everything else required electrically. There must be no errors on the reports between the PCB and schematic when doing a Comparison on Pads Layout Link unless absolutely unavoidable and documented.

---

<sup>1</sup> Design is the physical design/3D modelling area. At the time of writing that is the SolidWorks operator.

10. The board will then be sent to Design for cleanup of the documentation layers on the PCB and all the documentation around the PCB on the layout.
11. Engineer is then to review the layout and release it back to Design
12. Design is to create the gerber files for board production. Design is to also create a blueprint document with BOM and assembly instructions. Engineer input may be required where part attributes are vague.
13. Engineer is to use a Gerber viewer to approve the Gerbers for release.

## **Engineering practices for printed circuit layout.**

All the examples are made with Pads Logic and Pads Power PCB from Mentor Graphics.

Why not Altium as that is the way that many people go. When I moved to LA in 2001 the company where I worked had already used Pads for some years and Altium was just coming out. I became an expert on PADS very quickly and Altium was the new kid on the block. The company never saw enough advantage in Altium over PADS to spend the money to switch over. The licenses for these programs can run to US\$10,000 per license. And we had 3 of them at the time. More now.

### ***Connectivity Check***

While it is very important to use the Connectivity checking function on a printed circuit layout, the connectivity checking does not tell you anything is connected in the correct sequence. If there is an error on the schematic, this function will propagate it to the PCB. Therefore it is important that the schematic is correct. There were some incorrect IC and LED decals supplied with PADS originally and while efforts have been made to eliminate these incorrect parts and decals it is important to still check carefully. Particular offenders have been LEDs and 8 Pin dual opamps reversing the + and – input pins. The pin number sequence in the CAE decal in the schematic are not necessarily the pin sequence sent to the PCB decal.

Looking in the Pins tab of the part attributes as shown in the following graphic, it can be seen that the pin sequence and pin numbers in a gate are not necessarily the same. Caution is advised.

Part Information for Part - SW--4PDT-PUSH-8MM\_TALL

General | PCB Decals | Gates | Pins | Attributes | Connector | Pin Mapping

Tip: Multiple values in one column can be edited at once by using the Edit button.

Pin Group	Number	Name	Type	Swap	Seq.
Gate-A	2		Bidirectional	0	1
Gate-A	1		Bidirectional	0	2
Gate-A	3		Bidirectional	0	3
Gate-B	5		Bidirectional	0	1
Gate-B	4		Bidirectional	0	2
Gate-B	6		Bidirectional	0	3
Gate-C	8		Bidirectional	0	1
Gate-C	7		Bidirectional	0	2
Gate-C	9		Bidirectional	0	3
Gate-D	11		Bidirectional	0	1
Gate-D	10		Bidirectional	0	2
Gate-D	12		Bidirectional	0	3
Gate-E	13		Undefined	0	1
Gate-E	14		Undefined	0	2

Reset  
Edit  
Add Pin  
Add Pins...  
Delete Pins  
Renumber...  
Copy  
Paste  
Import CSV

Check Part OK Cancel Help

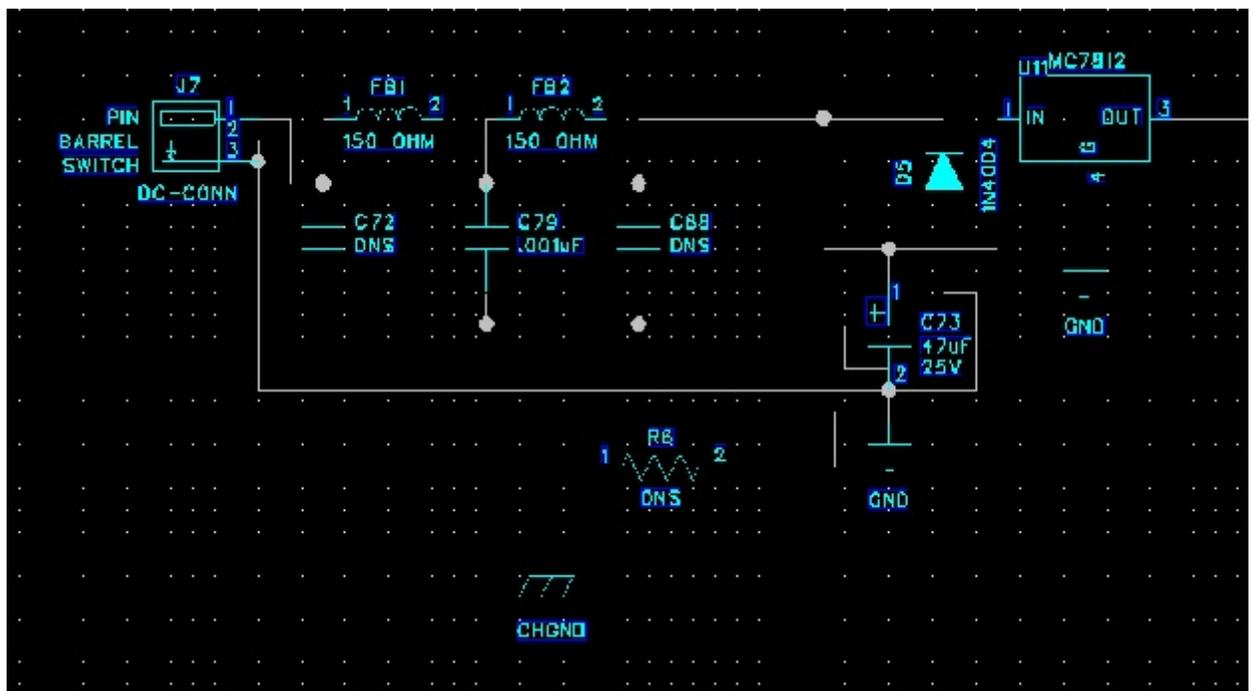
## PCB Decals

As much as possible, PCB decals should have their origin point in the physical center of the part to allow easy rotation. And in the case of SMT parts, the center of the decal will then also be at the pick and place center.

## Grounding

### Star Ground

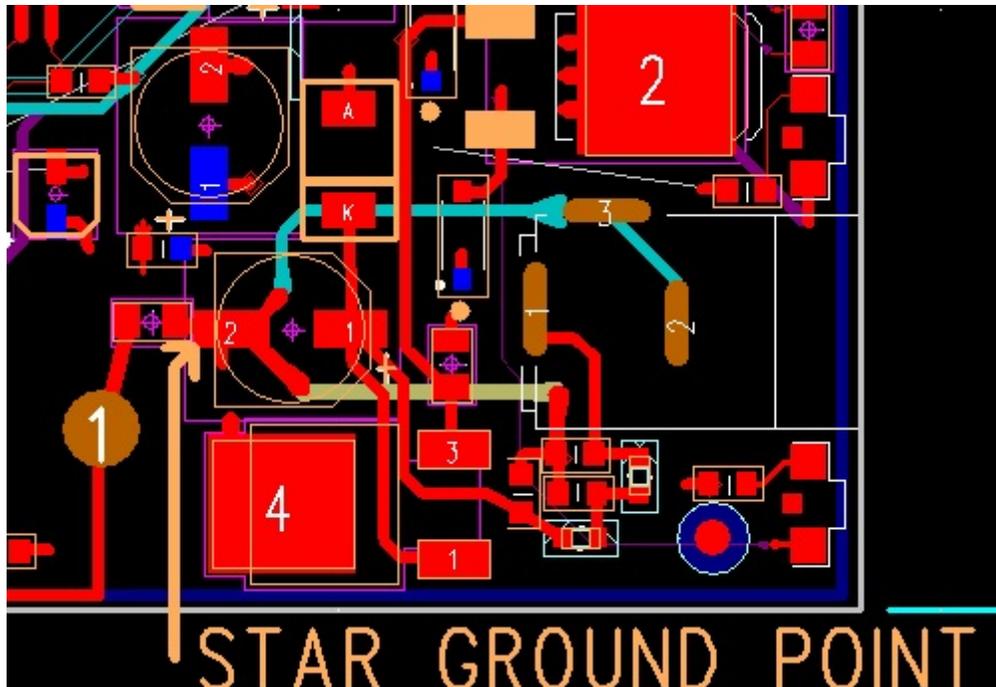
It is recommended that every active PCB has one single point where the various ground paths meet. Especially mixed signal analog/digital PCBs. In the graphic below, that is C73 pin 2.



One can see the ground for the input, the chassis ground and the single ground setup all meet there. If there are separate analog, digital or other grounds, this is also where they would meet.

There is more on this in the DC Adapter section.

The routing is shown in the next graphic.



The star ground shown has the power jack ground shown in the cyan trace and the ground of the input filter shown in tan going to pin 2 of the capacitor. As well from pin 2 is the zero ohm jumper going to chassis ground. There is a short trace going down from pin 2 which is to a via connecting to the general ground planes. It can be noted that none of these grounds share any conductors to the pad on pin 2.

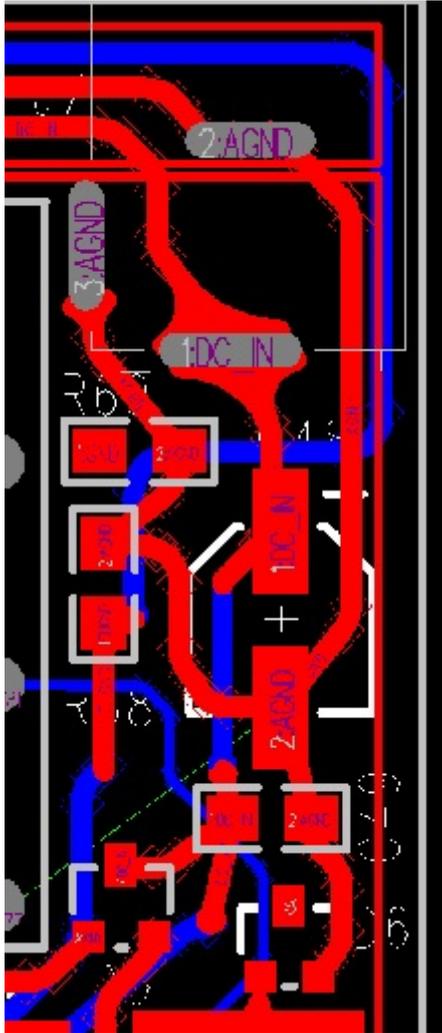
## Multiple Ground System

The writer prefers to use multiple grounds back to the star ground point with separate electrical grounds for analog, digital, power input, chassis ground and other ground connections.

Connection between them being made with zero ohm resistors at the ground connection of the power input filter capacitor. Care must be taken that analog, power, digital, chassis and other grounds share no other common connections. If connections must be made elsewhere they must be isolated with small inductors. Connections to the zero ohm jumpers must either directly to a plane of with conductors as wide as possible. At least 25mils wide, preferably wider.

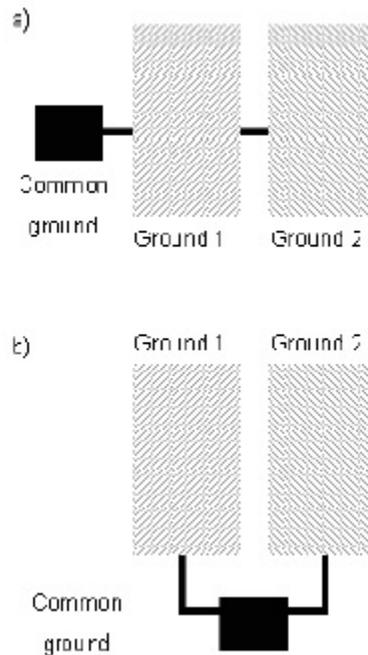
In this graphic from the Twinline R62 R68 are the zero ohm jumpers for the connection id the various grounds and they connect to the ground of C48 through a ground plane... The point here is that are all located at the power input as close as practical to the ground lead of the input filter capacitor.

In the case where the power input has no connection to ground and there is a ground splitter, the ground plane grounds do not connect to the power input filter but to the ground splitter output as close as possible to the output of the ground splitter.



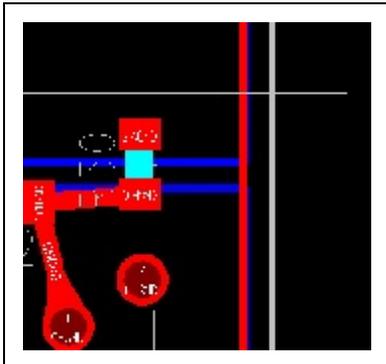
The various grounds should only connect to each other at the ground terminal of the power input filter capacitor. In no case may the ground current from one ground plane pass through a different ground.

**Figure 8:** Do not let one ground plane pass another ground plane to get connected to the common ground (a). Every ground plane must have its own path to the common ground to reduce noise (b).



**Figure 8. Poor and Good Placement of the Common Ground in a Split Ground Environment**

## Zero Ohm Resistors

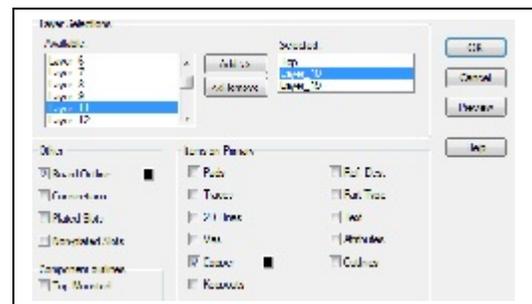


Zero ohm resistors are used for connecting various traces with different names to the same point. They are especially used for keeping analog, power and chassis grounds apart. As per the graphic to the right showing R30. This is in the parts library as part type RSM0R-0805. This is to be used on both through hole and surface mount PCBs as no actual part is stuffed.

Looking at R30, a light blue band can be seen connecting the pads.

When running the gerbers for the layer the zero ohm jumper is on, one must turn on Layer 10 and select Copper for the connection to be made. See the graphic to the right.

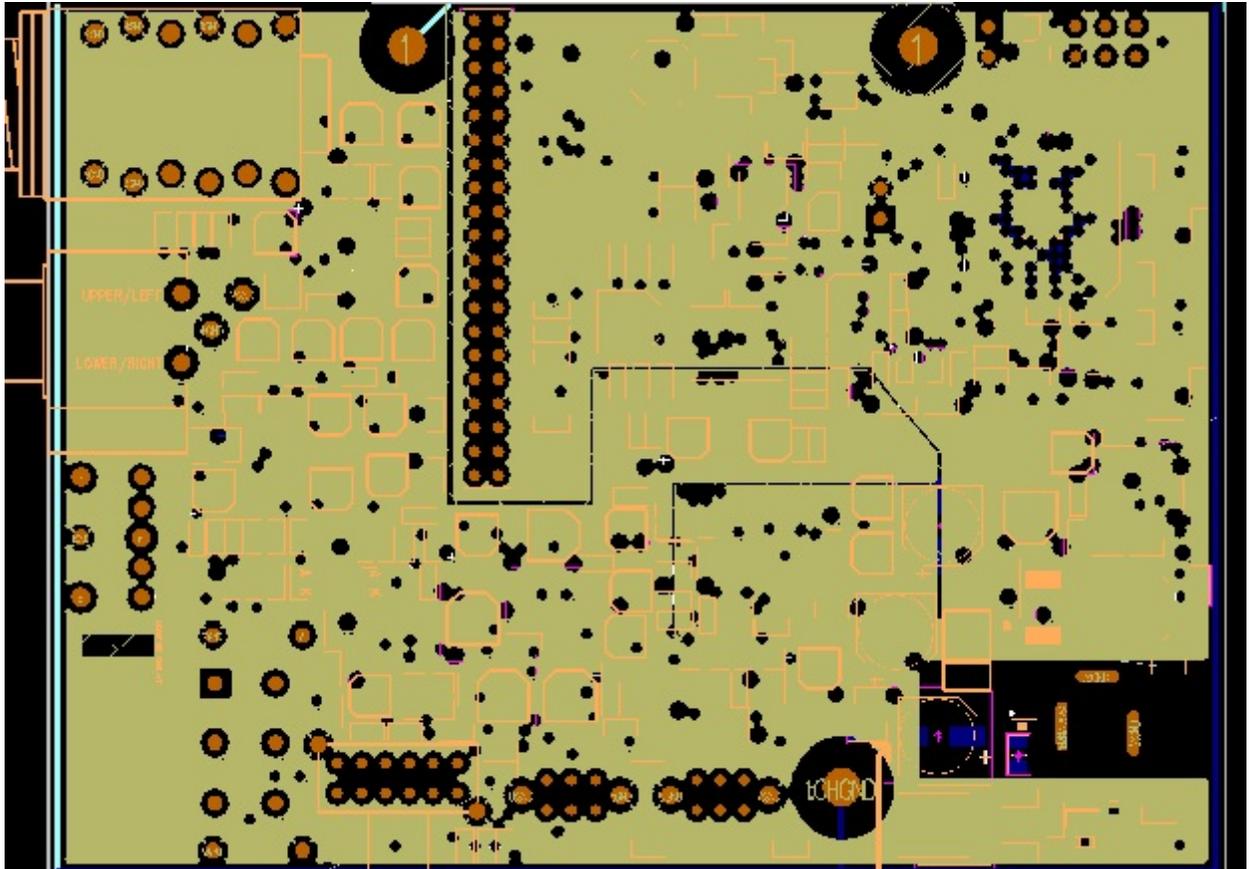
When running the BOM make sure the component is shown as DNS.



## Single Ground System

This is where a PCB designer prefers there to be a single ground. It is very tempting to have just a single ground for everything. Treating ground like a sewer and throwing everything onto it. This is OK sometimes but in the modern regulatory world, a product with a board with digital circuitry on it, especially in a mixed signal environment will generally not pass Electro Magnetic Compatibility (EMC) testing.

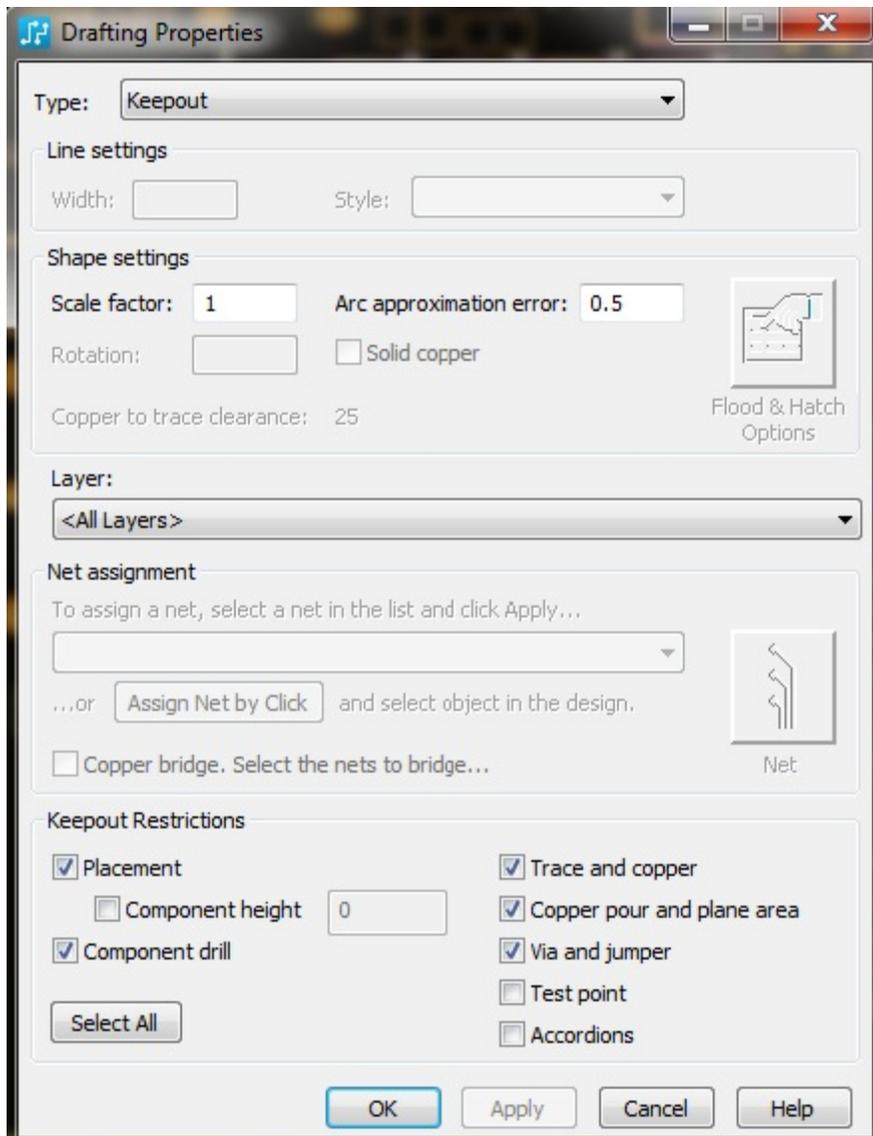
To get around this by putting cutouts or keep outs in the ground planes to make sure digital, analog and power circuit and other ground returns only go to the star ground point are used.



In the above layout keep outs can be seen that section the PCB into various areas. The left side is the analog while the upper right is the digital section. A separate section near the center is for LED drivers. These cutouts are arranged to funnel each type of current return to the star ground point next to the power connector cutout on the lower right corner.

It is important that no conductors or components cross these cutouts. The drafting properties of the cutouts have to go through all layers and not allow traces, copper or placement. The only exception to the no placement rule is that CODEC chips like the CS4270 which have an analog and digital side. However this will cause a DRC error when checking the keep out clearances.

See the PADs drafting properties illustration following.



There is also a school of thought that the keep outs should be used only to guide routing and component placement but copper coverage should be continuous. In which case the “copper pour and plane area” box would not be checked.

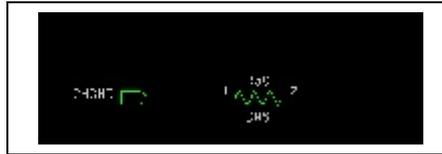
<http://www.aerospacepal.com/pcb-plane-cuts-kill-emc/>

## **Chassis Ground**

Care must be taken that no current flows through chassis ground connections. Any current flow will impair the effectiveness of shielding.

The chassis ground must be connected to the power input filter capacitor ground (star ground) connection through a trace as wide as possible. Chassis ground has to be routed in most products to at least one mounting pad so that a connection is made to the case when the product is assembled. While no current flows through a chassis ground conductor it is important that the impedance be kept low which means a trace as wide as practical.

Besides the single connection to the power ground, balanced audio connectors such as XLR connectors may have a chassis ground connection on pin 1. Unbalanced inputs are not connected to chassis ground.



ground connection connected to

## **Stackup**

On a 6 layer board there are many possible layer stackups. The writer has used, from top to bottom:

COMPONENTS

GROUND – NO ROUTING

ROUTING

ROUTING

POWER

GROUND WITH MINOR ROUTING – Where not quite everything could not be done on layers 3 and 4.

All layers would have ground pours though to fill in any unused space on each layer.

Here are some links with opinions on the subject.

<http://www.aerospacepal.com/6-layer-pcb-designstackup/>

<http://www.hottconsultants.com/techtips/pcb-stack-up-1.html>

<http://www.hottconsultants.com/techtips/pcb-stack-up-3.html>

<http://www.bitweenie.com/listings/standard-4-and-6-layer-pcb-stackups/>

## Switching Regulators

Switching regulators can be a huge problem for emissions causing problems passing FCC requirements. It is recommended that the manufacturer's spec sheet be obtained and checked to see if a recommended board layout is available. If there is one, follow it as precisely as possible.

## Miters

Traditionally miters have been done at 45° angles. In the case of high speed circuits it has been recommended that arcs be use instead. These are not difficult to do in Pads and the writer likes how they look as well. It is suggested that the designer consider the use of arcs miters in cases where PCBs will be in products sent for EMI approval. It has been estimated that arc for miters reduce EMI emissions by 2 to 10db which can be the difference between a pass and a fail.

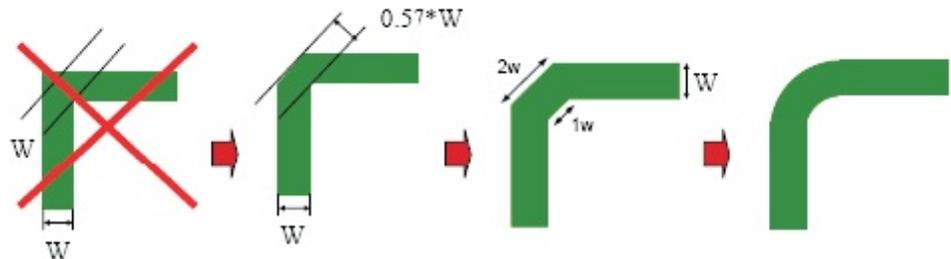
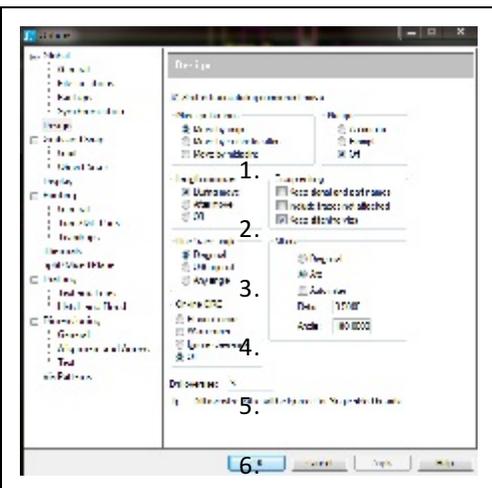


Figure 13. Poor and Good Right Angle Bends

In order to do arcs in PADS the board must be set up for them. Go to Tools-Options. Click on the Design selection. Set "Miters" to "Arc", the Ratio to 3.5 and the Angle to 180.00 as per the following graphic.



After your board routing is completed to make the corners into arcs:

Press ESC to clear the existing command

Right click the mouse and choose Select Pin Pairs

Press the Home Key

Select the entire PCB

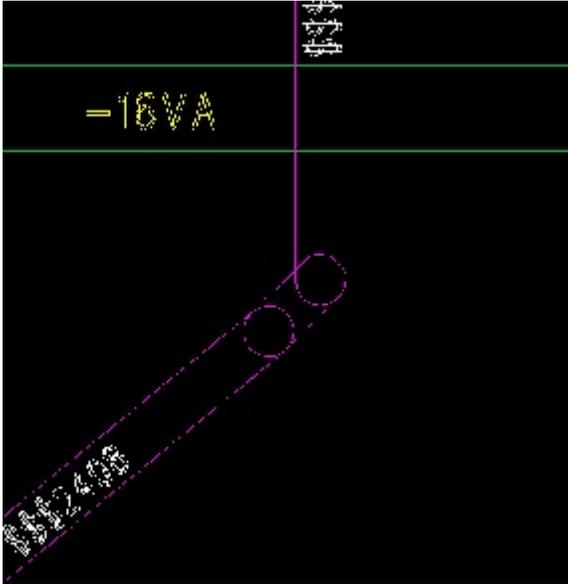
Press the "M" key

Choose "Add Miters"

7. Click outside the board area to deselect the pin pairs
8. Right click the mouse and choose Select Anything

However some sources feel that arcs make no difference with signals below 1 GHz

If you get a corner that refuses to make an arc that is because one of the traces is split. Press the "O" key to switch to outline view. See the following graphic.



1. Highlight the point where the trace is split and press delete.
2. Once the split is gone, highlight the angle itself
3. Press the M key
4. Select Add Miter
5. Press "O" again to return to normal view.

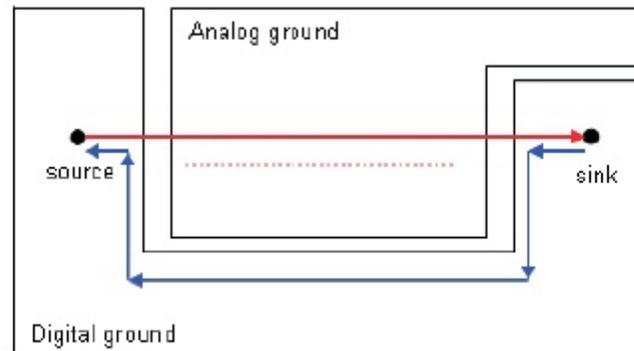
## ***Routing***

### **Autorouting**

Do not trust the autorouter. While it may make all the connections on a PCB, the program has no idea of how the wiring of components needs to be sequenced and will almost always do power and ground sequencing incorrectly. It is recommended to not even use it as the time needed to correct its errors will often exceed the time needed to manually route a PCB.

## Keepouts

Digital signals must not cross analog ground planes and vice versa in mixed signal PCBs. Keep outs should be used to insure traces do not cross between the different sections of the PCB.



**Figure 7. Loop Area and Crosstalk Due to Poor Signal Routing and Ground Splitting**

In digital boards there are often sets of traces that need to be run in as close to parallel as possible and end up about the same length. These sets may include

SDA and SCL

MCLK, SCLK, LRCK and SDATA. SCLK may be called BICK. LRCK may be called LRCLK. SDATA may be called SDIN, SDOOUT and may have a number after the name in some cases.

Similarly, balanced audio lines should also run as parallel as possible and be about the same trace length. These trace names often start or end with "+" and "-"

On very dense boards, one may want to make the pads of through-hole parts smaller on the component and internal layers of the PCB while leaving the pads on the solder side full size for good soldering.

## DC power input connector wiring

Unlike products that run from AC line power, many smaller audio products operate from a DC power supply or adapter. Where power comes from an external DC source, usually some sort of AC to DC converter. A common type of power supply or power adapter is often called a "Wall Wart" as shown below. These may be regulated or unregulated.



Regulated adapters, as shown to the left are often marked as suitable for line voltages from 90V to 260V or so for worldwide applications. These give pretty close to the exact output voltage marked on them from close to zero load current up to the maximum rated load current marked on them. If these generate noise it is usually high frequency noise above the audio range. If not properly wired these will produce a high frequency hash in the audio signal, a low level grittiness in the sound and rarely, a whine.

Unregulated power sources usually only have a single input voltage marked on them like 120V or 230V. Unregulated adapters, when a load is not connected will give an output voltage about 25% above the marked voltage. For example, an adapter marked at 15V/400mA will deliver about 19V with no load connected.

Increasing the load current will cause the voltage output to fall and the rated voltage is given when the load current matches the adapter's rating. These adapters often have a substantial amount of 100Hz or 120V ripple on their "hot" lead as. This will also appear in the audio signal as a sort of buzz if the adapter's ground input is not routed properly.

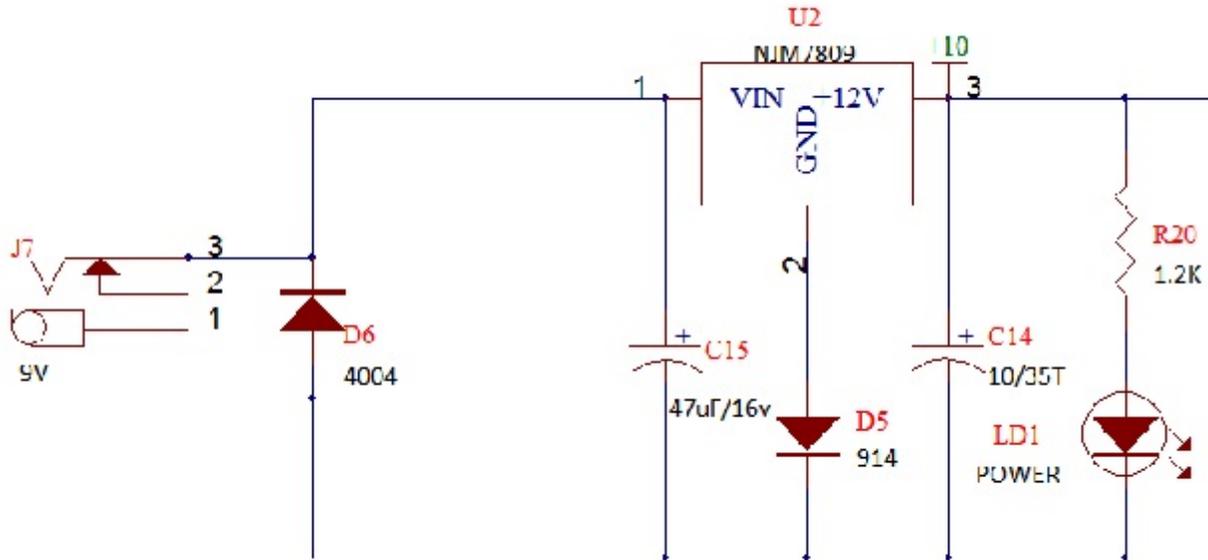


Both types need special care in the schematic design and product board layout to prevent noise from getting into the product you are designing.



The DC jack shown on the left is a common type of connector used for power adapters, especially unregulated ones. Many regulated adapters may be equipped with a micro USB connector.

Note that the conventional way of drawing schematics rarely gives the correct way to sequence the connections. Especially ground connections. A conventional schematic as follows gives no clue that the sequence of the wiring of the hot leads and grounds is very important.

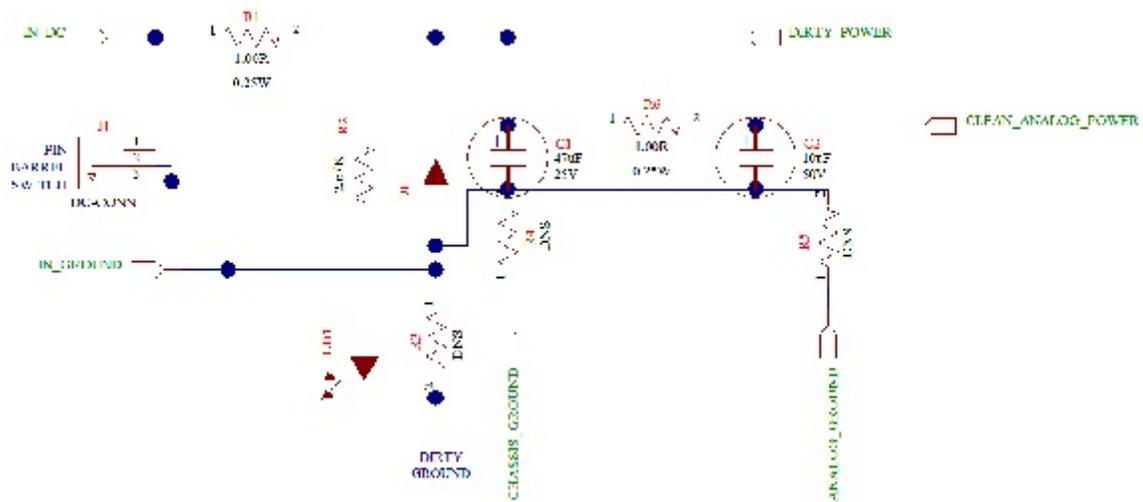


The above schematic implies a routing where the input capacitor, C15, is off at a “T junction all by itself. This is the worst way to wire it as it would require the ripple current to enter and exit the capacitor from the same wire. This is like trying to use a single pipe to pump dirty oil into an oil filter and expect to get clean oil back out the same pipe at the same time. In DC circuits it is important that all the power currents flow in one direction only for best performance.

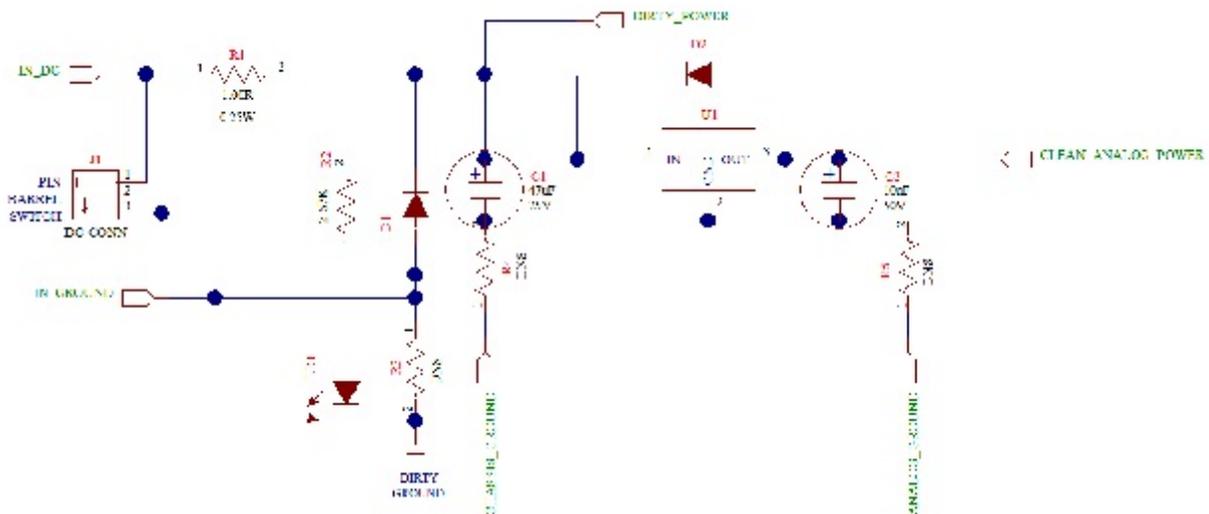
The issue is caused by the rectifier current from the power adapter which is not a continuous flow. In the case of a 50/60 Hz power adapter, the current comes in pulses at a 100/120Hz rate (double the AC line frequency in most cases) even though the voltage may look smooth or only have a small amount of ripple. . The filter cap in the adapter may not be wired correctly to minimize these pulses. Noise from the rectifiers is an annoying low level buzziness from a 50/60 Hz power supply and white noise when the source is a switch mode power supply.

It is very critical for the pins of the power input connector to NOT be directly connected to the ground plane of a PCB. The ground side of the power input will be noisy whether the source is a linear supply or a switching power supply. Both the power input and the ground must be connected to a filter capacitor before going to any other place. The only exception would be to a polarity protection diode for the ground side. However this must still not connect to the ground plane. Ground plane connection is to be made only after the connection to a capacitor ground, never where the DC power connector is mounted. On the hot side the connection to the capacitor may be made through a series filter resistor or inductor.

Following is how schematics should be drawn to illustrate the correct routing of traces. The first schematic is unregulated and the second is regulated using a three terminal regulator IC.



UNREGULATED



REGULATED

It can be seen the DC input goes directly to C1 via R1 and the polarity protection diode. Then by a totally separate trace the positive power goes from the input capacitor C1 to the regulator's input.

The negative side of the power input goes to C1 then from C1 to the ground reference of the regulator. Thence to the negative lead of C2 and then through R5, a zero ohm jumper to the analog ground plane.

Then from the positive output of the regulator the current flows to the + lead of C2, the plus lead of C2 being the source of clean analog power.

R1 and in the unregulated version, R6 , may be one of several types of parts. The writer often uses a 1 ohm 1/4W resistor to serve as a lower cost fuse in many products. However this part could also be an inductor or a polyswitch type protection device.

So called “dirty power” is taken at or before C1 and has a separate connection to general purpose or “dirty ground” through R3. One can see how the LED operates from dirty power and ground, sharing no current going to or from the clean power current loop other than through C1. All circuitry that does not carry an analog signal needing to be very clean should use the dirty power and ground including digital circuitry. Higher power analog signals such as the output stage of a headphone amplifier should also have a separate ground connection. Remember, “ground” is NOT a sewer where you can dump anything and not worry about it. Ground is a very leaky sewer and unless you are careful stuff can leak out of it.

How important is it to route power and ground correctly? Sometimes not at all. Sometimes the distortion and noise of a product is a little high but barely worth another board spin. Most of the time no one will complain until you maybe fail FCC testing. The writer is constantly complimented how quiet their designs are. How much air the clearness there is in the sound. Basically, doing this properly only takes a small bit more time when designing the board and nothing per unit. Even just in case, why not make the best possible product one can if the cost is the same.

## **OLD PRO TRICK**

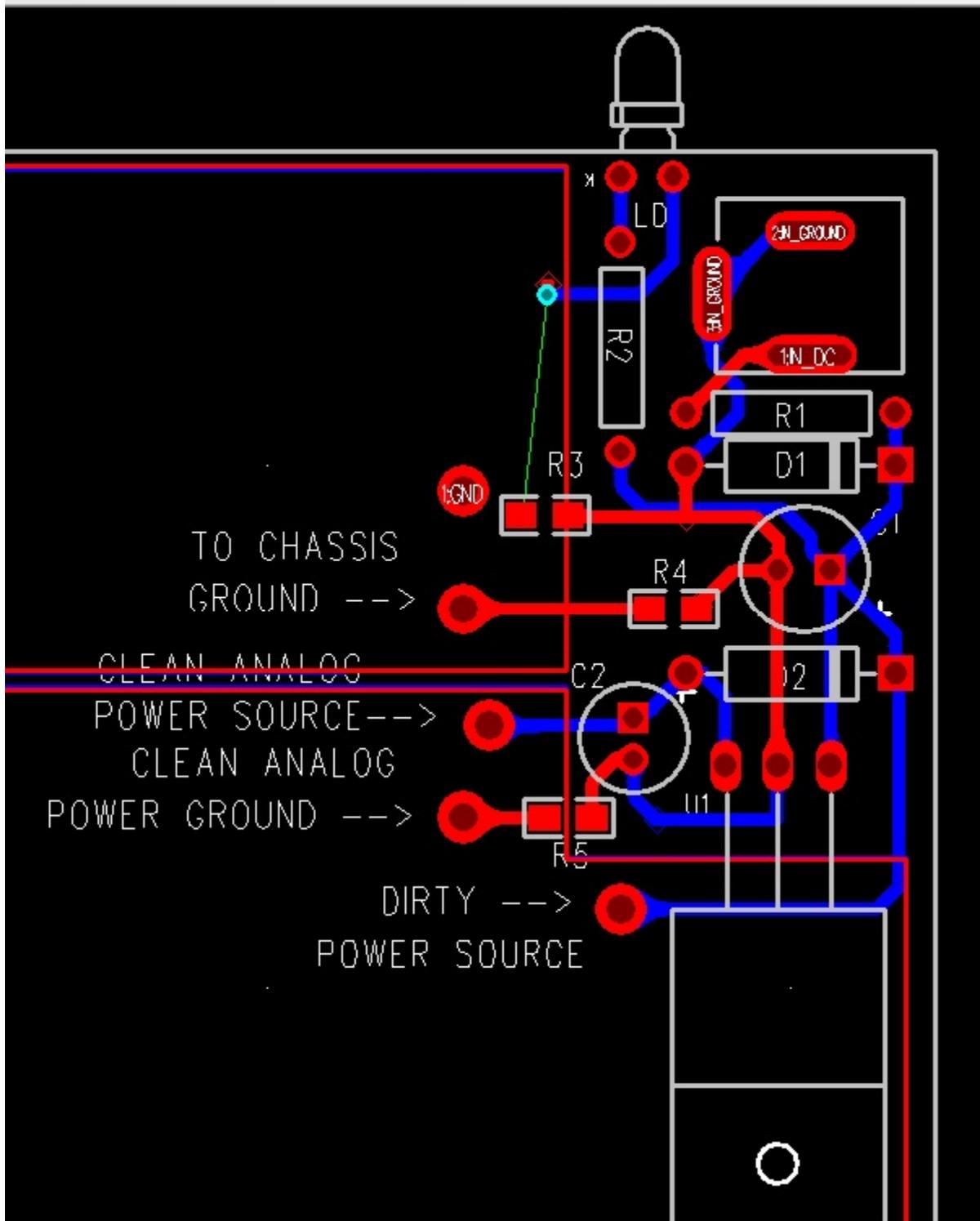
The old pro’s trick to minimize noise and EMI from a DC power input is:

**Every DC input power filtering capacitor has to have a separate set of input and output traces. There must be at least 2 traces to each pad of the capacitor to separate the input and output currents.**

Think of an oil filter. You need separate input and output pipes for the clean and dirty oil. Same for the power you put through a filter capacitor.

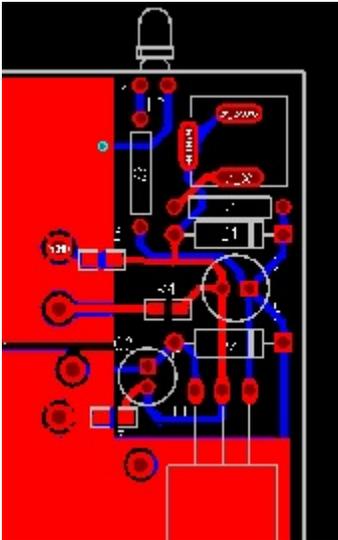
This also applies to the ungrounded lead of all power filter and bypass capacitor leads elsewhere in the circuit where the lead is not connected to a ground plane.

To illustrate the layout, in the following graphic, one can see the leads from the DC input jack go directly to the reverse polarity diode then direct to C15 before they go anywhere else.



The ground plane is cut away around the DC input to make sure that power comes only from after the input capacitor. The power for the input of the voltage regulator, pin 1, comes directly then from the + pin of C15.

The ground for the regulator is connected to the ground plane. The ground for C15 is routed to the ground plane with a heavy low impedance conductor.



The output, pin 3 of the regulator goes to the + pin of C14 before it goes elsewhere. With the – lead of C14 having a low impedance return to the – lead of C15 as well. Where C15 and C14 have their minus leads connected is the ground reference point for the design. This would be considered the “star ground” point in this design.

Chassis ground would be routed there as well.

A power ground to operate LEDs, relays or other high power devices would be connected by a zero ohm jumper directly to either the negative lead of C15 or even to the ground lead of the jack, in this case, pin 1 of the DC jack so that share no conductors at all with the audio ground.

If this sounds confusing, remember that you have to visualize the current flow of the power wiring for each point. The current flow of the DC power source must come from the supply directly to the input capacitor. The positive (+) input of the power adapter must go to the + lead of the input capacitor and no place else. Likewise, the negative (-) input of the adapter must go to the – lead of the input capacitor and no place else. Neither can connect to a ground plane.

Do not think of the Power Connector as the input point. The connector is merely a bump in the power lead in wire. The actual power input is the input capacitor.

## OLD PRO TRICK

The old pro’s trick to isolate different grounds to minimize noise and EMI

Give every different ground type a different name and combine them only where they will not share any return current using zero ohm jumpers

This is necessary as most PCB design software will not allow one ground to be connected to another with a different name. When you connect them, they all get the same name. Thus the PCB and schematic files will no longer match. Resulting in errors if someone else has to update the design. If you lay down unassigned copper on the boards to connect these grounds, the Dynamic Rules Checking will flag a clearance error and you should never run gerbers for a PCB if there are any unresolved errors.

But who wants to explain to accounting why there are all these zero ohm jumpers that appear to serve no purpose.

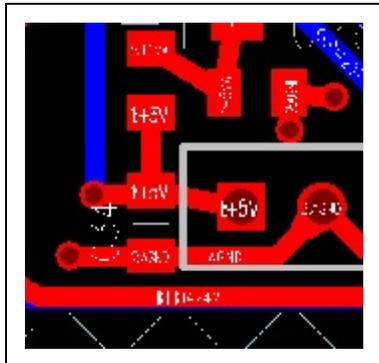
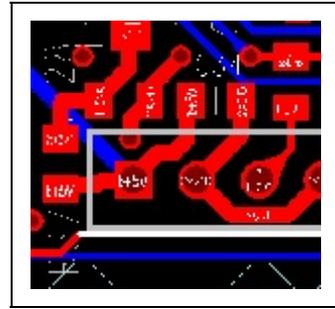
In another part of this document you can find how to create free zero ohm jumpers that can be used to connect conductors with different names without causing rules violations to show. Look for the section called “ Zero Ohm Resistors”

## Bypass Capacitors on ICs

A bypass or filter capacitor is generally any capacitor on the board that has one lead connected to circuit ground or common. On the “hot” or ungrounded pad, they must be routed with at least two conductors going to the pad. A filter or bypass capacitor with only one trace going to it is essentially useless. In most cases there would be 2 traces. One input trace coming from the power source and one trace (or more) going to loads.

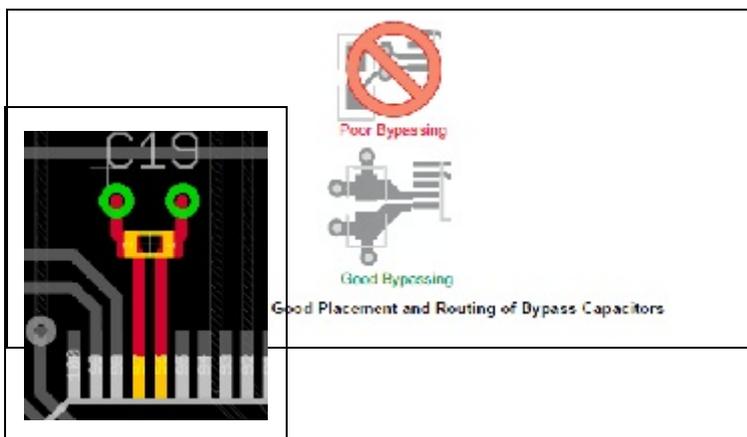
A filter or bypass capacitor is like the gasoline filter on a car. You do not run unfiltered gas and filtered gas in and out of the same pipe. The pipes are both one way and separate input and output pipes are provided in a gas filter. The power is the fuel for your circuit and you need separate inlet and outlet conductors for the power the same way that a gas filter on a car has separate inlet and outlet pipes.

C34 in the graphic to the right is an example of a filter capacitor wired incorrectly. With only one trace going to it, it cannot do its job very well and is essentially useless.

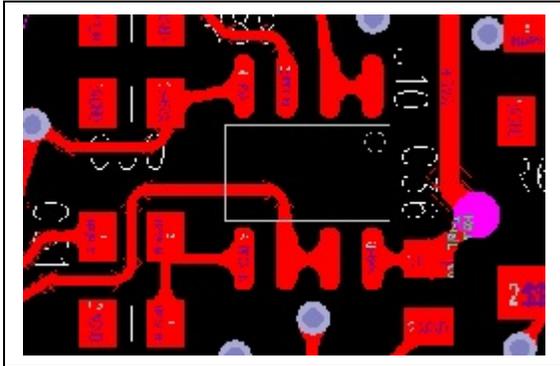


In the graphic to the left C34 has been wired correctly. On the hot side the +5V input goes to the capacitor then to elsewhere in the circuit. Similarly the ground lead goes to the ground lead of C34 before going to a pad to the ground plane.

This sequencing of power is rarely made clear on the schematic. Connections to filters is more commonly showed as a part with a single lead going to the part. The board designer must take care to place the bypass capacitor correctly. Bypass and filter capacitors are identified as in most cases, one of the pads connects to ground. All IC power pins must have a bypass capacitor as close to the power pins as possible. The only exception is that if there are multiple adjoining power pins that may share a capacitor. The power from the power source must ALWAYS go to the capacitor before going to the IC. The power must never be directly connected to the IC. The ground side of the bypass capacitor must be connected to whichever ground is required for that IC. They are never to be connected to the Chassis ground.



This is a good way to do a bypass in a big chip using a 0402 capacitor with the capacitor forming a low pass filter between the power and the IC. The capacitor must be as close as practically possible to the IC.



In this graphic from the Decoder Ring one can see the bypass capacitors on pins 4 and 8 of the IC. The positive power trace +6VA goes to the capacitor C36 then to pin 8 of the IC. Likewise negative power trace -6VA goes to C50 then to pin 4 of the IC.

Power must never go to the IC then to the capacitor.

On larger ICs, the manufacturer's spec sheet may have bypassing recommendations and these should be closely followed.

On multi-layer boards power and ground planes should be adjoining layers so that the copper fill of those layers will themselves serve as a decoupling capacitor. If components are on one side only, the ground layer should be next to the component layer. That is, when parts are on layer 1, the ground plans should be layer 2 and the power layer would be layer 3.

When an SMT bypass capacitor connects to a different layer through a via, the via should be jammed up to the bypass capacitor's pads to make the trace as short as possible in order to minimize series inductance. The vias holes to a ground or power plane need to be as large as practical to reduce series inductance.

Bulk capacitance of larger value capacitors should be distributed on the board but is not required for every IC.

#### References:

[http://www.interfacebus.com/Design\\_Capacitors.html](http://www.interfacebus.com/Design_Capacitors.html)

<http://www.sigcon.com/Pubs/pubsKeyword.htm#bypass%20capacitors>

<http://www.sigcon.com/Pubs/edn/thewayhome.htm>

[http://www.sigcon.com/Pubs/news/2\\_3.htm](http://www.sigcon.com/Pubs/news/2_3.htm)

<http://learnemc.com/decoupling-for-boards-without-power-planes>

<http://learnemc.com/decoupling-for-boards-with-closely-spaces-power-planes>

<http://www.designers-guide.org/Design/bypassing.pdf>

<http://www.intersil.com/content/dam/Intersil/documents/an13/an1325.pdf>

<http://www.ultracad.com/mentor/esr%20and%20bypass%20caps.pdf>

## Power supply filter capacitor wiring.

These are wired as bypass capacitors discussed in the previous section. Traces must always go from the power source, to the capacitor then to the load. This includes both sides of the capacitor. Ground planes may have to have cutouts so that the input ground goes to the capacitor first before connection to the ground plane.

A filter capacitor with only one trace going to either lead is not able to do its job properly. In the lead going into and out of the capacitor, DC current can only flow one way. Which requires separate input and output traces.

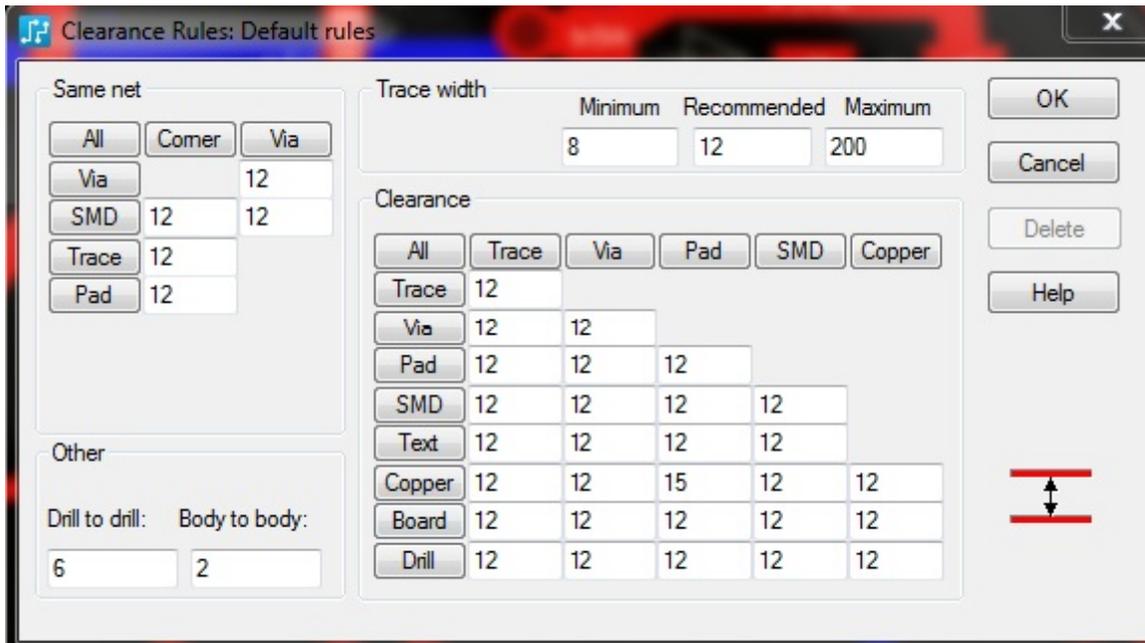
This and the previous section are especially critical on low noise circuits, circuits with any radio frequency parts, any sort of oscillator, a power converter or any digital circuitry.

## Trace Width

While 12 mil wide traces are common for signal lines and as low as 9 or 10 mils is common for dense designs. Traces carrying even the minimum amount of power to circuit elements should be at least 25 mils wide. This includes the output traces of headphone amplifier circuits.

## Trace Clearance

Normal clearance is set to 12 mils for everything. The graphic below shows However in some cases on SMT boards this maybe set lower. Traces carrying between 35 and 60 volts such as any trace ca



## Teardrops

Teardrops should be used as much as possible where traces meet pads to strengthen these points. This makes the board somewhat less likely to be damaged if rework or repair is needed. If a teardrop causes an issue in a very small SMT part, by putting a break in the trace inside the pad, formation of a teardrop will be inhibited.

In the Options Menu in the Routing\General page, the “Generate Teardrops” option should be checked and in the Routing\Teardrops page “Display Teardrops” and “Auto Adjust” should be checked.

If a tear drop refuses to form it is generally because of a break in the trace within or near the pad. The procedure to fix this is described in the MITER section.

Also note that some pad shaped like squares will not form teardrops at all.

### RESEQUENCING INSTANCE NUMBERS

After the PCB has the instance or reference numbers resequenced, they have to be imported back to the PCB. It is suggested that after this is done one checks to check one of the text files that pops up that no traces or components have been deleted.

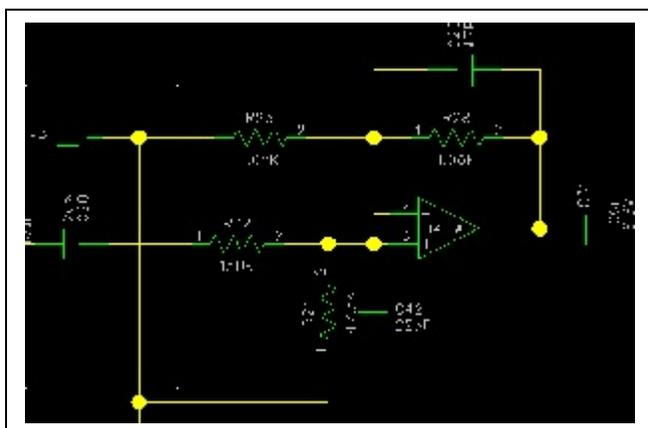
## Trace Naming

It is suggested that major traces receive specific names. This can make board layout issues easier to troubleshoot. In PADS Logic in the Options select Design. Then make sure the box for “Allow Named Subnets Without Labels” is checked.

When in the PCB layout, enter “NNT” and press Enter to toggle trace names on and off.

NNP turns on pin names.

## Routing Op-Amps



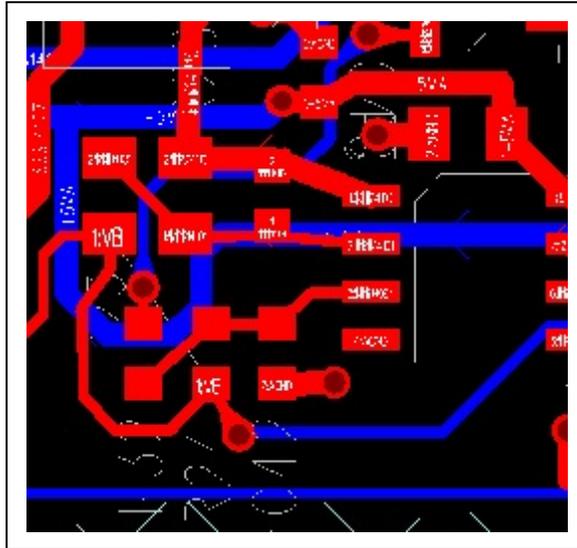
In routing an opamp the traces on the + and – input leads are critical as the feedback components.

### FEEDBACK COMPONENTS

The feedback components in this case are C25 and R28. These must be as close to the op-amp as possible. In most audio it does not matter whether the resistor or capacitor is closer to

the op-amp. However in circuits with noise like RF circuits, oscillators, etc, many feel the capacitor should be next to the

In the graphic the 0603 package is next to pins 1 and 2 of the op-amp. is in parallel with that. Finally R23, is to the common (VB). While the trace to the – or inverting input of an op-amp considered a “virtual earth” connection, has a high input impedance determined values of the gain setting resistors (R23 and can be as susceptible to RF pickup other trace.



generators engineers op-amp.

capacitor Then R28 connected connected i s in reality it by the and R28( as a n y

Similarly the trace connected to the + or inverting pin must also be short as possible. The bypass capacitor to ground (C42) may not be there in all cases but if it is used, it must be as close to the input pin as possible. As well the input bias resistor (R24) must also be as close as possible to the input to reduce the impedance of the trace as close to the op-amp as possible. The resistor R32 in conjunction with C42 forms a low pass filter. The connection between this resistor and the capacitor must be as short as possible. If the connection to the op-amp has to come from a long distance, then that long trace would be on either side of C20.

n o n -

R32 serves a secondary function where if the previous stage is an op-amp, and R32 was zero ohms, the capacitor C42 may make that op-amp oscillate. Having R32 causes the frequency pole created by C42 to change to a point where the preceding stage will not oscillate. In cases where the signal through C20 comes from an input jack R32 acts to limit the possibility of static discharge damage to the op-amp.

Also note on the graphic how the power goes to the capacitor then to Pin 8, the power pin of the IC.

## Clock Lines

In circuits with digital to analog or analog to digital converters and CODECs there are several critical traces.

- MCLK
- SCLK (aka BICK)
- LRCK (aka LRCLK or Word Clock)
- SDIN (aka SDATA)
- SDOUT (aka SDATA)

Only units with CODECs will use both SD traces. These traces carry digital signals up to 24.576 MHz. The signals have leading and/or trailing edges that must arrive at the converter/CODEC at the same time within about 5 nano seconds. That is 1/200<sup>th</sup> of 1 millionth of a second.

These traces must be of the same length as much as possible within a centimeter or so. They should also run parallel with each other as much as possible and have the same number of vias. And as few vias as possible.

The trace length function in the CAD software should be used to make sure these are as close to each other in length as possible. As well, the longer the overall length of these, the more critical that length matching becomes. A “long” length would be over 125mm or 5 inches.

As well in AES receivers, Ethernet lines, USB data lines or other digital lines with balanced signals it is also important to keep those running side by side and to have the same length as much as possible.

## ***Component Spacing***

### External Parts

Consistent spacing of external components is important to achieve a consistent look. External parts are defined as those that require coordination with the external housing. Usually mounting holes in the case and seen by the user. However they also include mounting pads. The final determination for the placement of these parts is determined by the designer of the external enclosure in consultation with the design engineer. However the engineer can determine the required sequence of the external components in consultation with the requirements of marketing.

All the dimensions listed are in mils. 1 mil = 0.001 inch. The spacings given are center to center on all components. If wider spacings are used, they must be consistent on a product.

Push Switch to Push Switch = 350

Push Switch to 16mm Pot = 550. These pots are also known as RV122 or R800 9012 or R800 9036 series.

16mm Pot to ¼” jack = 750

16mmPot to 16mm Pot = 700

¼” jack to ¼” jack = 900 to 1000. 800 can be tolerated but would mean that the larger plugs may not be able to be used.

¼” Jack to XLR connector = 1000

Push Switch to XLR = 675

XLR Power Jack (4 or 5 pin) to ¼” jack = 875

LEDs mounted 90 degrees under a push switch are normally mounted at the same position as the push switch.

Depth behind board edge

Push Switch = 555 or 590. The lower dimension is where a button goes through the panel. The larger dimension is where the switch is recessed and there is no button.

¼” Jack = 355

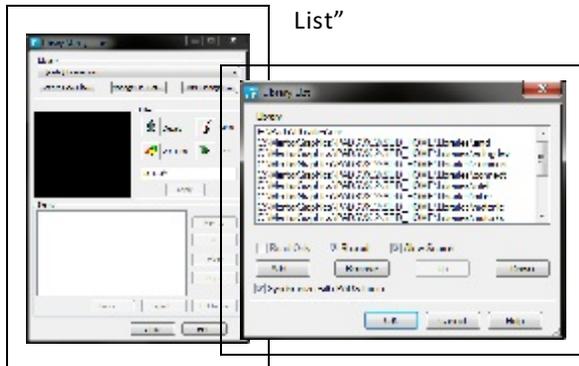
LED – Mounted 90 degrees = 100

Tab Route for LED = -15

## Libraries

The common library is in the Engineering folder on the server. The writer has the engineering directory on the server mapped as drive "E:\". Thus the common corporate library is at E:\PADs\Libraries\

The library the writer uses for parts is the one called "usr". The library manager may be used to make this your default directory. Open it by opening the File menu and selecting Library. Select "Manage Lib



Use the Add command to find the "usr" library on the server. Then use the Up button to move it to the top position making it the default position.

The library manager also allows one to easily scan libraries looking for specific parts. As well, if one creates a part and feels they have made an error with it, the Library manager is the only way to remove an unwanted part. One can select either Decals or Parts and use the dialog box right under there to enter wild card parameters to limit the number of items selected.

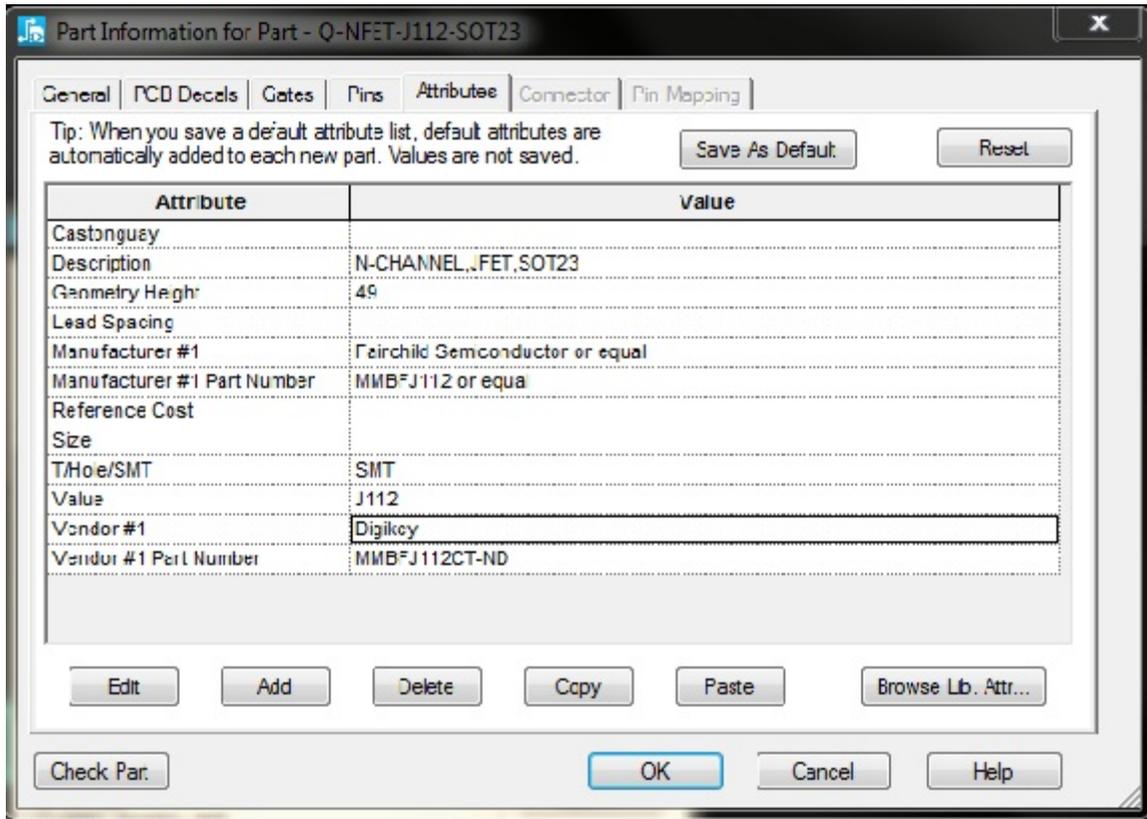
Most parts have a prefix to the part number so that like parts are grouped with like. After that will be the part type or value then voltage if relevant and sometimes the case type.

- Type - PREFIX
- Transistors - QNPN or QPNP in most cases. However sometimes users have created these under the actual part numbers as well.
- FETS – QFET, Q-NFET, NFET
- Capacitors - CRE, CSE, CRC, CSC or CRM. R meaning radial and S meaning SMT. E = Electrolytic, the second C = Ceramic and M = Metal Film or Film. CAC is an axial ceramic capacitor. After the value will be the voltage and the case size or lead spacing.

- ICs - N- in most cases. They may also be suffixed by the case style. However at times some users have created ICs without using the prefix "N-".
- Resistors - RSM or RM. RM meaning radial metal film and RSM meaning SMT. 1% metal film parts are the preferred type. Carbon film resistors are RF types for carbon film and for wire wound are RW
- Resistors, Vertical Mount - RVM
- Potentiometers - RPT.
- Trim Pots – RTRIM, VRES or VR
- Switches – S or SW
- Blocks for block diagrams - BLOCK
- Connectors and Jacks – J, CON, HEADER or SIP
- XLR Connectors – J or XLR
- Diodes – D or DIODE
- Zener Diodes – D or DZ
- Bridge rectifiers – BRIDGE or D
- Fuse – F or FUSE
- Ferrite Bead – FB
- Mounting Hole – MT or MTH
- Pad – PAD
- Fiducial – Fiducial
- Inductors – IND
- LEDs – LED
- Opto Couplers – OPTO
- Relays – RLY
- Transformers – XFMR
- Crystals - XTAL

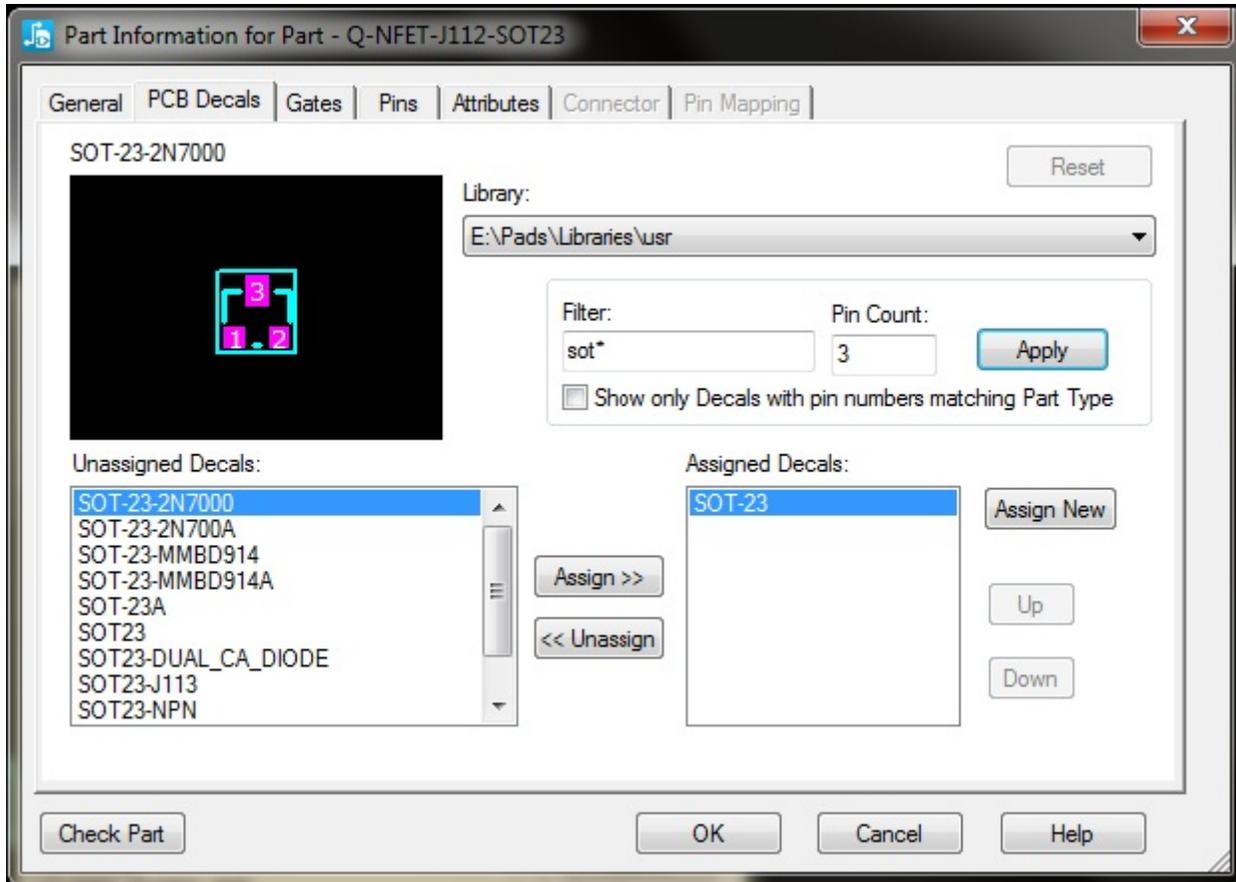
There are quite a few parts that show up saying "No Decal" These are usually parts that were imported from a board created in Protel and imported into PADS

When parts are created this graphic shows the most commonly used fields in the Attributes tab of the Parts Information box.



Geometry.Height is usually in mils.

In the PCB decals tab make sure one and only one decal is assigned. If there are more than one in the Assigned Decals box errors will happen.



Double check that all pins are correct in your PCB decal. SOT23, LED decals and 8 Pin IC decals have sometimes had issues due to some bad decals in the libraries supplied by Mentor Graphics. If a part is imported from a different library this has been an issue in the past.

Never export parts from a schematic to be saved in the library to overwrite an existing part unless you are really, really sure the part in the library already has issues.

## **Text**

For decal text, the line width is always 0.1 or less than the text size. That is, if the character size is 100 mils, the line width should be 10. If the line width is greater than 10% of the height, the characters may be blurry. In the cleanup process, check the board carefully to see that none of the characters look “fat” (like graffiti letters). Recommended reference designator height on 0805 and smaller parts would be 50 mils (1.27mm) height and 5 mils (0.127mm) line width. Otherwise 66, 70 and 100 mils character heights are commonly used.