

PRACTICAL ELECTRONIC PROJECT FABRICATION



By

Jim Weir

RST Engineering

&

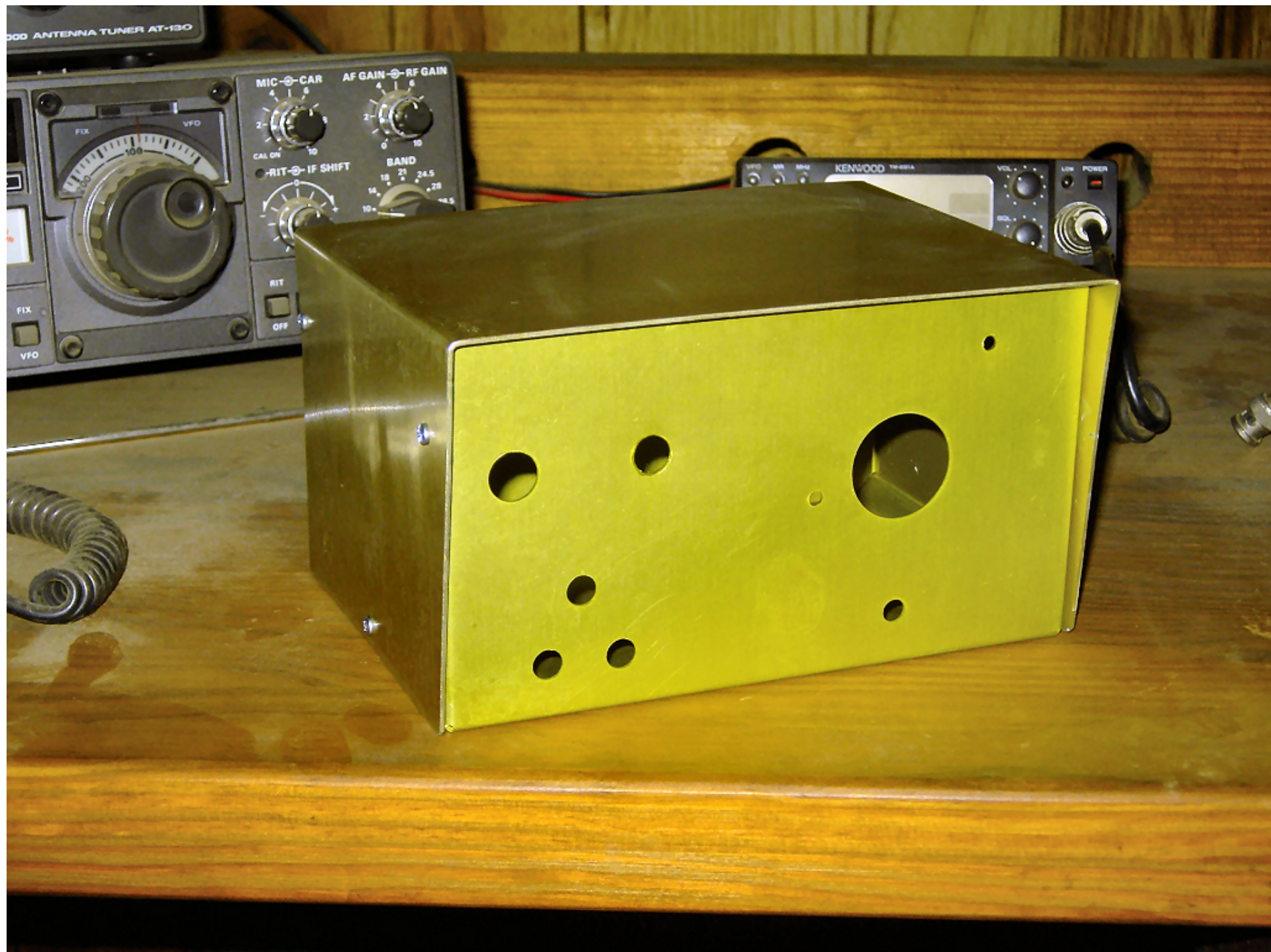
Sierra Community College

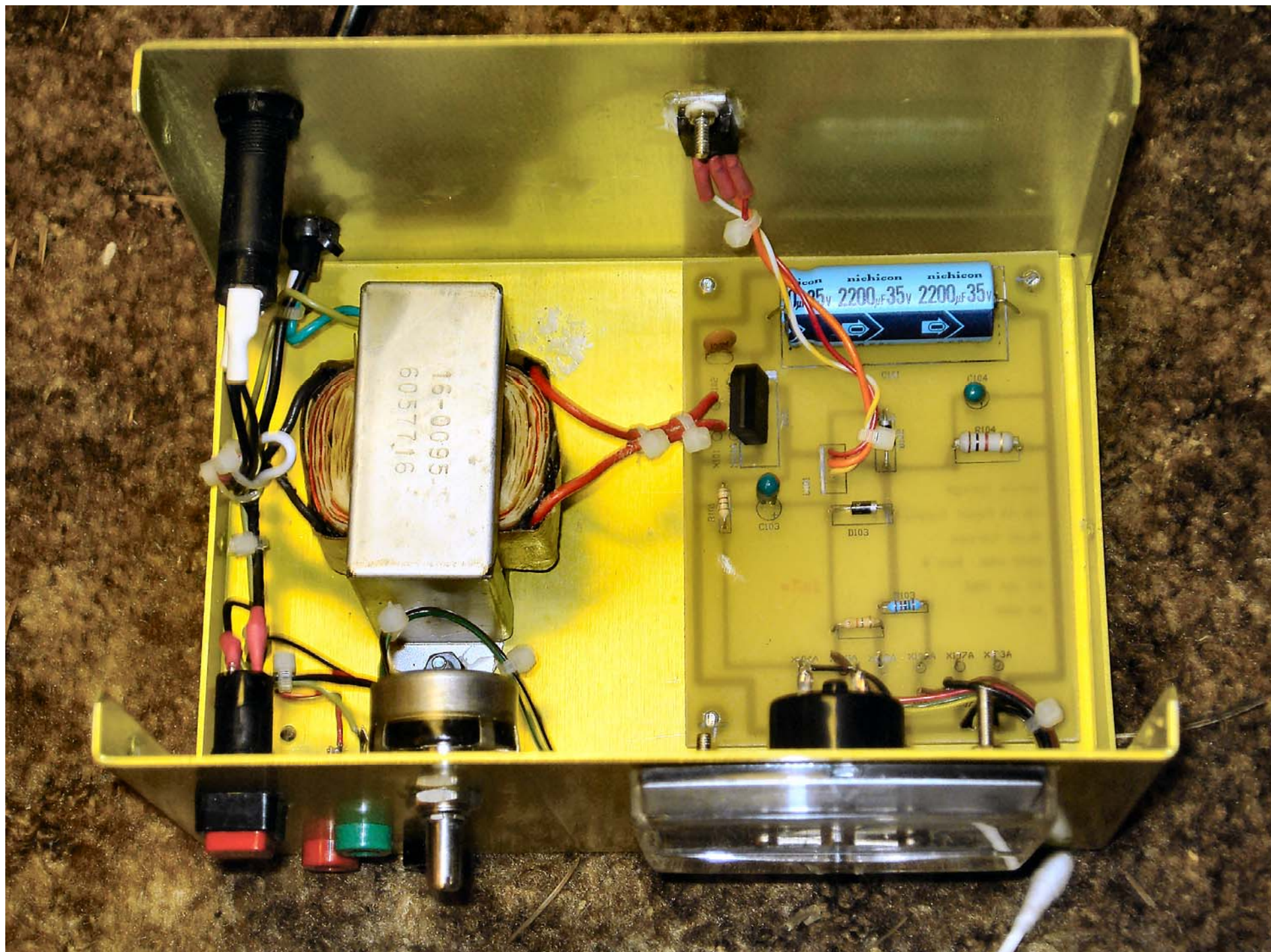
Dedicated to my ex-wife, who made this book financially mandatory.

Introduction:

- A. This book is intended for the shop technician or design lab technician who wants a plain language beginning course in electronic project fabrication from raw material.
- B. The book is divided into three main sections ... tools, metals, and pc boards.
- C. The book is written so that common machine shop tools and simple drug-store chemistry are the only required materials other than the aluminum for the chassis and the raw pc board material.
- D. The book is project-oriented. The focus of this book is the fabrication of a simple power supply. However, the principles of construction and assembly will be applicable to a wide range of projects.







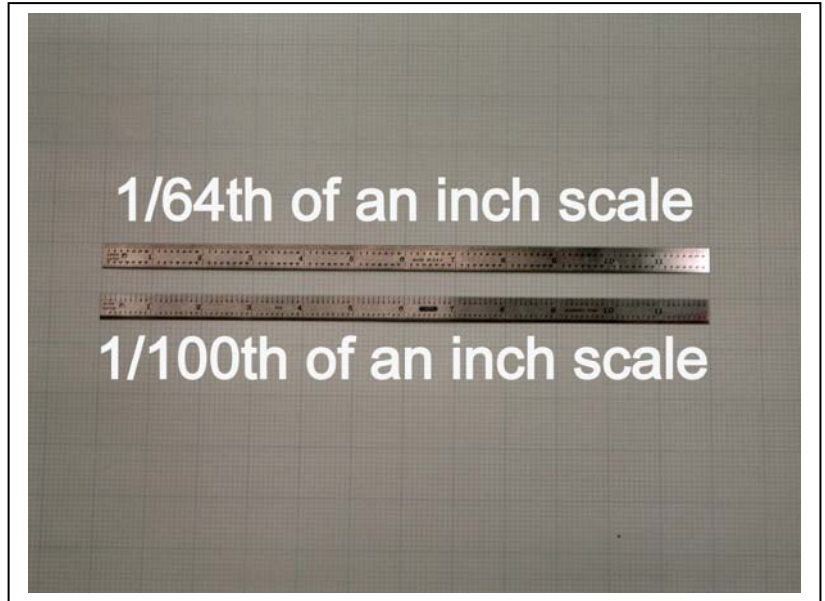


II. METAL FORMING TOOLS

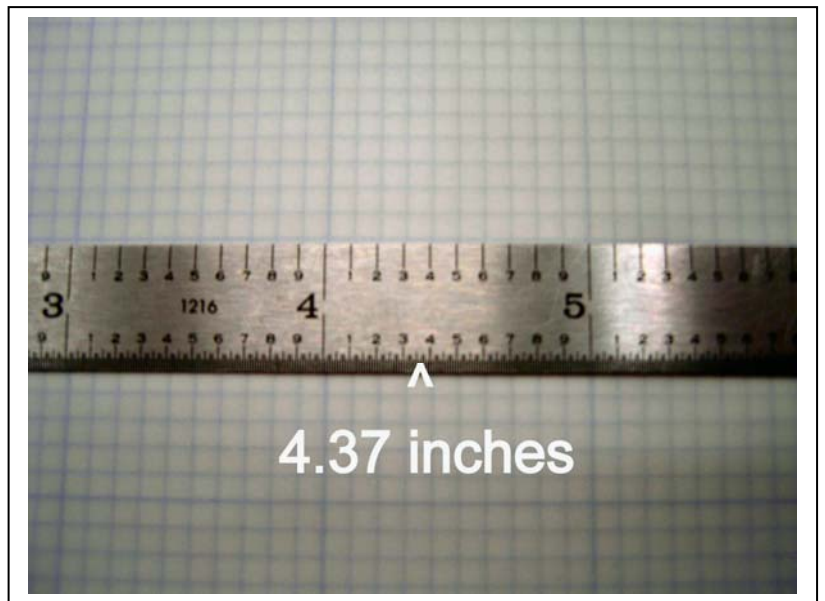
1. Rulers.

- a. Rulers come in all sizes and materials. Short little 6 inch pocket rulers and long 100 foot flexible tape rulers. Rulers come made of plastic, wood, metal, and other materials. However, for the purposes of this course, we will use STEEL rulers with lengths between 6 inches and 18 inches. The standard ruler for this course, unless otherwise specified will be a stainless steel ruler 12 inches long.

- b. The class standard ruler has two sides. One side measures in $\frac{1}{64}$ ths of an inch. The other side measures in $\frac{1}{100}$ ths of an inch. We will use the "metric" english $\frac{1}{100}$ ths of an inch side in this class. Why? Well, tell me quickly how much $3\frac{9}{64}$ ths of an inch added to $4\frac{3}{16}$ ths of an inch is? Now tell me what 3.14 plus 4.18 inches add up to. Much easier to use "metric" inches, isn't it?

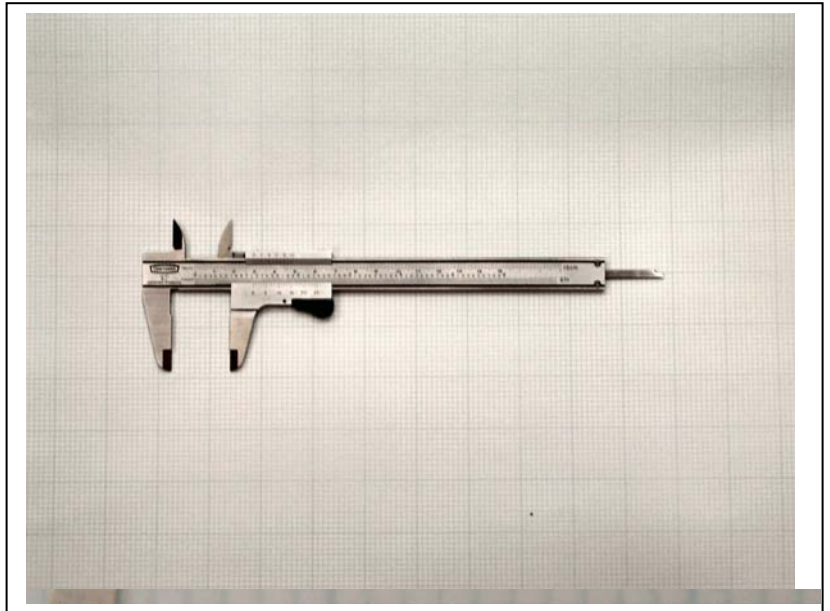


- c. Notice in the photo to the right that the top scale is divided into inches and tenths of inches, while the bottom is divided into inches and hundredths of inches. When we are not too fussy about how long something is to be made, it is far easier to use the tenth-inches side, but if we want something rather accurate, we use the hundredths of an inch side of the ruler. Note that with a ruler like this, the closest you can hope to get something is out in the second decimal place. We call this "ten mil" accuracy because the best you can measure is to ten thousandths of an inch (a "mil" is a thousandth of an inch).



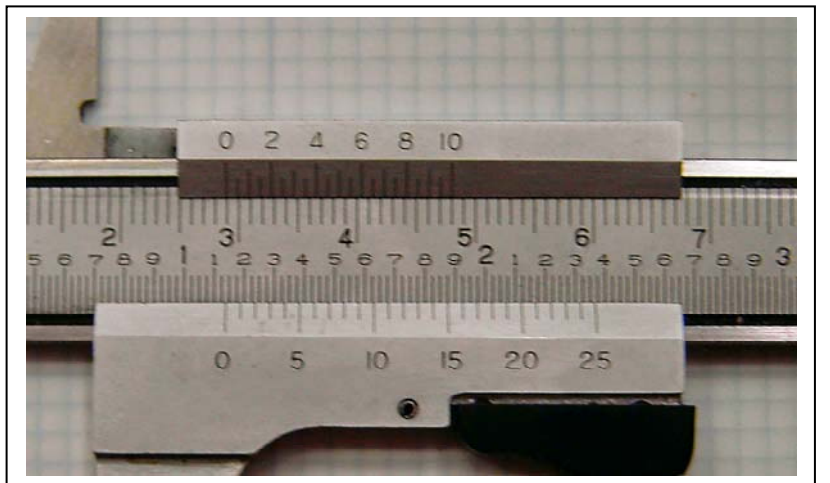
2. Vernier Calipers

- a. When we try for better accuracy than a steel ruler can give us, we turn first to the old tried and true analog vernier caliper. The vernier caliper is capable of giving us accuracy to one-thousandth of an inch (one mil) if properly used.



- b. "Properly used" in the preceding paragraph means that the caliper hasn't been used as a hammer or a pry-bar by a previous class so that it's basic accuracy hasn't been compromised. A very accurate "gage block" may be used if there is question about the basic accuracy of the instrument. In addition, you have to be able to properly "interpolate" the last digit for accuracy down to the 1 mil level. A clever arrangement of lines on the scale will allow this interpolation.

- c. Note in the photo to the right on the bottom scale, the 0 (zero) line is somewhere between 1.1 and 1.2 inches. Note also that there are four equally spaced small lines between 1.1 and 1.2 inches. Thus, carving the tenth of an inch up between .1 and .2 inches means each of the small lines is 0.025 of an inch. Note that the 0 line is about halfway between the first and second small line. We



therefore might guess that the caliper is set somewhere around 1.135 to 1.140 of an inch. (halfway between 1.125 and 1.150 of an inch). To get that last set of digits down to a mil, note that the bottom scale goes from 0 to 25. Note also that somewhere around 14 on the bottom scale the lines on the bottom scale line up with the lines just above them. We can then say with some confidence that the calipers are measuring $(1.125 + 0.014)$ 1.139 inches.

- d. Digital wizardry being what it is, we can eliminate the confusion of this interpolation stuff by means of a digital readout. There is no doubt in my mind that this small aluminum bracket is 1.498 inches wide by 3.996 inches long.



- e. Note the controls on the digital caliper in the photo to the right. The red button turns the power on and off (there is an auto-off if the reading doesn't change for a couple of minutes) . The yellow button allows you to zero the reading when the "jaws" are closed. The blue button on the top switches between inches and millimeters. The small round knob to the left of the black readout housing is a roller to move the jaws back and forth. The little knurled knob on the top fastens the jaws in one spot and keeps them from moving.



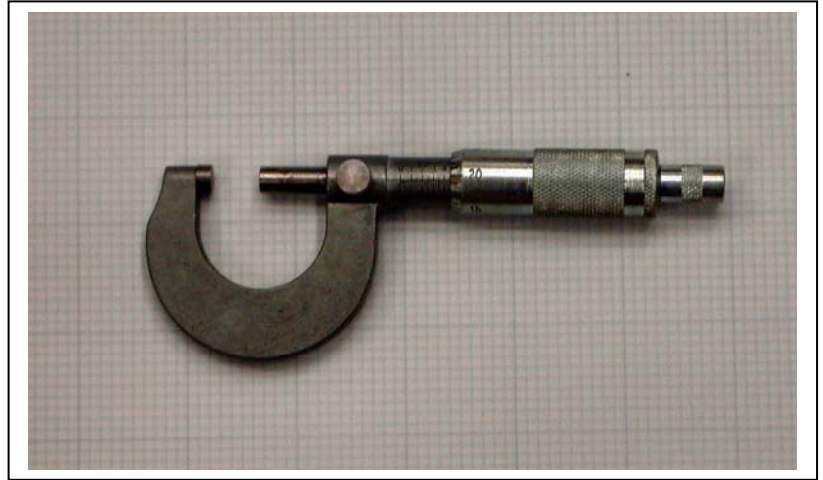
- f. In the photo to the right, you see how ANY caliper or micrometer should be put away. Note that there is an 11 mil gap between the jaws. When you put away a caliper/micrometer, there should be somewhere between ten and fifty mils of gap between the jaws. That way, should the instrument be dropped, the jaws will not "bounce" off one another, bending the jaws, and causing inaccurate readings.



3. Micrometers.

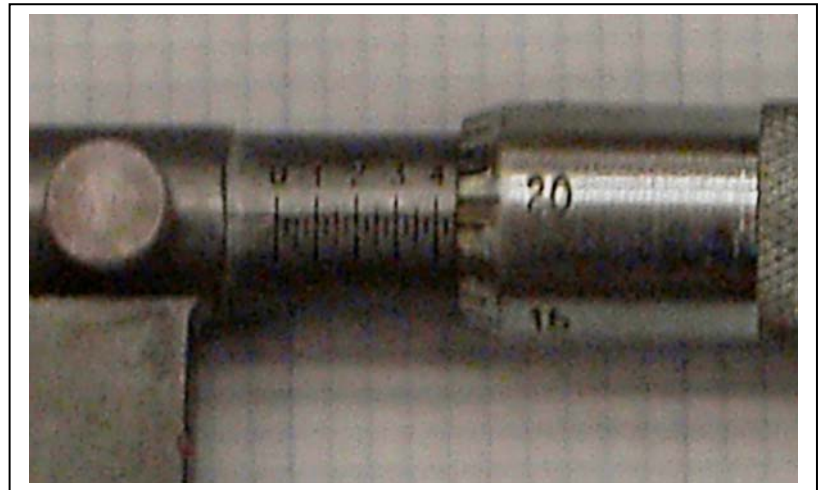
- a. A micrometer is nothing more than a caliper that has a rotary thread rather than a straight body. By rotating the thread we can measure the gap between the jaws of the micrometer.

- b. The principal parts of a micrometer, on the right, are the frame (c-shaped large part on the left), the barrel (large round part in the center of the picture that doesn't rotate), the thimble (large round part towards the right side that does rotate), the spindle (small round part protruding out the right side of the frame) and the anvil (small little nubbin on the left side of the frame opposite the spindle). The anvil and the spindle together are sometimes called the jaws of the micrometer.



- c. Also on the far right side of the picture is the ratchet. The ratchet is used so that the same force is always applied to the part being measured -- it is quite easy to force a micrometer a few thousandths of an inch by applying too much force to the thimble, and the ratchet prevents this.

- d. In a similar manner to the caliper above, note that the fixed numbers on the barrel are divided into tenths of an inch, and then each tenth of an inch is divided up into four minor divisions. Each of the minor divisions, then is 0.025 of an inch. Note in the micrometer on the left that the jaws are between 0.4 and 0.5 divisions. Note also that the edge of the thimble is past the first small division, so we know the



reading is going to be at least 0.425 inches. Note that the straight line running horizontally down the barrel intercepts the thimble at one division short of 20, or 19. Adding 0.019 to 0.425 gives us a reading of 0.444 inches for the distance between the jaws.

EXERCISE: There are eight small pieces of metal numbered 1 - 8. Use measuring tools to measure both the length and the width of each of the pieces.

Metal #	Micrometer Thickness (one only)	Ruler Length	Ruler Width	Vernier caliper length	Vernier caliper width	Digital caliper length	Digital caliper width
1							
2							
3							
4							
5							
6							
7							
8							

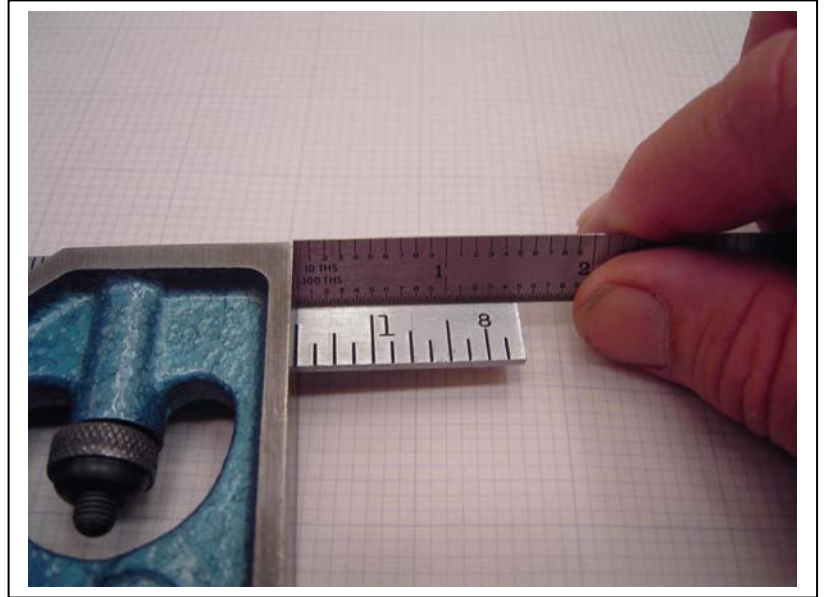
4. Squares

- a. The "combination" square is adjustable in length and gives fairly precise 90° and 45° angle guides. Once again, if a prior class used the combination square as a hammer or a bending bar, the accuracy of the square is not guaranteed
- b. The combination square also includes a bubble level as part of the stock head and is SUPPOSED to have a small pin scribe for marking lines on metal. Most scribes that have been around for a while have long since found their way into a black hole in the universe from which they will never reappear.
- c. In the photos on the right, we see the combination square. It is comprised of a ruler which has been grooved down the longitudinal axis, a stock head with a knurled knob to fix the head firmly at a spot on the ruler, a bubble level on the left hand side near the bottom of the stock head, and the missing scribe which used to fit into a small hole on the bottom of the stock head.

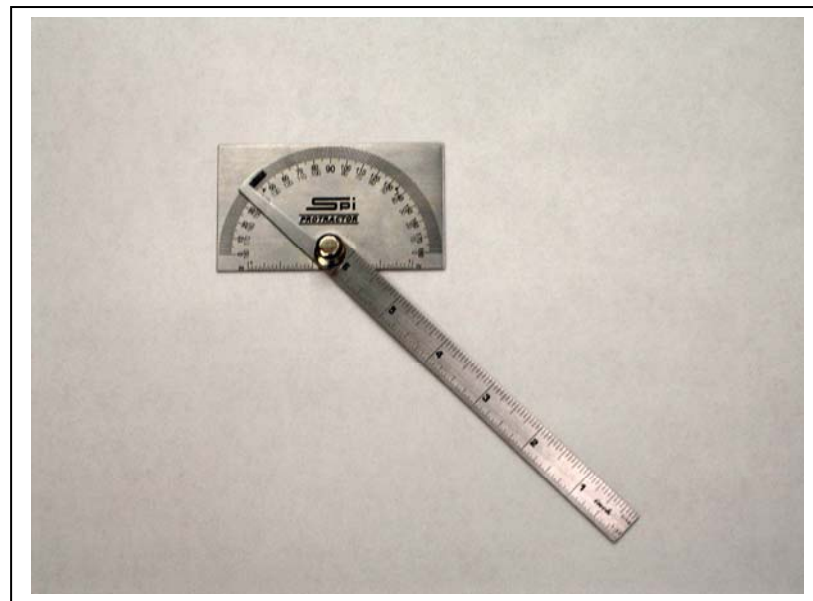


- d. In the photo on the above right, we can see the stock head set to somewhere around five inches. Note that there are sixteen divisions between four and five inches, so the combination square is out of line with our "metric inch" rule.

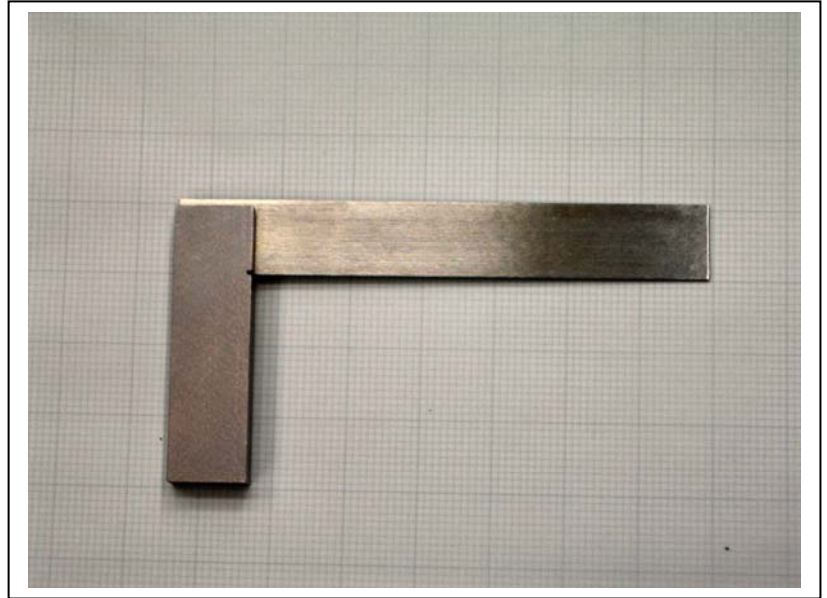
- e. The way around that is to set the square with the decimal inch steel ruler for the desired length. You would want to do that anyway, as the square lines are ten to twenty mils thick in and of themselves.



- f. There is a "square" whose angle may be varied to any angle you wish. This is called a variable angle square and is quite useful when setting angles for sloped chassis tops, odd cuts on a pc board, and other angles that are not 90° and 45°



- g. The last square in our trilogy is the precision square. This square is not used for layout or other uses that the combination square is capable of doing. This square is rather expensive and is precisely 90° . You use this square to check the other squares for "squareness". Again, it is an expensive piece of equipment and not to be used for anything other than checking other squares.

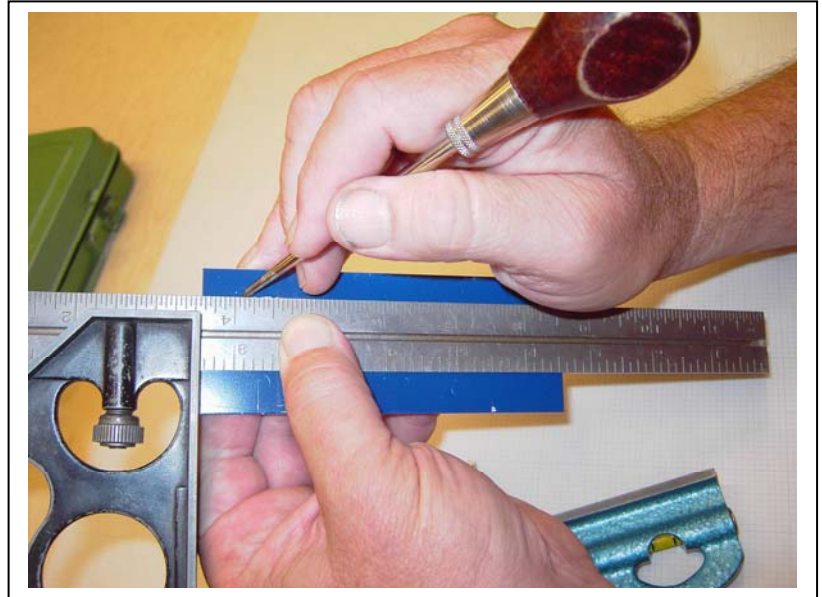


5. Scribes and Center Punches

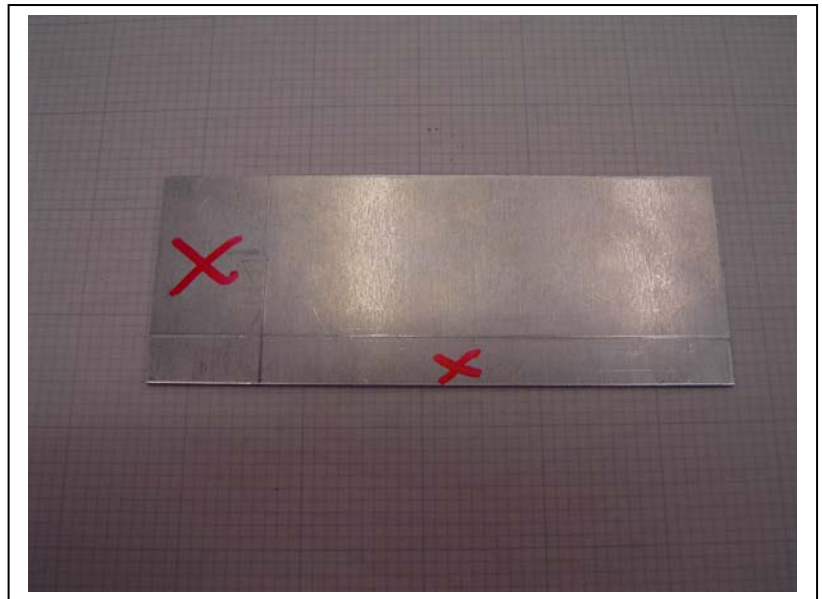
- a. A scribe is nothing more than a sharp pointed tool that will make a scratch line on metal or paint. The scribe shown here is nothing more than an ice pick that went to college. So long as it has a fairly sharp point on it, practically anything can be used as a scribe. (There have been some times in the field when I needed a scribe and didn't have a "real" one that I used a nail.)



- b. If you are doing very precise work and need to keep the surface of the metal as smooth as possible, you can use machinist's bluing on the metal, scribing only through the bluing and not on the aluminum itself. It takes a very steady hand and lots of experience to cut through the bluing without scratching the aluminum.



- c. For most purposes around the lab or the prototype shop, scratching the cut and drill lines directly onto the aluminum surface itself is the normal way of doing things. Here you see a 2 inch by 5 inch metal plate about to be machined down to the desired size of 1.5 x 4 inches. Note the red X marks. These show the part of the plate that will be scrap at the end of the process.



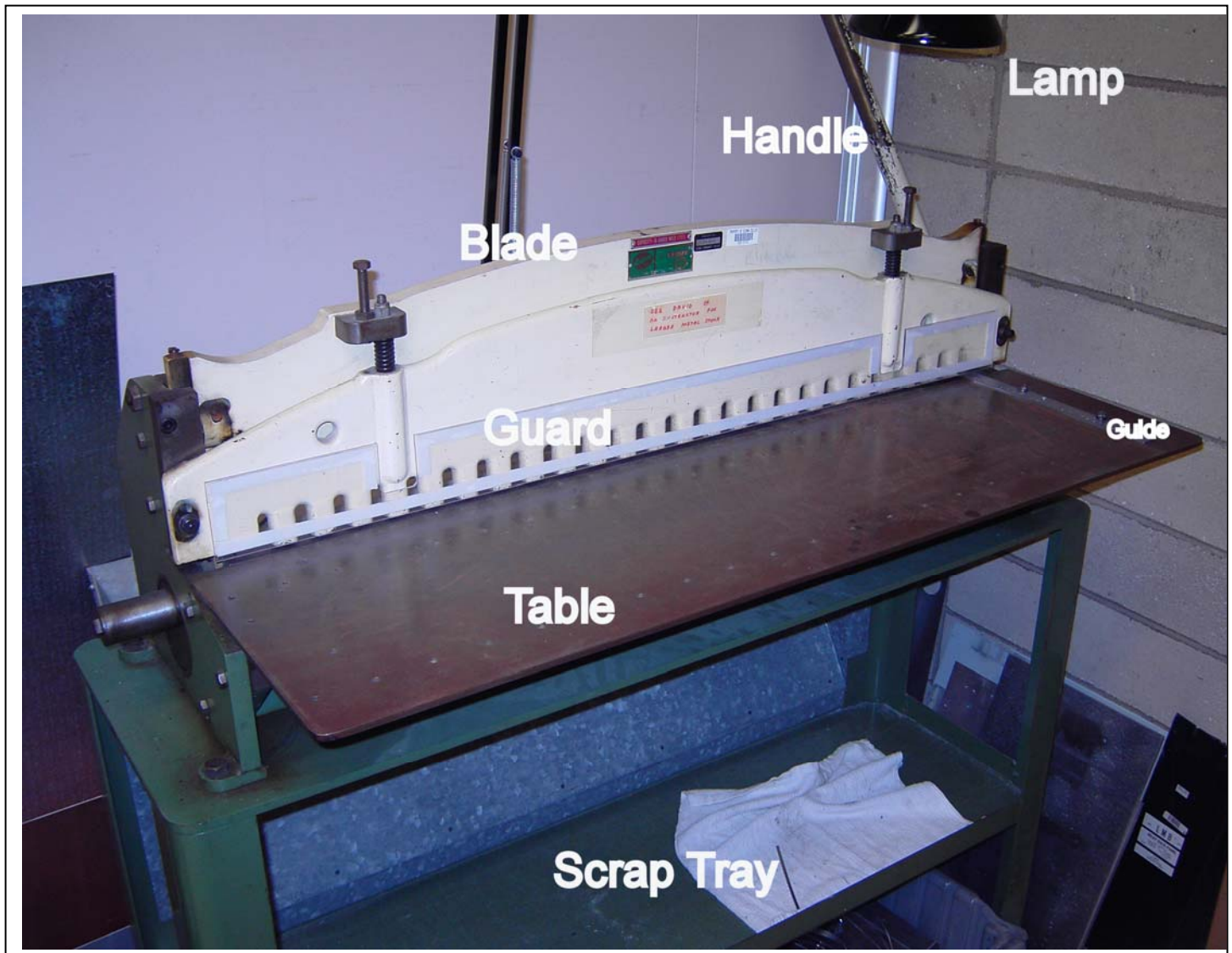
- d. A center punch is used to make a dimple in the metal at the exact center of a hole. Shown here is a manual punch that has the sharp point placed exactly where you want the center of the hole and the top (right) end of the punch struck with a hammer. The drill or punch then centers itself in the dimple and makes the hole at that point.



- e. When you've got a few dozen holes to center punch on a sheet of metal, the manual punch - hammer routine gets pretty tiring. A spring-loaded center punch makes that job a whole lot easier and is somewhat more accurate than the manual punch. Instead of a sharp blow from a hammer that can drive the manual punch a few mils off center, the automatic punch delivers a precise amount of force directly on target.



6. Shear



- The shear is the first of the "large" tools that we will discuss. The various parts of the shear are shown above.
- Once a semester, generally at the beginning, the shear is checked for "square". That is, if the metal to be cut is held tightly against the Guide, is the resultant shear a nearly perfect right angle? If not, the Guide is adjusted until the shear is a right angle. (What tool do we use for checking other tools for right angle?)
- The piece to be cut is laid flat on the table and squared up to the guide. Then the piece is slowly inserted underneath the guard until the shear line scribed on the metal lines up with the end of the

Guide. That scribe line is incredibly hard to see if the Lamp is not turned on and the person operating the shear is peeking in the gap between the Blade and the Guard. Students who are vertically challenged will find it to their great advantage to locate the OSHA approved step stool that is in the tool room.

- d. Once the scribe line is accurately lined up with the end of the Guide, the Handle is slowly brought down to drop both the Guard and the Blade simultaneously. It is very difficult (but not impossible) to get your fingers in a position where they can be sheared off along with the metal. Don't bring the Handle down fast; most people are watching their scribe line and bounce the steel Handle off of their heads.

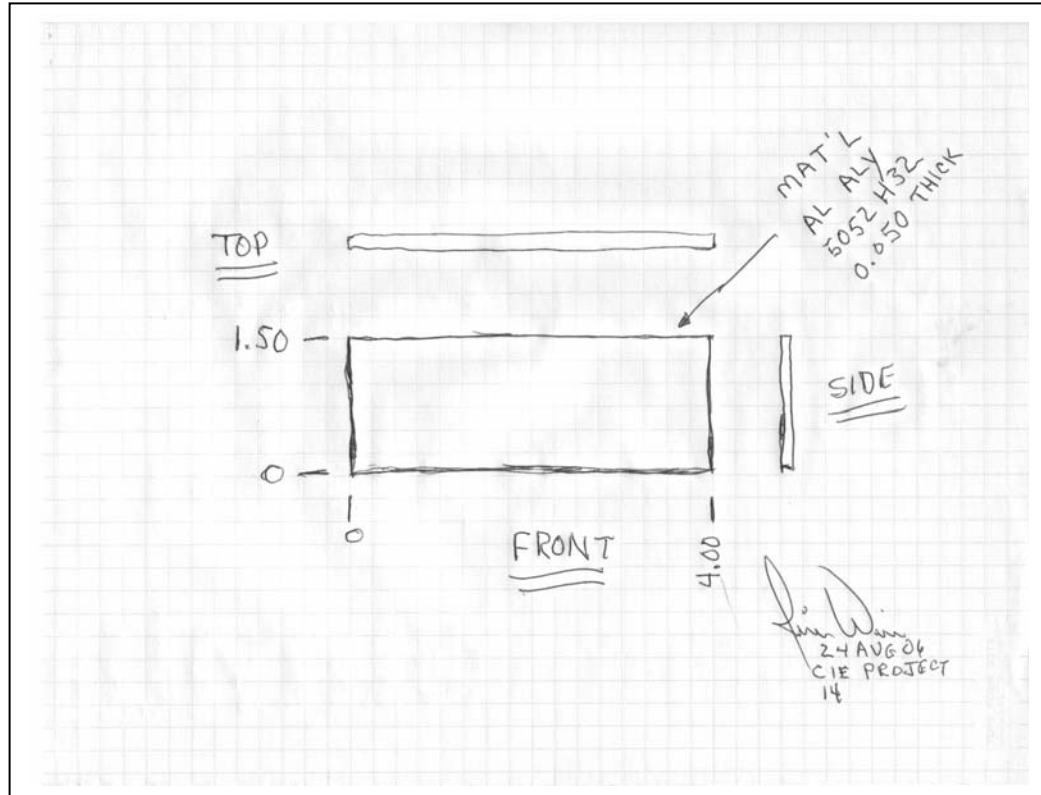
7. Corner Shear

- a. The corner shear is used when you want to take a triangular cut out of metal. Sometimes called a "notch shear" or a "box shear, it is used most often in making chassis where you need a box corner.
- b. In making a box corner, where you want the box to bend up without distortion, it is absolutely mandatory for the metal being cut to be square with the corner shear table. You do this with a combination square lined up with the front or either side of the shear table. CAUTION...we have had more than one combination square cut up when the person doing the cut was more concerned with the cut than whether the square ruler was out of the shear before bringing the handle down
- c. Also note that there is NO blade guard on this shear and it is certainly possible to get one or more human appendages partially into the shear.



EXERCISE: Use the steel ruler, combination square, scribe, and shear to make a flat aluminum plate 1.50 x 4.00 inches rectangular. Use the digital vernier calipers to check final size. Use the micrometer to measure the thickness of the metal.

- a. Make a paper and pencil rough sketch of the part like so (why do you sign and date it?):



- b. **Safety Rule #1. NOBODY** goes out of the classroom into the machine shop without safety glasses. Not for tools, not for parts, not for nothing. **YOU** may not be drilling or grinding, but somebody else may throw a chip directly into your eye from their work.
- c. **Safety Caution #1. While I prefer long pants and closed shoes, if you wear shorts or sandals, be aware while you are working of the possibility of heavy, sharp objects around you.**
- d. There is precut aluminum 2" x 5" in the stockroom. Ask the lab tech to get you a piece of this aluminum. Use the precision square to see if the metal you received is square. If it is not, use the shear to square up the metal by taking a small amount off of each of the 2" wide ends.
- e. Use the ruler, square, and scribe to scratch lines where you are going to cut the metal.
- f. Shear the metal along your scribe lines.
- g. Measure the length and width. It should be within 0.020 of the desired value (widths between 1.48 and 1.52; lengths between 3.98 and 4.02). If not, see if you can figure out what you can do to improve the accuracy and cut another piece of metal. Two tries only.

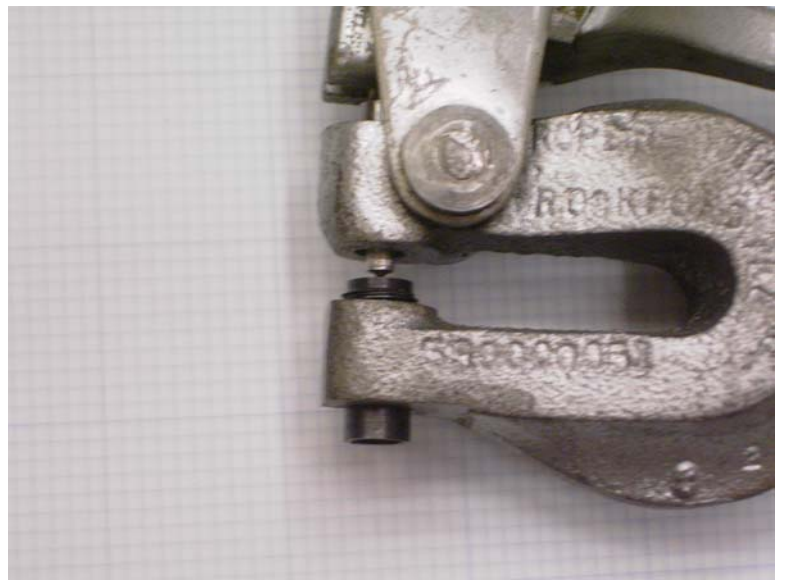
8. Hand Punch

- a. The hand-punch (sometimes called a "Roper-Whitney" after the best known manufacturer of hand punches) can punch a very clean burr-free hole in thin sheet metal. In general, we don't try and punch metals thicker than about 50 mils with a R-W punch.

Note that the hole to be punched can be no further away from the edge of the metal than the depth of the "throat" of the frame.



- b. The "business end" of a R-W punch. Note the silver punch in the top of the frame and the black die in the bottom of the frame. Each size punch has a corresponding die. When you change punch size you must change to the corresponding die size. While the die is relatively easy to change (it simply threads its way into the frame) the punch is relatively difficult to maneuver the jaws around to change.

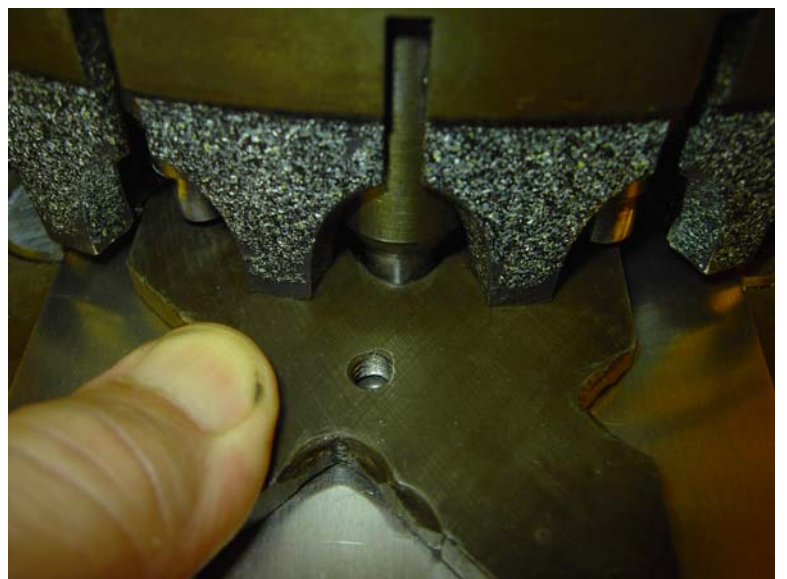
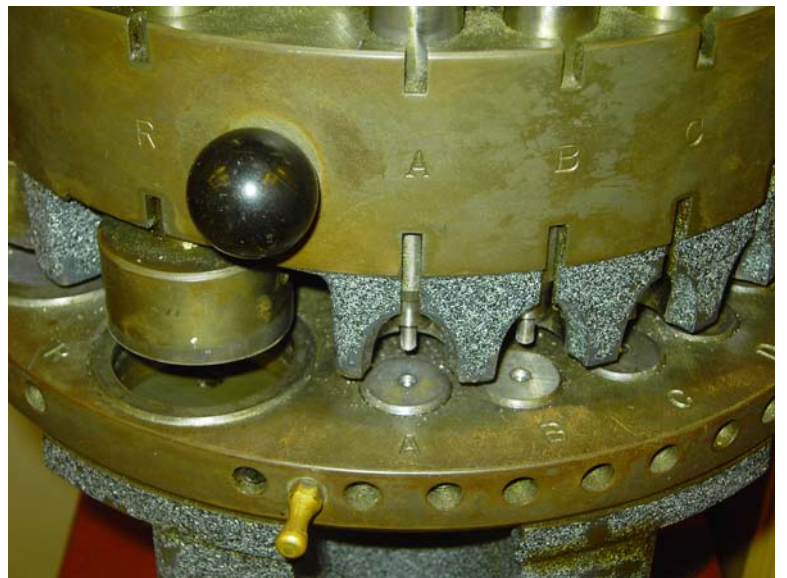


- c. A mated pair of punch (right) and die (left). Note the small point on the left end of the punch. This is to locate the center of the hole in the dimple left by the center punch.



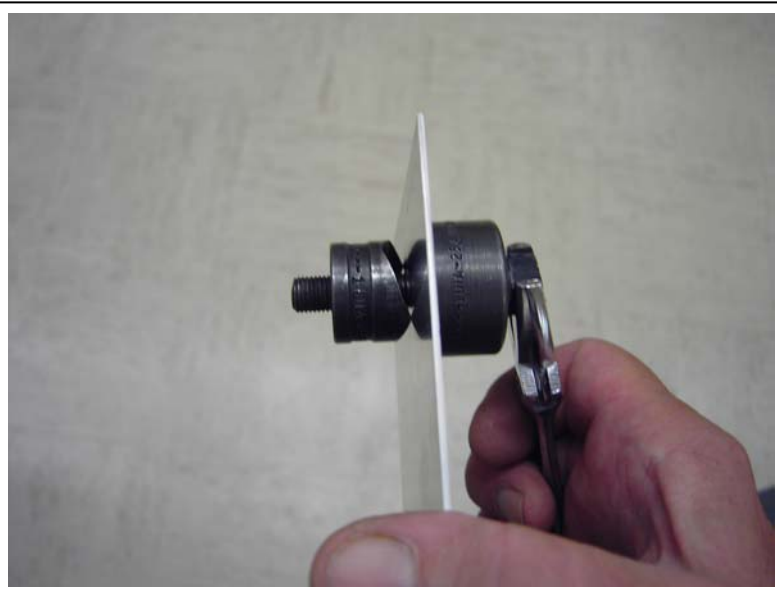
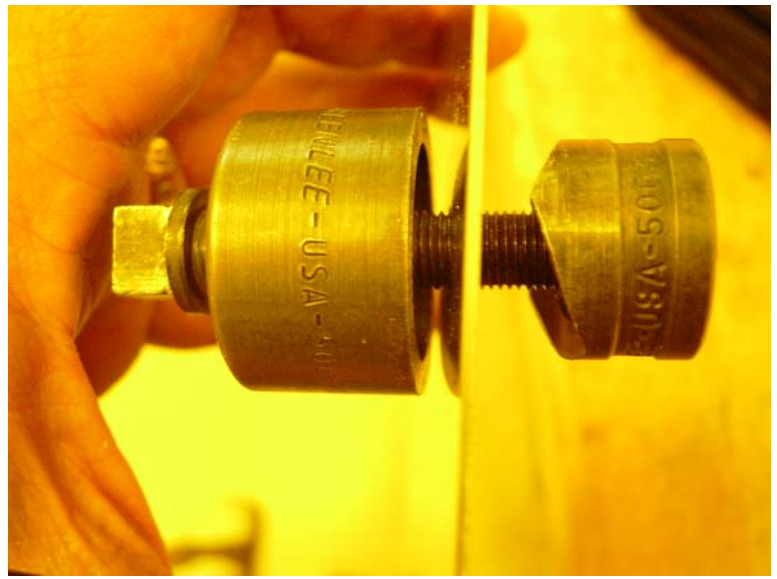
9. Turret Punch

- a. A turret punch is similar to the hand punch, in that a punch comes down and forces metal through a sharp die, punching a burr-free hole. However, instead of having to change the punch and die for different size holes, the turret punch contains all the punches and dies on a rotary platform. The turret punch contains sizes from $\frac{5}{32}$ inch to well over 1 ½ inches in diameter.
- b. The turret punches, too, have a small point at the end of the punch to accurately locate the dimple from the center punch.
- c. When changing hole sizes, one of the critical elements is to make sure that you have the same size punch as you do the die. All the holes are "lettered" from A through R, and it is critical to match up (for instance) a "B" punch with a "B" die.
- d. One problem with the turret punch is that it punches so accurate a diameter hole that the metal wants to "come up" with the punch and deforms the sheet. One answer to this is the punch-plate tool. This tool fits around the punch and keeps the metal sheet that you are punching flat.



10. Chassis Punch

- a. A chassis punch (sometimes called a "Greenlee" punch after the most common manufacturer of this punch) comes in three parts. There is the punch (cylinder on the right) the die (the cylinder on the left) and the bolt (the threaded rod going down the center of the punch and die).
- b. The advantage to the Greenlee is that you are not constrained on large sheets by the depth of the throat of either the Roper-Whitney or turret punches. With either of these last two punches, you are constrained to holes around the edge that you can reach with the available "reach" of the punch frame.
- c. The DISadvantage of a Greenlee is that it requires a good bit of arm strength (especially on large holes) to turn the wrench to bring the punch into the die.
- d. The method of use is to drill a hole in the metal sheet exactly where you want the hole and of a diameter exactly equal to the bolt diameter. You then slip the bolt into the die, through the sheet metal, and then thread and tighten the punch onto the bolt. A box or open end wrench is then used to turn the thread and pull the punch into the die.



11. Pliers

- a. Generically, pliers are used to get a firm grip on a part that you are maneuvering. They come in many sizes and form factors.
- b. Slipjoint pliers ("Channellocks") are so named because one of the jaws slips into a channel in the other jaw to set the opening width of the pliers.



- c. Locking pliers ("Vise Grips") come in several different form factors for different purposes.

The one shown at the right is the generic Vise Grip. There are several permutations of this basic design.



The one shown at the bottom are what are called "duckbills". The advantage of a duckbill is that you can work with a long straight section of metal and get a straight edge on it if you wish to bend it accurately.



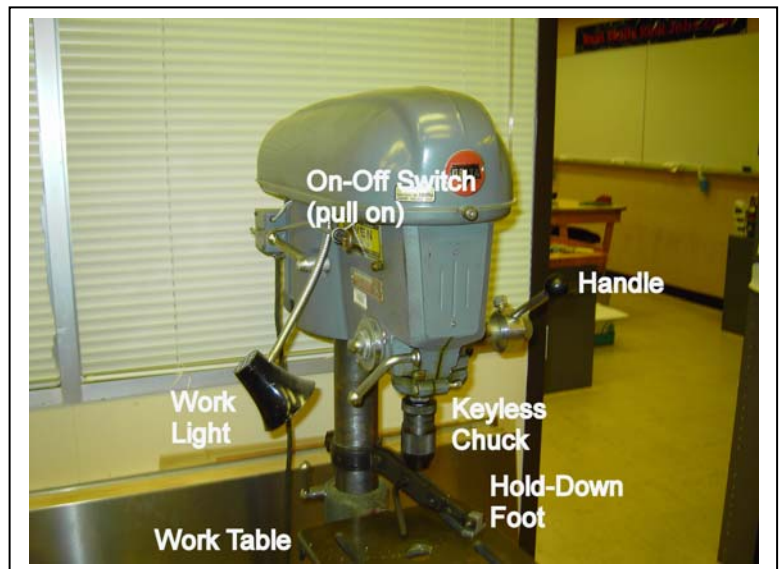
- d. "Gas" pliers are what the layman thinks of when you say pliers. Fairly good in a lot of applications, very good in none of them. Very difficult to use them for sheet metal as they tend to mar and scratch the surface of the metal.



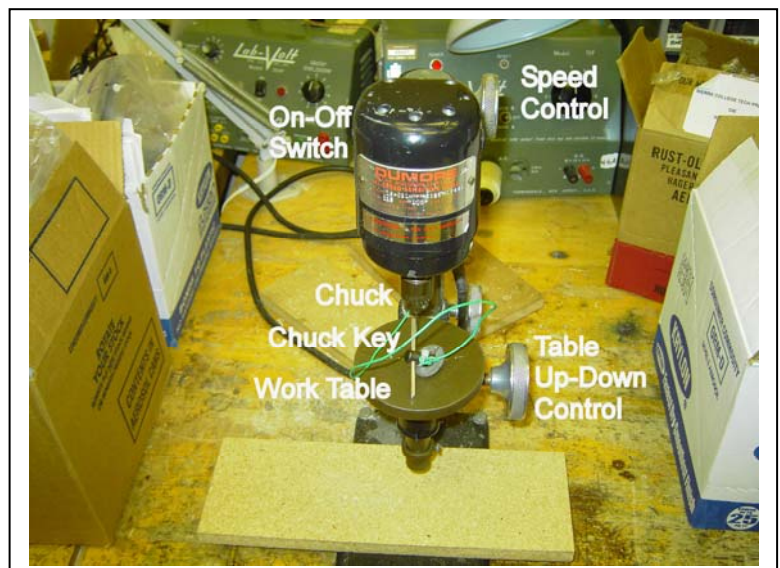
12. Drilling, Reaming, and Nibbling

- a. Metal Drill Press. The metal drill press is used with steel bits to drill holes in sheet metal that for whatever reason cannot be punched.

SAFETY -- Some drill presses are fitted with a safety chuck that is a collet and hand-tightened chuck. Others have a metal key to tighten the chuck. That metal key is a flying bullet if it is left in the chuck and the drill press turned on.



- b. A smaller but faster turning drill is used for drilling through-holes in pc board material. The faster turning drill will generally give you better holes in pcb material if you use a carbide bit. The down side to that is that carbide bits are expensive. **NOTE** that this drill press uses a keyed chuck and extreme care must be taken to insure that the key is out of the chuck before the drill is turned on.



- c. Drill Bits and Standard Sizes. Drill bit diameters come in both fractional sizes (i.e. $\frac{1}{16}$ inch) and numbered/lettered-decimal sizes (i.e. a number 53 -- 0.0635 inches). In general, home workshops work with fractional bits and industry shops work with numbered/lettered. For a lot of "normal" work in an industry shop, here is a table of drills for various screw sizes:

Machine Screws

Dimensions and Other Data

Screw		Threads Per Inch		Clearance Drill*		Tap Drill†			Head					Hex Nut			Washer		
no	dia	coarse	fine	no	dia	no	diameter		round		flat	fillister		across flat	across corner	thick-ness	OD	ID	thick-ness
							inches	mm	max OD	max height	max OD	max OD	max height						
0	0.060	—	80	52	0.064	56	0.047	1.2	0.113	0.053	0.119	0.096	0.059	0.156	0.171	0.046	—	—	—
1	0.073	64	72	47	0.079	53	0.060	1.5	0.138	0.061	0.146	0.118	0.070	0.156	0.171	0.046	—	—	—
2	0.086	56	64	42	0.094	50	0.070	1.8	0.162	0.070	0.172	0.140	0.083	0.187	0.205	0.062	1/4	0.093	0.032
3	0.099	48	—	37	0.104	47	0.079	2.0	0.187	0.078	0.199	0.161	0.095	0.187	0.205	0.062	1/4	0.105	0.020
		—	56			45	0.082	2.1											
4	0.112	40	—	31	0.120	43	0.089	2.3	0.211	0.086	0.225	0.183	0.107	0.250	0.275	0.093	5/16	0.125	0.032
		—	48			42	0.094	2.4											
5	0.125	40	—	29	0.136	38	0.102	2.6	0.236	0.095	0.252	0.205	0.120	0.312	0.344	0.109	3/8	0.140	0.032
		—	44			37	0.104	2.6											
6	0.138	32	—	27	0.144	36	0.107	2.7	0.260	0.103	0.279	0.226	0.132	0.312	0.344	0.109	5/16	0.156	0.026
		—	40			33	0.113	2.9									3/8		0.046
8	0.164	32	—	18	0.170	29	0.136	3.5	0.309	0.119	0.332	0.270	0.156	0.344	0.373	0.125	3/8	0.186	0.032
		—	36			29	0.136	3.5									7/16		0.046
10	0.190	24	—	9	0.196	25	0.150	3.8	0.359	0.136	0.385	0.313	0.180	0.375	0.413	0.125	7/16	0.218	0.036
		—	32			21	0.159	4.0									1/2		0.063
12	0.216	24	—	2	0.221	16	0.177	4.5	0.408	0.152	0.438	0.357	0.205	0.437	0.488	0.156	1/2	0.250	0.063
		—	28			14	0.182	4.6									9/16		
¼	0.250	20	—	—	17/64	7	0.201	5.1	0.472	0.174	0.507	0.414	0.237	0.437	0.488	0.203	9/16	0.281	0.040
		—	28			3	0.213	5.5						0.500	0.577	0.250	5/8		0.063

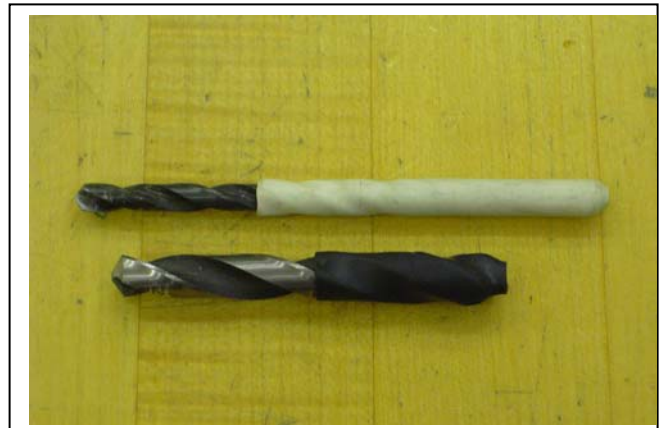
All dimensions in inches except where noted.

* Clearance-drill sizes are practical values for use of the engineer or technician doing his own shop work.

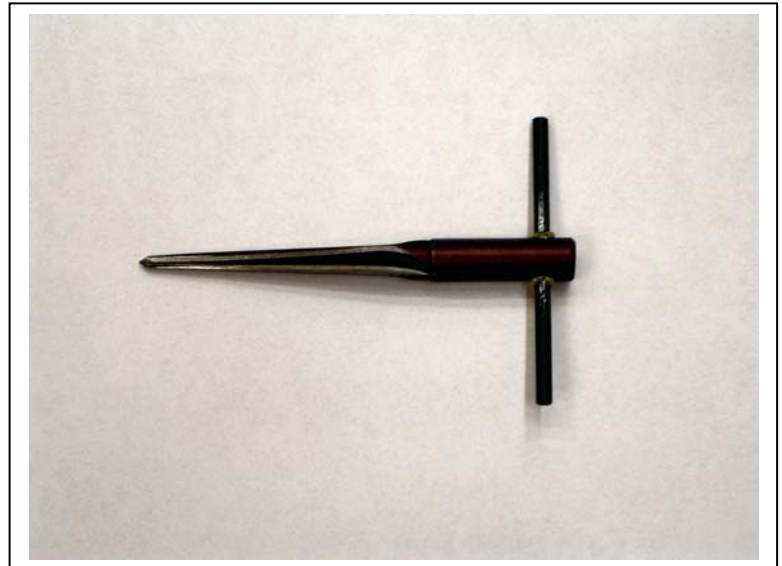
† Tap-drill sizes are for use in hand tapping material such as brass or soft steel. For copper, aluminum, cast iron, bakelite, or very thin material, the drill should be a size or two larger diameter than shown.

Brought to you courtesy of PST Engineering, Grass Valley, CA 95945 530 272 2203 sales@pst-engr.com

- d. Deburring. In general, drilling a hole will create slivers of metal remaining on the part that have been tossed out by the bit. These slivers go by the name of "burrs". A good deburring tool is a large drill bit wrapped with masking tape or shrink sleeving to keep the sharp bit flutes from attacking your hands.



- e. When you need to have a hole that is not close to any standard punch or drill size, you can use a T-handled reamer to ream the hole to the exact size you need.



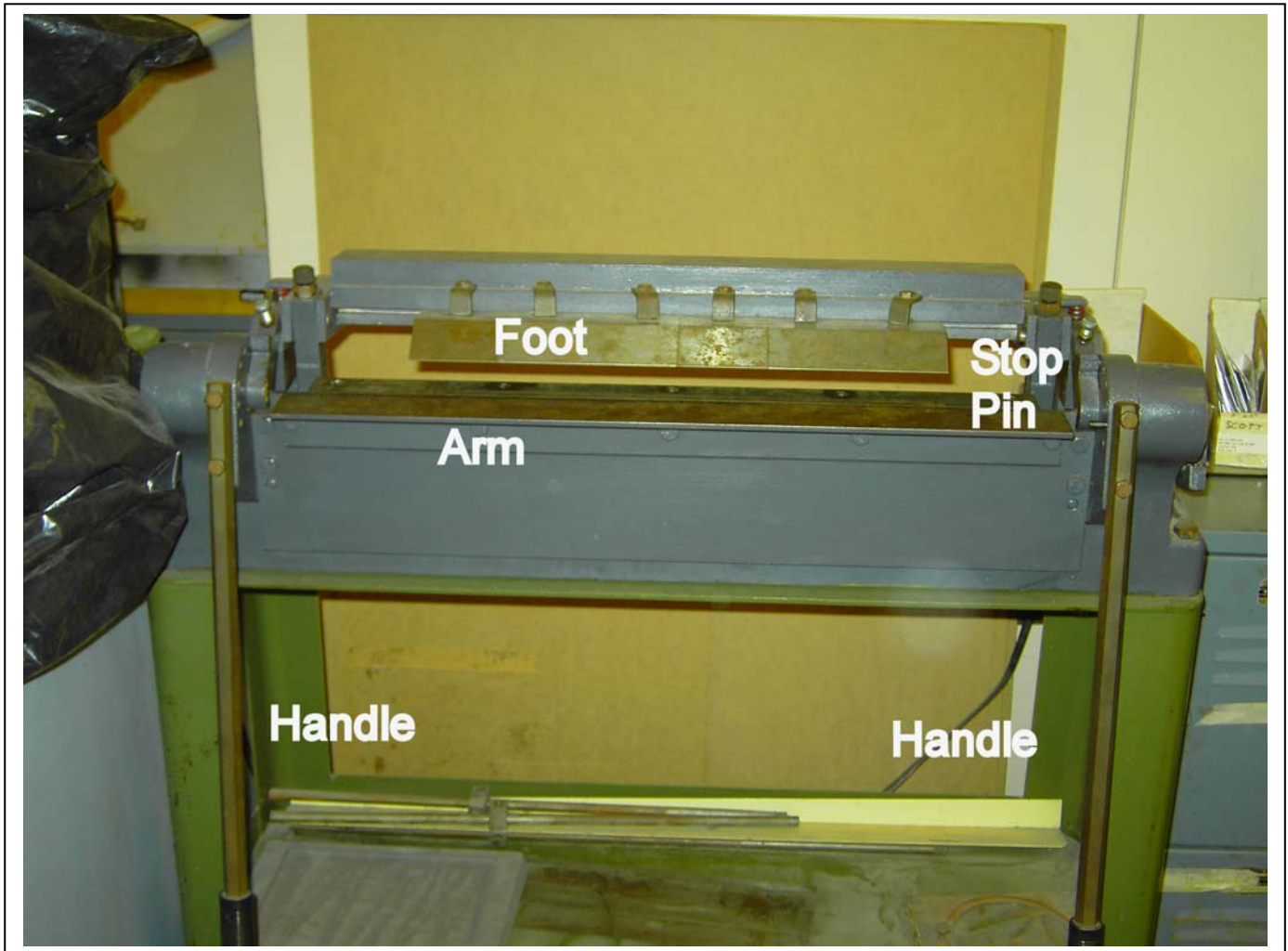
- f. When the hole has to be some shape other than round or not the same size as any special punch that you have, you can use the nibbling tool to take rice-grain size pieces out of the metal.

The procedure is to drill a hole slightly larger than the tool diameter on the bottom of the nibbling tool, insert the tool up to the bottom of the tool, then nibble away at the unwanted metal.

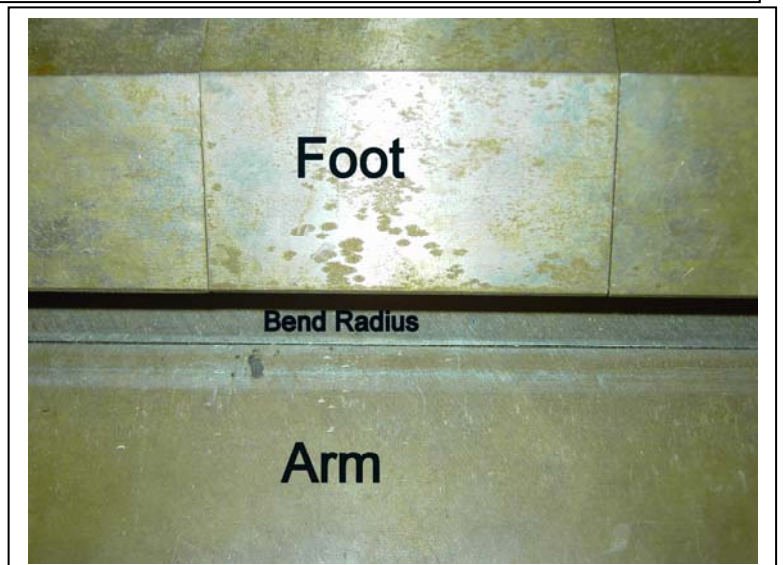


13. The Bending Brake

- a. The bending brake is used to bend metal at almost any angle that you choose. It consists of the Foot (which holds one side of the metal flat and rigid), the Base (which is what the Foot presses against), the Arm (which is the movable piece that does the bending of the metal), and the Stop Pin (which keeps the brake from overbending the metal).



- b. In order to bend a piece of metal without cracking it, you need to set a BEND RADIUS, which allows the metal to have a smooth radius rather than be torn or cracked. The BEND RADIUS is set by how far the Foot is back from the Arm.



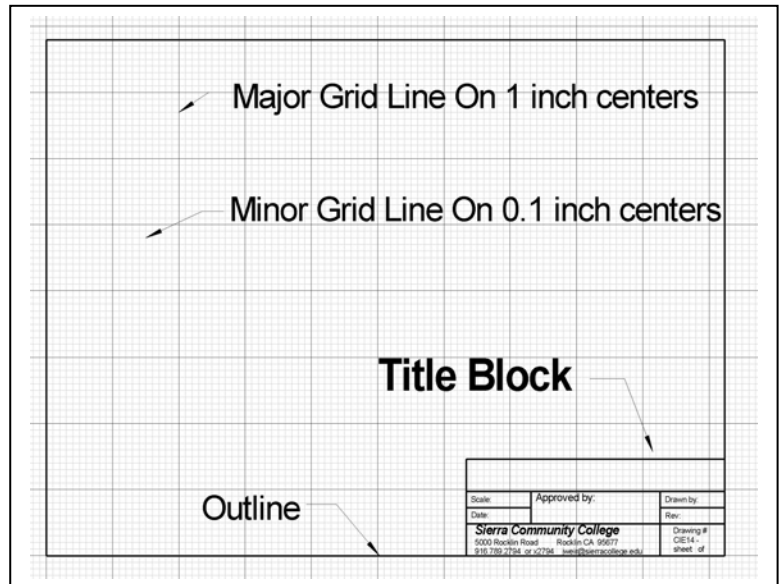
14. Engineering Drawings. The pencil and back-of-envelope sketch we did to fabricate the little metal plate in the previous section is OK for quick-and-dirty work. However, for something that has half a chance to go into production, we need something a little more formal.

a. The "a little more formal" looks like this. We have a preprinted sheet of paper with the following things on it:

1. A Title Block
2. A Grid with major and minor divisions
3. An Outline

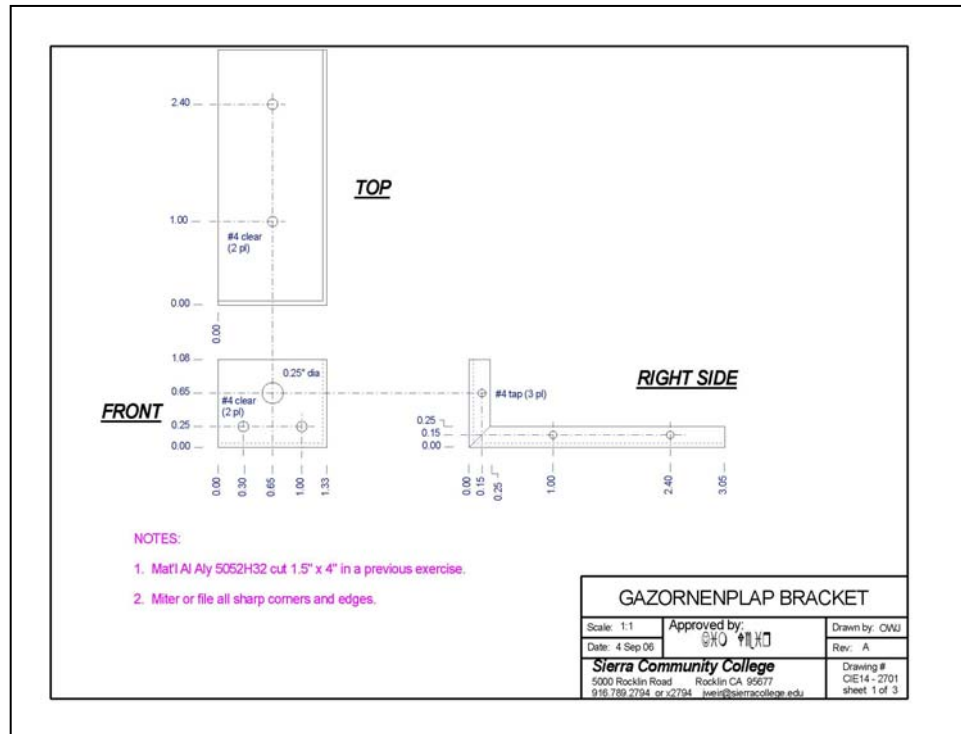
b. The Title Block has the following:

1. The actual title of the drawing (Chassis Cover, Barf Bracket, etc.)
2. The scale that the drawing bears to the real world. For example, a scale of 2:1 would mean that a part that was drawn 4" x 2" would actually be manufactured 2" x 1".
3. The date that the drawing was COMPLETED.
4. Who drew it.
5. The REVision level starting with A and proceeding up the alphabet.
6. The organization that drew it.
7. A drawing number.
8. The signature of the person approving it.

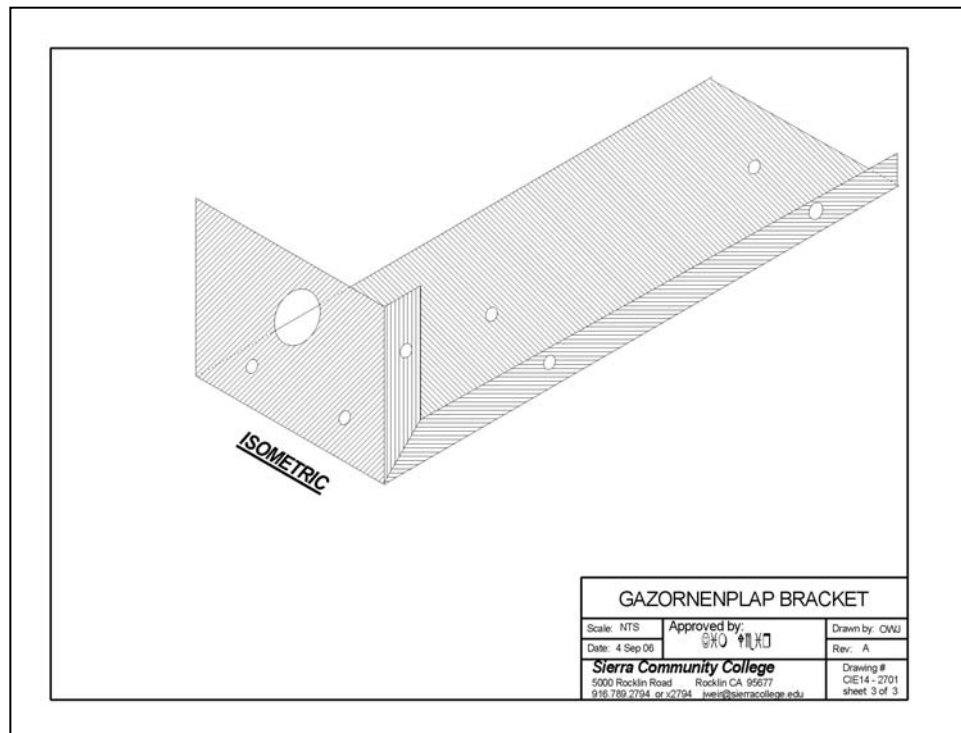


- c. Unless it is a very special part, with lots of unusual cuts and bends, we need three views of the part in order to make it out of sheet metal. We need a FRONT view, a TOP view, and a RIGHT SIDE view. If we are not making the part ourselves, this will be sufficient. However, if we are making it ourselves, we need a third drawing called a BEND drawing, which tells us how we want to punch, drill, and bend this puppy up. If we want to get REAL fancy, we can do an isometric ("3-d") view to remind ourselves just what it is we are trying to make.
- d. A blank engineering drawing sheet suitable to print out for pen or pencil is in the ImagesPDF folder on the CDROM under image number TitleGraph.pdf.

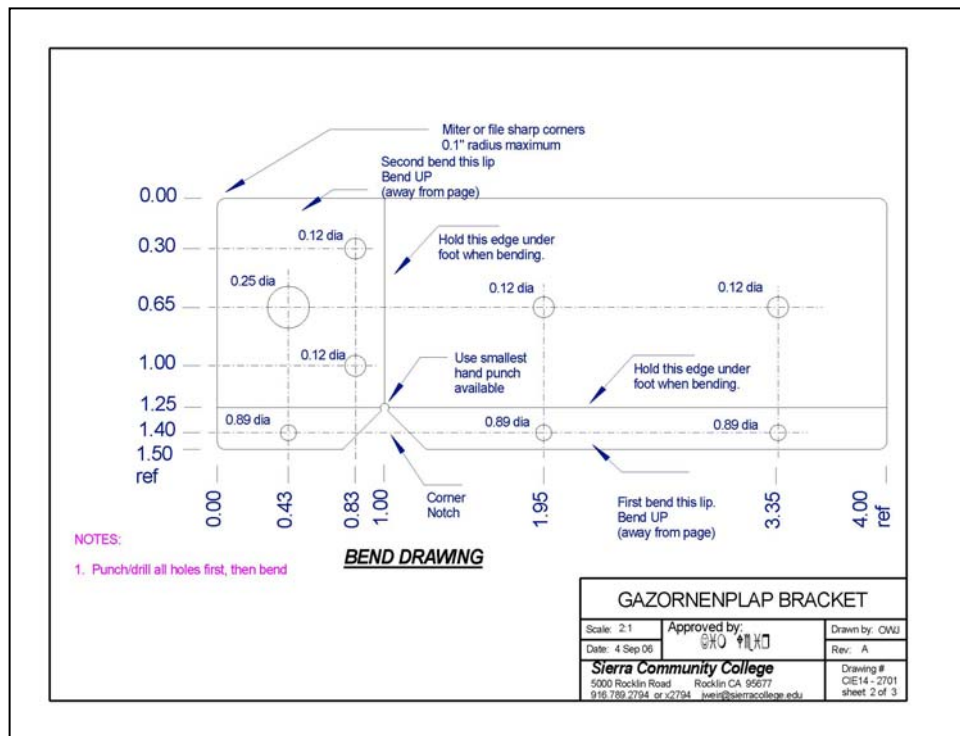
15. Now that we know how to make an engineering drawing, let's bend up and punch/drill our little 1.5" x 4" metal plate into a bracket like this (make your own engineering drawing using these dimensions):



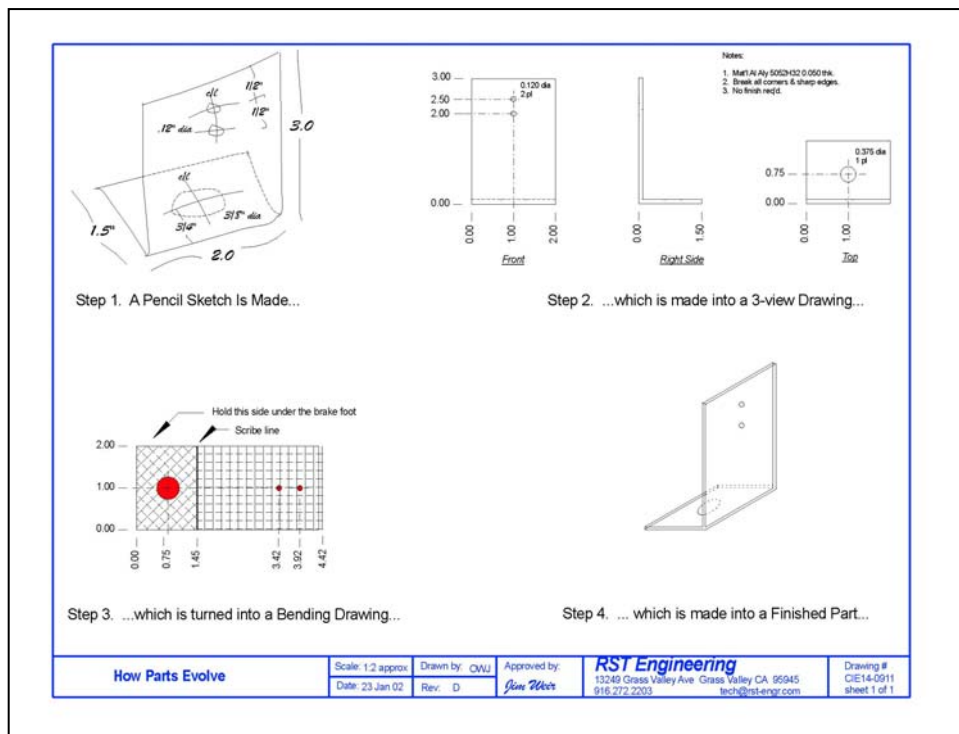
16. Or, if you want an isometric drawing to visualize what this little bracket is going to look like:



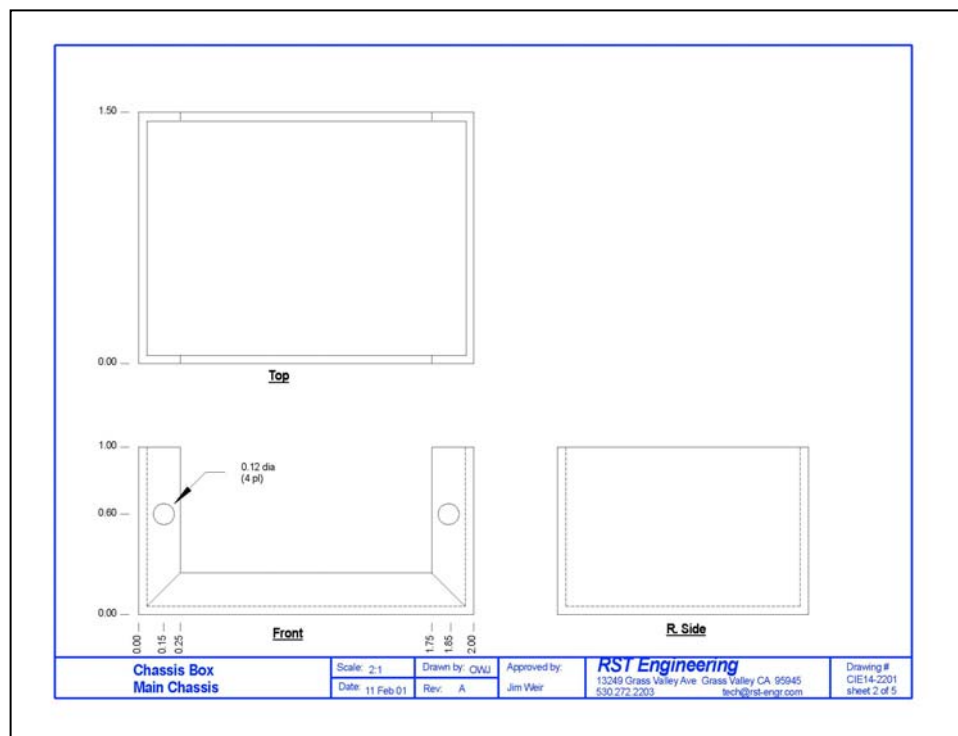
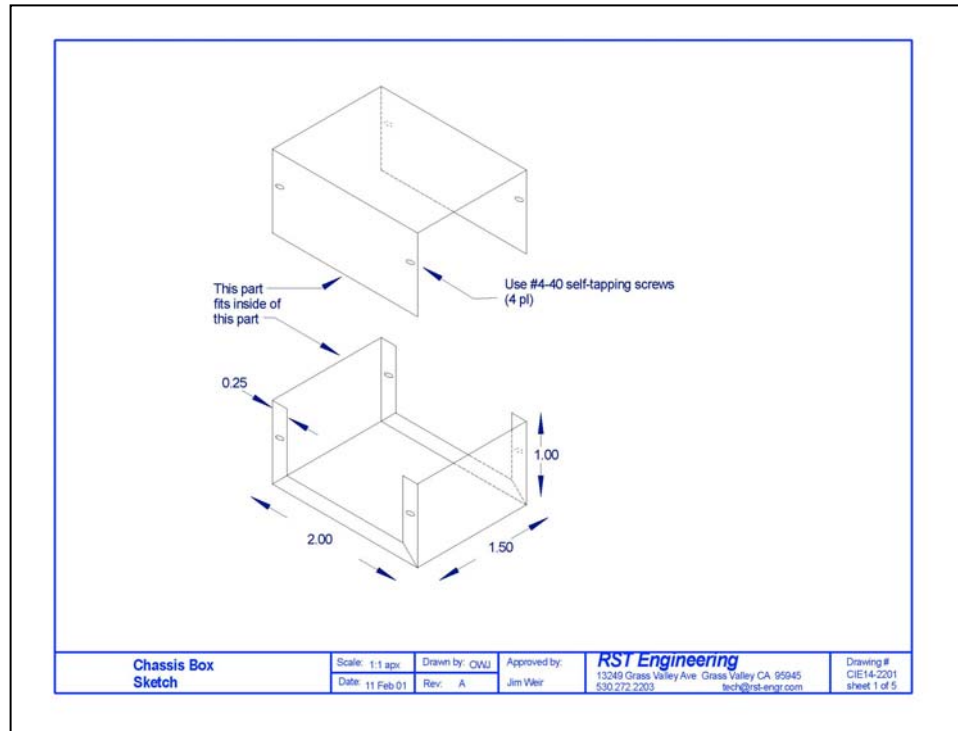
17. And now the bend drawing so that we can mark and scribe on the metal plate (again, do the drawing before you do the scribing and bending):

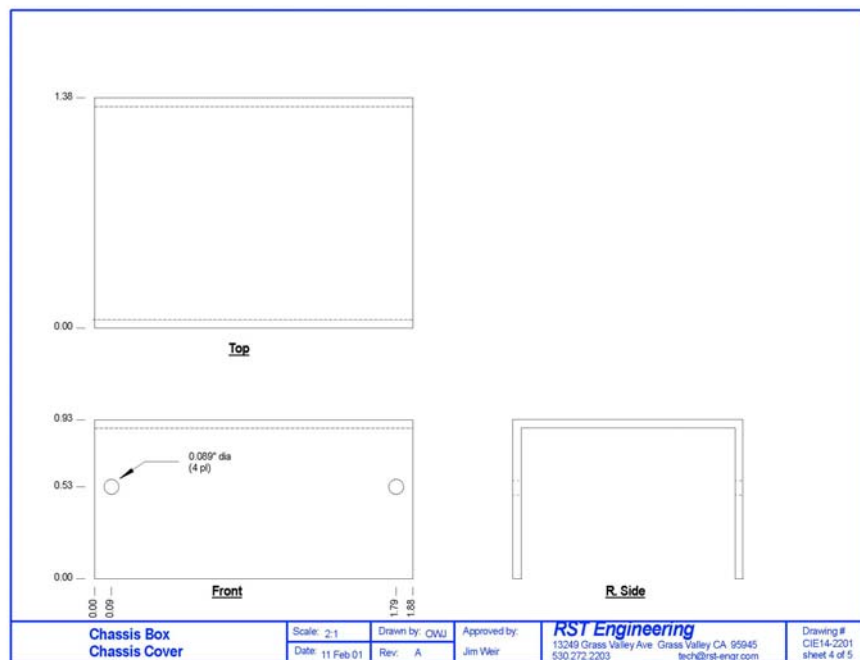
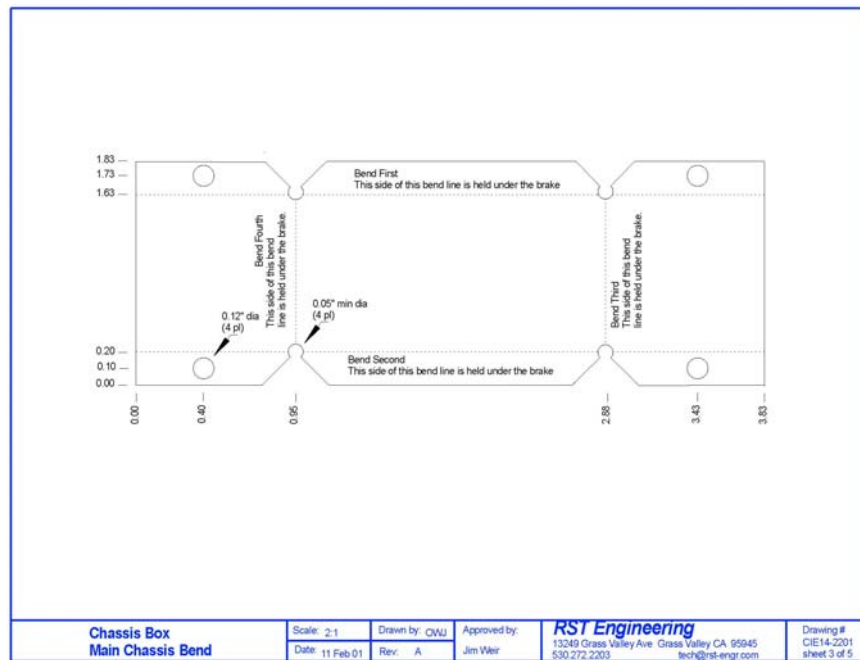


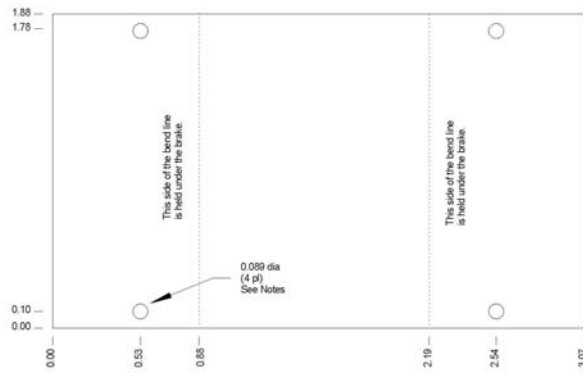
18. Let's reiterate the steps that we take to manufacture a part:



19. Now that you all have some experience with metal bending and punching/drilling, make a small utility box as shown in the following five drawings. Do the engineering drawings first, then do the metalwork. Note that all of these drawings are in the ImagesPDF folder on the CDROM under their correct drawing number.







NOTES

1. Do NOT drill or punch the 4 each 0.089" holes at this time.
Mate this part with the other half of the chassis box and use the
holes on the other half to locate these holes exactly.

Chassis Box
Chassis Cover Bend

Scale: 2:1
Date: 11 Feb 01

Drawn by: QWJ
Rev: A

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave. Grass Valley CA 95945
530.272.2203
tech@rst-engr.com

Drawing #:
CIE14-2201
sheet 5 of 5

20. I'll just bet that you are wondering why I called the bracket a "2701" number and the little chassis a "2201" number. It is true that setting up a drawing numbering system is one of the more difficult tasks of a new company. Unfortunately, there is no universal standard, nor is there a "suggested" standard. It seems that "every company for itself". In the attached drafting numbering scheme, note that 2701 is a bracket fabrication number and 2201 is a chassis number.

DRAWING NUMBER ASSIGNMENTS

This scheme shall be used to assign drawing numbers to engineering drawings produced in accordance with this document.

Unless otherwise specified, use numerals in place of !@# that are sequential within the category.

The 3 or 4-digit model number shall precede the 4-digit drawing number with a hyphen between the two numbers (viz. 901-2100 for the first faceplate drawing number for family 901).

Family 999-0!@# defines a generic drawing that is valid for all model numbers.

STANDARD DRAWINGS

0000 - 0099	(undefined)
0100	Model/Serial Labels
0101 - 0199	Decal
0200 - 0299	PC Board
0300 - 0399	Schematic
0400 - 0499	Test Equipment
0500 - 0599	Antenna
0600 - 0609	3-d and Conceptual
0610 - 0619	Dimensioned Models (Paper Models)
0620 - 0699	Mechanical Drawing
0700 - 0899	Electrical Part
0900	Top Drawing
0901	Working Drawing
0902	Master MPL
0903 - 0909	(undefined)
0910 - 0999	General Documents - Miscellaneous

ARTWORK SCREENS

1000 - 1099	(undefined)
1100 - 1199	Faceplate
1200 - 1299	Main Chassis/Cover
1300 - 1399	#2 Chassis/Cover
1400 - 1499	#3 Chassis/Cover
1500 - 1599	#4 Chassis/Cover
1600 - 1699	Mounting Tray
1700 - 1799	Bracket
1800 - 1899	Electrical Part
1900 - 1999	Miscellaneous

FABRICATION DRAWINGS

2000 - 2099	(undefined)
2100 - 2199	Faceplate
2200 - 2299	Main Chassis/Cover
2300 - 2399	#2 Chassis/Cover
2400 - 2499	#3 Chassis/Cover
2500 - 2599	#4 Chassis/Cover
2600 - 2699	Mounting Tray
2700 - 2799	Bracket
2800 - 2899	Electrical Part
2900 - 2949	Chassis Schematic
2950 - 2999	Miscellaneous

INSTALLATION DRAWINGS

3000 - 3099	(undefined)
3100 - 3199	Faceplate
3200 - 3299	Main Chassis/Cover
3300 - 3399	#2 Chassis/Cover
3400 - 3499	#3 Chassis/Cover
3500 - 3599	#4 Chassis/Cover
3600 - 3699	Mounting Tray
3700 - 3799	Bracket
3800 - 3899	Electrical Part
3900 - 3949	Miscellaneous Mechanical
3950 - 3999	Miscellaneous Electrical

ASSEMBLY DRAWINGS

4000 - 4099	(undefined)
4100 - 4199	Faceplate
4200 - 4299	Main Chassis/Cover
4300 - 4399	#2 Chassis/Cover
4400 - 4499	#3 Chassis/Cover
4500 - 4599	#4 Chassis/Cover
4600 - 4699	Mounting Tray
4700 - 4799	Bracket
4800 - 4899	Electrical Part
4900 - 4949	3-D views
4950 - 4999	Miscellaneous

PC BOARD SCREEN DRAWINGS

5000 - 5099	(undefined)
5100 - 5199	Top (or Composite) Reference Designator

5200 - 5299	Top Solder Mask
5300 - 5399	Top Parts Value
5400 - 5499	Bottom Reference Designator
5500 - 5599	Bottom Solder Mask
5600 - 5699	Bottom Parts Value
5700 - 5799	X-Ray Trace View
5800 - 5899	Schematic
5900 - 5999	Miscellaneous

PC BOARD TRACE & DRILL DRAWINGS

6000 - 6099	(undefined)
6100 - 6199	Top (or Composite) Trace Artwork
6200 - 6299	#2 Layer Trace Artwork
6300 - 6399	#3 Layer Trace Artwork
6400 - 6499	#4 Layer Trace Artwork
6500 - 6599	#5 Layer Trace Artwork
6600 - 6699	Bottom Trace Artwork
6700 - 6799	PCB Graphics Files
6800 - 6899	Master Composite All Layers
6900 - 6999	Gerber Files

7000 Series [Reserved]

DOCUMENTATION

8000 - 8099	(undefined)
8100 - 8119	Operations and Maintenance Manual
8120 - 8139	Construction and Assembly Manual
8140 - 8199	(undefined *)
8200	Engineering Change Order Log
8201- 8399	Engineering Change Order & Updates
8401 - 8649	Update Data Not A Part Of An ECO
8650 - 8699	Special Test Equipment
8700 - 8799	Subassembly & Miscellaneous Parts Lists
8700 - 8709	Documentation
8710 - 8719	Chassis/Metalwork
8720 - 8729	Hardware
8730 - 8739	Wire
8740 - 8749	Connectors
8750 - 8759	Switches
8790 - 8799	Miscellaneous
8800 - 8899	PC Board Parts Lists
8800	PCB #1 Subassembly MPL
8801	PCB #1 (Drawing #6801) Resistors
8802	PCB #1 Capacitors
8803	PCB #1 Semiconductors
8804	PCB #1 Magnetics (Relays, Transformers, etc.)
8809	PCB #1 Miscellaneous

8810	PCB #2 Subassembly MPL (and so on through 8899)
8900 - 8949	Chassis Parts Lists
8950 - 8959	Manufacturing Cover Letters
8960 - 8969	Travelers
8970 - 8999	Miscellaneous

9000 Series [Reserved]

- * Special Cases:
 - 000-0980 through 0999 is blank graph paper
 - 999-8140 is the Master Parts List
 - 999-8141 is the Approved Vendor List
 - 999-8142 is the blank form for adding a part to the Master Parts List
 - 999-8150 is the Engineering Drafting Conventions
 - 999-8160 is the Part Numbering System
 - 999-8170 through 8199 is standard procedures and processes

- *Engineering Dwgs:
 - 000-0600 is general engineering drawing block outline lower right hand corner
 - 000-0601 is general engineering drawing block outline along bottom
 - 000-1100 is Faceplate Screen with notes
 - 000-2100 is Faceplate with material & hole size notes
 - 000-2200 is Chassis with material & hole size notes

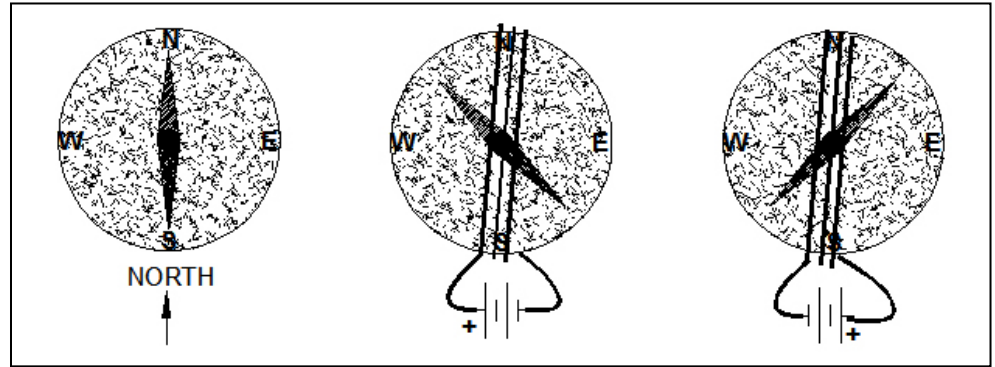
21. This concludes the Mechanical Metal-Forming Tools section of this book. Next we will investigate some electrical tools to help us with the project.

III. ELECTRICAL TOOLS

A. ANALOG AND DIGITAL VOLT/AMPERE/OHM METERS

1. Current Meters

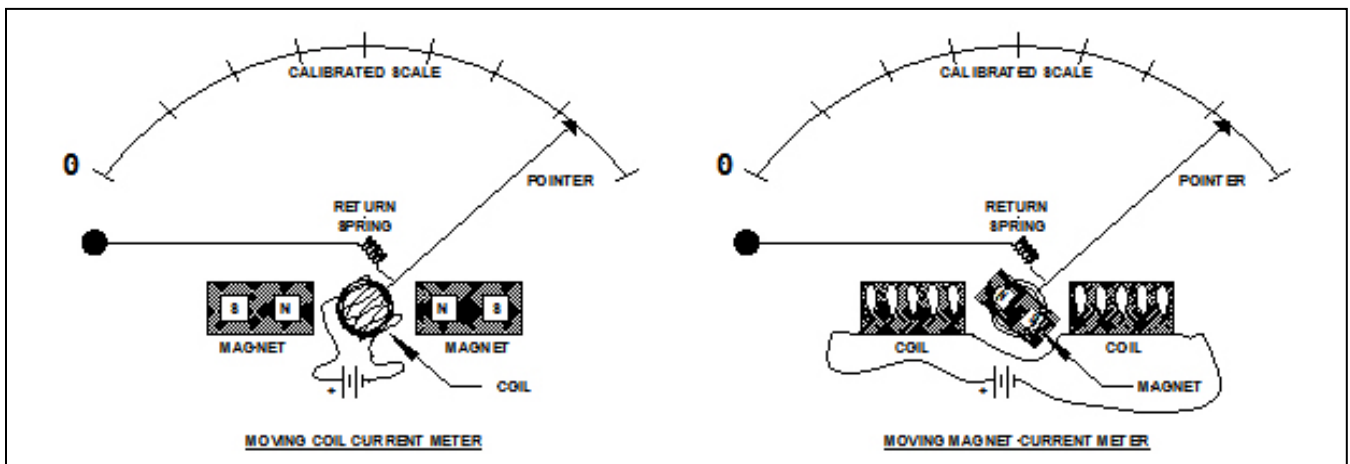
- a. A very long time ago, when we were just learning what "electricity" was all about, we found out that if we wrapped a few turns of wire around a compass and passed an electrical current



through the wire that the compass needle swung away from magnetic north. If we reversed the current the compass needle also swung away from magnetic north, but in the opposite direction.

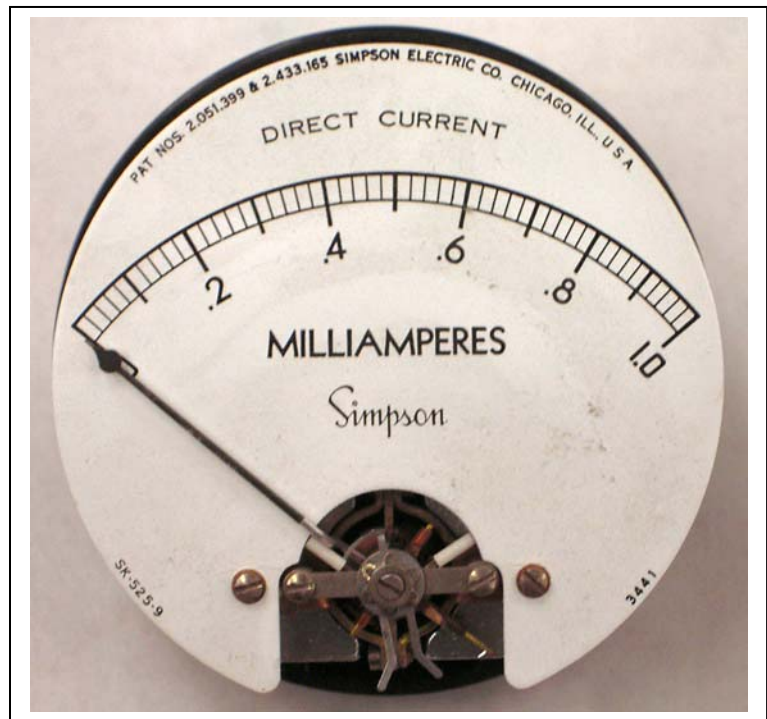
The problem with measuring electrical current this way is that you had to point the compass at north, wind the wire parallel to the needle, and then take your measurement. While this was clumsy to say the least, it was all we had for a few dozen years.

- b. When we became a little more knowledgeable about electricity, we found that we could take a relatively strong magnet that completely overwhelmed the earth's magnetic field, fasten a needle to that magnet, wind a coil of wire on either side of the magnet, fasten a spring to the needle and make a true moving magnet current meter. This was a great advance in the state of the current measuring art. No longer were we tied to a compass and the earth's magnetic field.



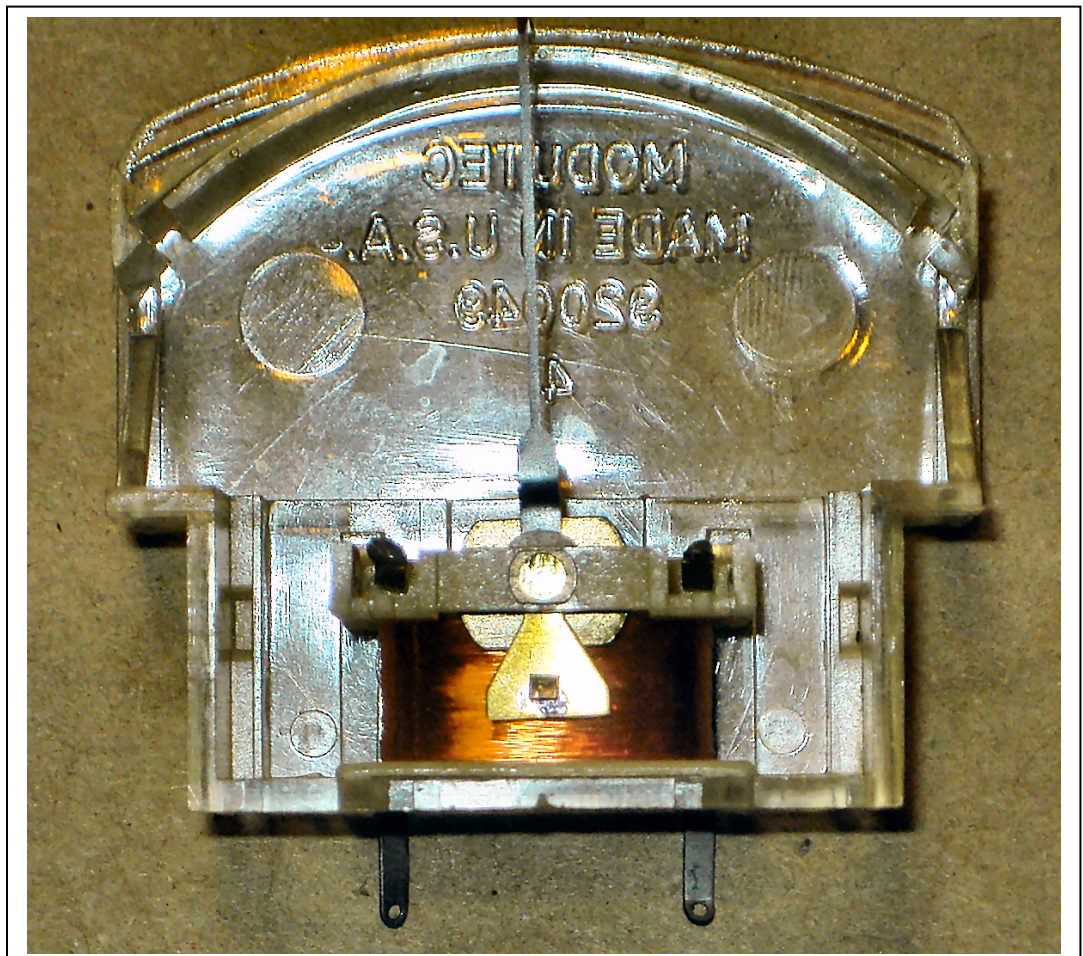
- c. Things haven't changed much over the last couple of hundred years except for the fact that we found a good way to fasten the needle to the coil of wire and have a fixed magnet provide the magnetic field. Thus today we have moving-magnet and moving-coil versions of a current meter.

Here is a picture of one those moving-coil meters. This meter goes from to one milliampere full scale of current.



of
zero

- d. Here is a close-up of the "coil" end of a moving magnet meter. Note the orange copper coil; the magnet is hidden inside the coil of wire and attached to the needle pivot.



- e. Up until the late 1980s, these pointer (or ANALOG) meters were the standard. When we perfected the DIGITAL readout to where it was a tenth of the cost of an analog meter, the digital meter became the standard.

Here is a photo of an analog "multimeter". A multimeter uses internal circuitry to allow us to read current PLUS voltage PLUS resistance.



The digital "multimeter" (more later) shown here sells for less than \$4. A comparable analog meter (as shown above) will sell for \$75 or more. Note that the digital meter is reading 33.4 milliamperes.

Current is measured in AMPERES. A milli-ampere is $\frac{1}{1000}$ of an ampere.

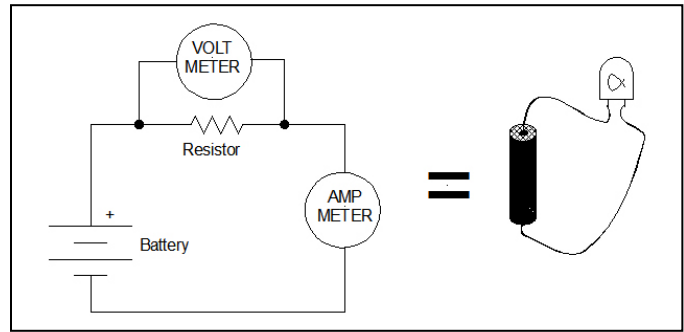


2. Ohm's Law

- a. There are three basic quantities in Ohm's Law -- Current (measured in amperes), Voltage (measured in volts) and Resistance (measured in ohms). (Andre Ampere, French, 1775-1836; Alessandro Volta, Italian, 1745-1827; Georg Ohm, German, 1789-1854)

Ohm was the first to recognize the relationship between voltage and current. He said that the voltage in a circuit divided by the current flowing in that circuit was the resistance. Or, to put it in mathematical terms, $R = \frac{E}{I}$ where R is the resistance (in ohms), E is the voltage (E lectromotive force) in volts, and I is the current in amperes. A bit of algebra allows us to rearrange this equation to solve for voltage if we know resistance and current $E = I * R$ and current if we know voltage and resistance $I = \frac{E}{R}$.

Take a look at the very simple circuit shown to the right. Here we see a battery going through a resistance of some sort with the current being measured by the AMPMETER and the voltage being measured by the VOLTMETER.

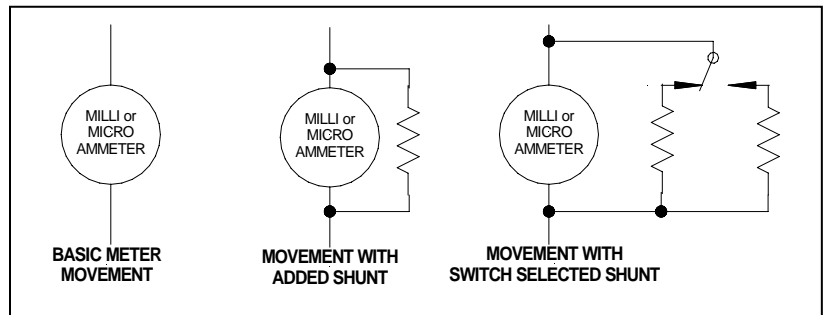


It is *IMPORTANT* to remember that an AMPMETER measures current in SERIES with the circuit, and a VOLTMETER measures current in PARALLEL with the circuit.

This SCHEMATIC (symbol) circuit on the left might be as simple as the PICTORIAL (picture) circuit on the right. Here on the right we see a light bulb being lit by a flashlight battery. If the flashlight battery measures 1.5 volts and the ampmeter (usually spelled "ammeter") measures 100 milliamperes, we can say that the bulb's resistance is $R = 1.5 / 0.100 = 15$ ohms (abbreviated 15Ω , where Ω is the Greek "omega").

3. Changing Scales On An Ammeter

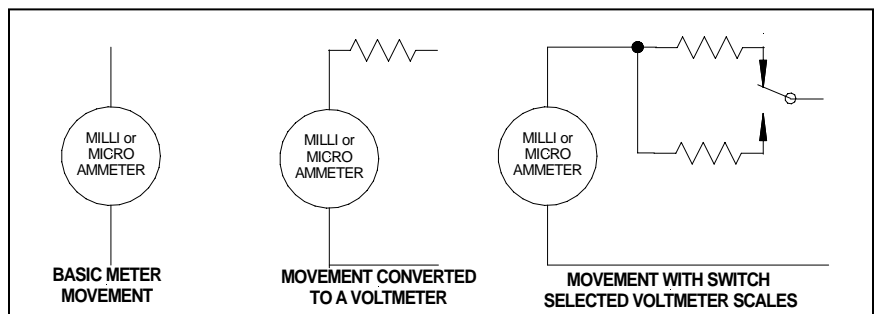
- a. In general, we start off with a basic "meter movement" that is measured in milliamperes (thousandths of an ampere) or microamperes (millionths of an



ampere) and shunt the meter movement with an external resistor so that some of the current goes through the meter and some of it goes through the resistor shunt. While it is not necessary for you to know at this point how to calculate meter shunts, suffice it to say that by switch-selecting various shunts you can have the same meter movement read almost any current above the basic movement current that you wish. For example, if your basic meter movement is one milliampere, you could shunt the meter with a resistor such that with an ampere of current flowing in the circuit, 999 milliamperes goes through the shunt and one milliampere goes through the meter. You have converted your one milliampere meter into reading one ampere full scale..

4. Turning An Ammeter Into A Voltmeter

- a. As we saw above with Brother Ohm's law, if we know the RESISTANCE and the CURRENT in a circuit, we know the voltage. In particular,



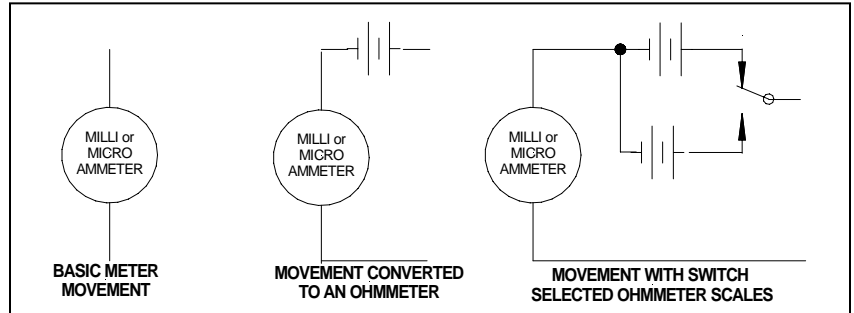
if we have an ammeter (and I'm using the generic term here for ammeter, milliammeter, and

microammeter) with a known resistance in SERIES, we have converted our ammeter into a voltmeter.

- b. In particular, for the purposes of this class, if I have a meter with a basic 0-1 mA (zero to one milliampere) full scale range and I want to turn it into a 0-15 volts (zero to 15 volts) voltmeter, I can do so by putting a ($R = \frac{E}{I}$ $R = \frac{15}{0.001}$) 15,000 ohm resistor in series with the meter. The meter supplied with your power supply kit is a 0-1 mA meter that will be converted to a 0-15 volt voltmeter with the addition of a 15,000 (15 kilohm, or 15k) resistor.

5. Turning An Ammeter Into An Ohmmeter

- a. I don't think it takes a great leap of intuition to see that if we have a basic meter movement that measures current knowing the circuit's voltage and current, and we can turn a meter that knows current with a

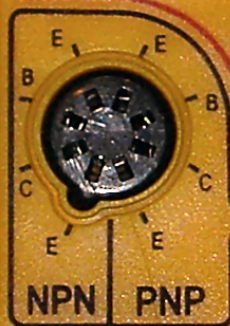


known resistance into a voltmeter, we should be able to turn a meter that knows current with a known voltage into something that measures resistance.

6. The Digital Multimeter

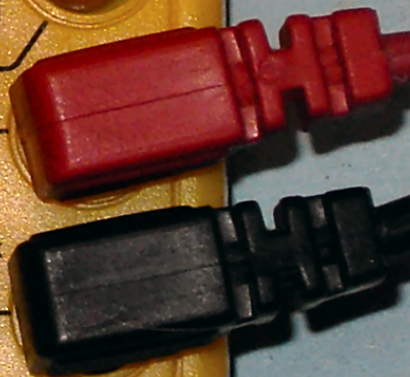
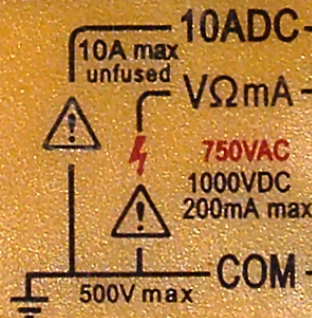
- a. As we noted above, digital multimeters have taken over the marketplace for instruments that measure voltage, current, and resistance (plus some other goodies that we couldn't come close to doing with analog instruments).
- b. The meter chosen (see next page) has scales for direct current voltage (DCV), alternating current voltage (ACV), direct current amperage (DCA), Ohms (Ω) and a few other goodies that we don't need to worry about (transistor and diode testing)
- c. It measures full scale dc voltage from 200 millivolts full scale to 1000 volts (one kilovolt) full scale, ac voltage from 200 volts full scale to 750 volts full scale, dc current from 200 microamperes (μA) full scale to ten amperes full scale, and ohms from 200 ohms full scale to 2000 kilohms (2 mega ohms) full scale.
- d. Note the RED (positive) and BLACK (negative) test leads exiting the bottom of the photo.
- e. Note also that this particular model has an "OFF" position. If you do not turn the multimeter off when you are through using it, the internal battery will slowly run down to where it is dead.
- f. Inside the meter (see page after next) you can see the battery (bottom of the picture) and immediately above it a little glass tube with metal ends. This is the meter fuse. If you try and measure volts on any of the amperage scales, you will blow the fuse. Not to worry; we've all done it. The important thing is to replace the fuse, not to put the meter back on the shelf hoping the next person will find the problem. (Look closely at the white triangle above the fuse for the current rating of the replacement fuse).

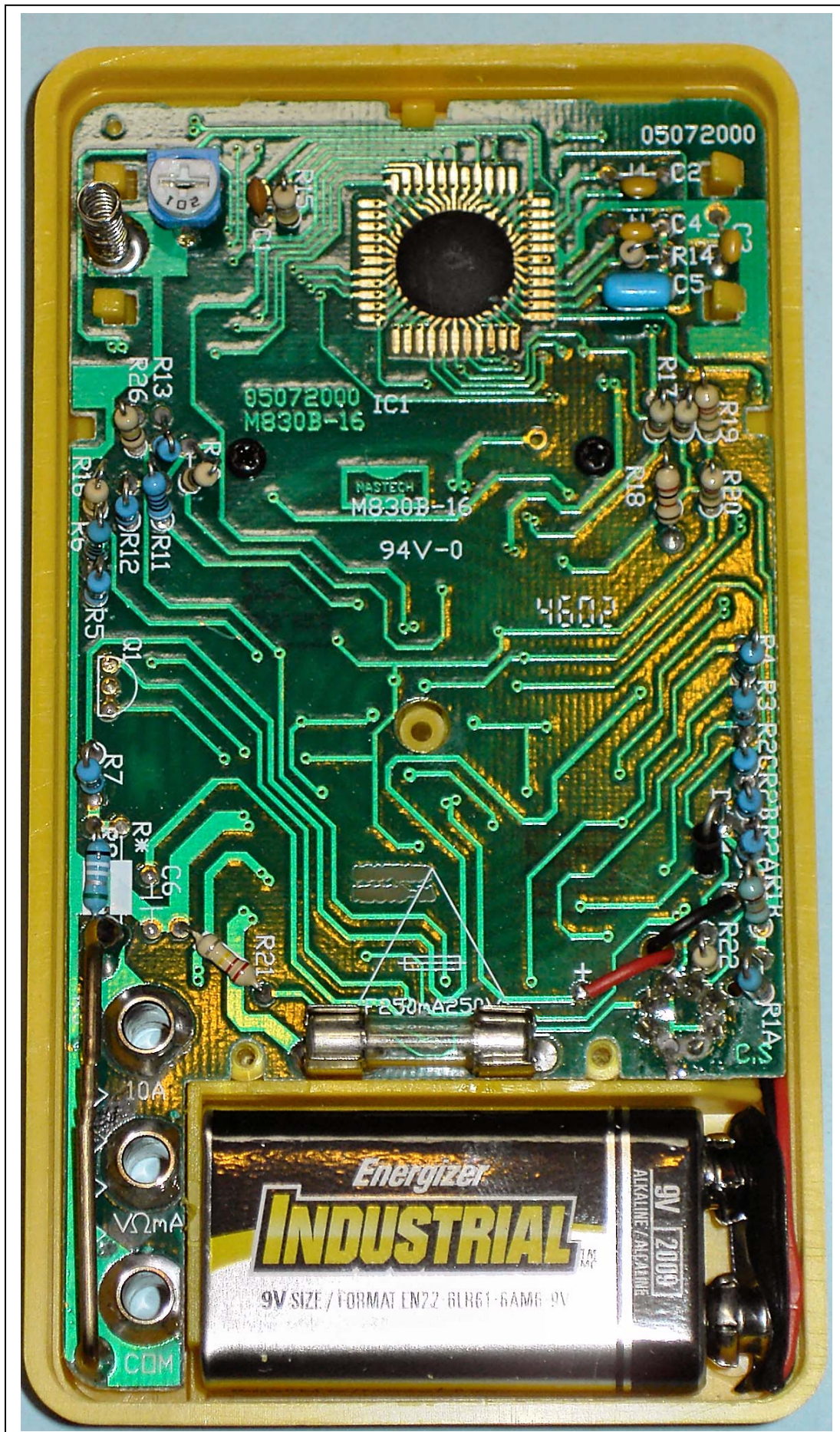
00.0



CEN-TECH

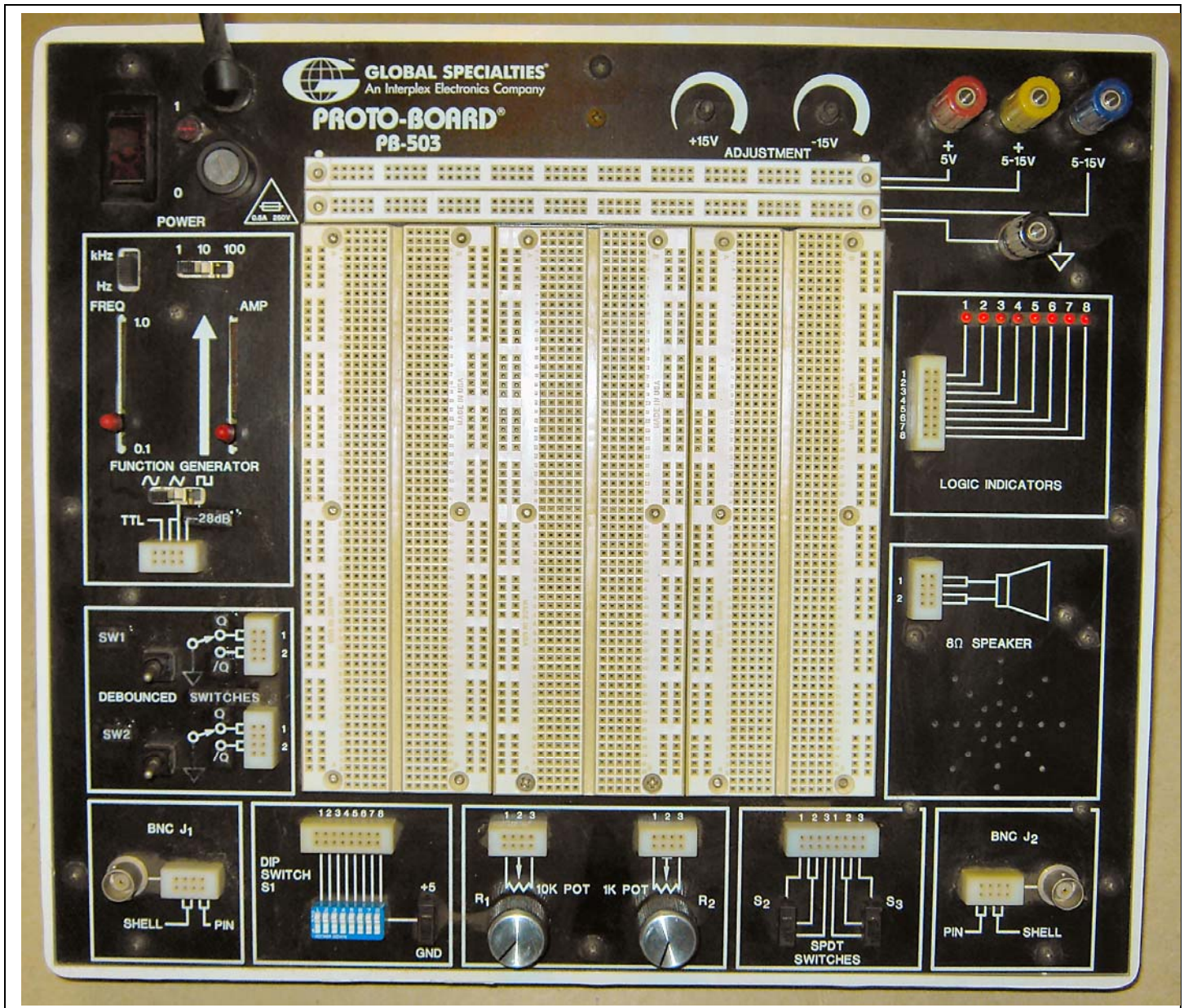
P30756





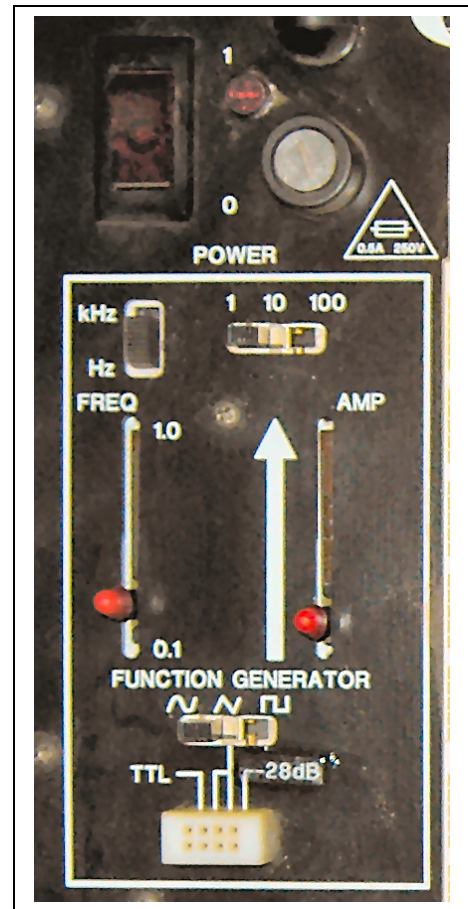
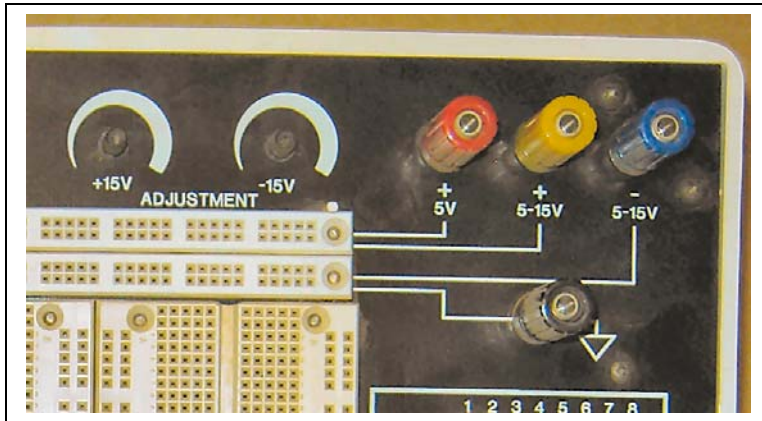
7. Exercises For The Analog Milliammeter and the Digital Multimeter

- a. The first thing we will have to do is to acquaint ourselves with what is called a "trainer". While these may come in many shapes and many sizes, they are all somewhat similar. All of them have a power supply of some sort, generally a fixed 5 volt supply and a variable positive and negative supply. Most of them have an audio generator ("function generator"). This is a typical trainer:



- b. What we will do to start this exercise is to set the positive supply to 12 volts so that we can use this supply for the rest of the exercise.
1. Set the digital multimeter to read dc voltage (DCV) on the 20 volt full scale range.
 2. Plug the trainer in to a 110 volt AC wall outlet.
 3. Turn the power switch on

- Note that the power switch has an internal light when the trainer is turned on



- Connect the black (negative) lead from the multimeter to the black binding post on the trainer. Note that the black plastic ribbed insulator on the top of the binding post is threaded and there is a hole that the multimeter can neatly pass through. Tighten the threaded insulator down to hold the black multimeter lead in place.
- Touch the red multimeter lead to the red binding post. It should read 5 volts plus or minus a tenth of a volt (4.9 to 5.1 volts)
- Touch the red multimeter lead to the yellow binding post. It may read anywhere from 2 to 20 volts. Turn the "+15V ADJUSTMENT" from one stop to the other and note that the voltage will go anywhere from below 5 to above 15 volts. Caution, this power supply can put out up to one ampere and destroy delicate components if improperly connected. Adjust the + supply to 12 volts on the multimeter.
- Just to see what happens, repeat the same procedure on the blue binding post. What happened to the readout on the digital voltmeter (i.e. what was the difference between the reading on the yellow and the blue binding posts).

9. Take the precision milliammeter from its box. Note that the meter face says it is a 0 to 1 milliampere meter. CAUTION -- in the real world, I could have purchased this meter with any printing I wanted on the face from volts to ohms to degrees Fahrenheit to Roentgens to square squids. However, since I measured these meters for myself, I know for a fact that they are a 1 milliamper meter.
10. Turn the meter over and look at the back. Note that the left brass contact thread has a small (+) plus sign to the left. By process of elimination, the right contact must then be the (-) minus contact.
11. Note that there may be a couple of small black cylinders across these terminals. These are "stupid diodes" that may prevent me from destroying the meter movement if I do something really stupid (don't count on them, though)..



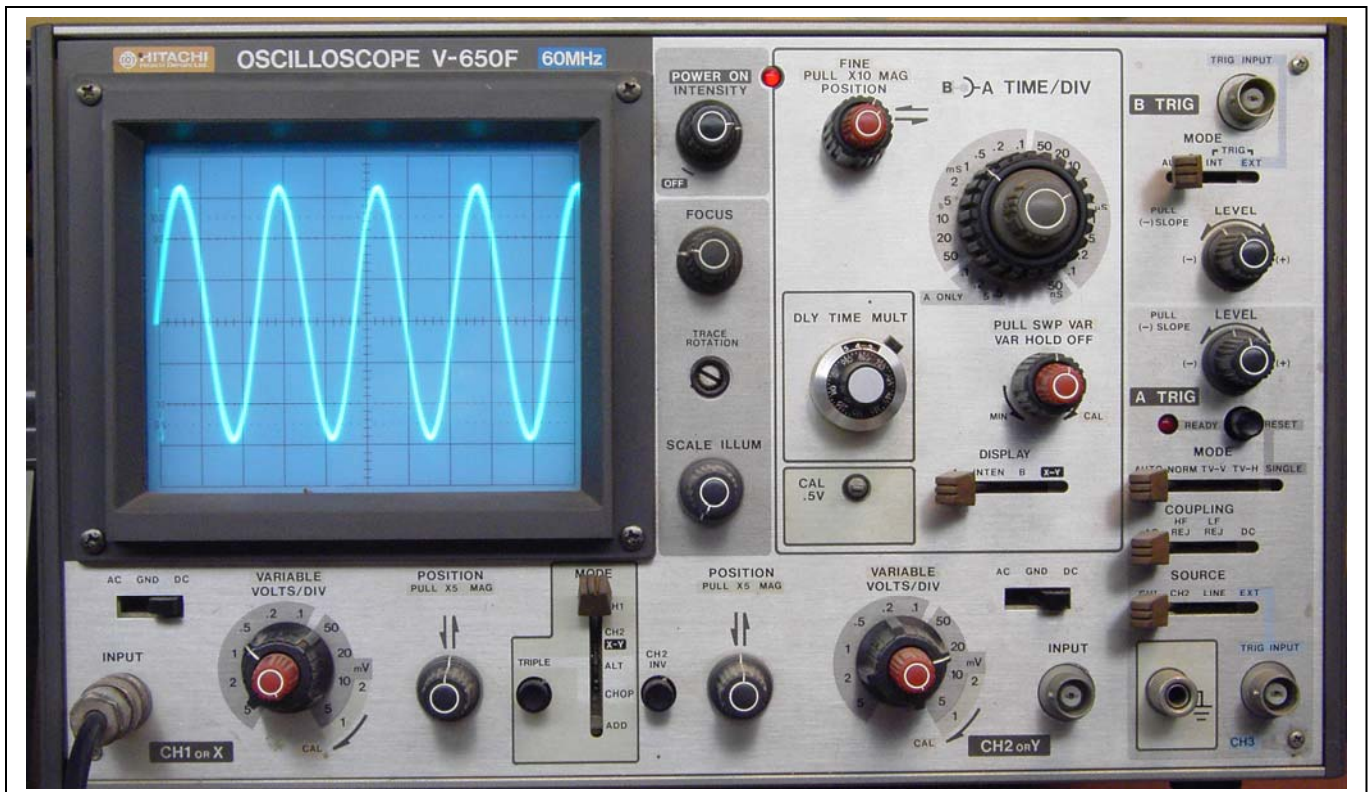
12. Let's make a 15 volt voltmeter out of this milliammeter, then we'll measure the 12 volt supply we set up on the trainer.
 1. Have the lab technician get you a FOUR BAND 15,000 ohm (15k) resistor. Note that the four bands reading from left to right are brown, green, orange, gold. With the chart in the room, interpret this resistor as a (brown = 1), (green = 5), (orange = 3 zeroes), (gold = 5% tolerance) or 15,000 ohm 5% resistor.
 2. Measure the resistor using the digital multimeter. First disconnect the multimeter from the trainer. Then set the multimeter selector to " Ω 20k". Then connect the multimeter leads across the resistor in either direction you choose -- a resistor does not have "polarity" (or a plus and a minus side). Record the value you read from the meter. Calculate whether or not this resistor is within tolerance. (A 15k 5% resistor may be anywhere between 14250 and 15750 Ω).

3. Use a cliplead to connect the (-) minus terminal of the meter to the black ground or common binding post on the trainer.
 4. Note that each of the colored binding posts on the trainer have a white line extending from the binding post itself to one of the horizontal lines of holes across the top of the trainer. Any of the holes in the line associated with the corresponding binding post is connected to that binding post. For example, any hole in the top row of four rows is connected to the +5 volt supply, the next line down is connected to the + variable supply, the next line down to the - variable, and the next line down to ground, or common.
 5. Insert one lead of the $15\text{k}\Omega$ resistor into any of the + variable supply holes. Connect a cliplead to the other end of the resistor.
 6. Stop here. Call the instructor or lab tech over. Have them insure that your work so far is correct. With the instructor or tech watching, connect the free end of the cliplead to the (+) terminal of the meter.
 7. What did the meter read? What should it have read? (Hint -- what is $12/15$?)
 8. Take the digital multimeter and switch it from Ω to dc current (DCA) on the 2000μ scale ($2000\mu\text{A} = 2\text{ milliamperes}$). Remove the cliplead from the (-) terminal of the meter and connect it to the black lead of the multimeter. Similarly, remove the cliplead from the (+) terminal of the meter and connect it to the red lead of the multimeter. What did the multimeter read? How close were the analog and digital meters to reading the same value?
- c. Take twenty resistors, one at a time, mark down what the TRAY says their value is supposed to be, measure the resistor with your ohmmeter, and compare the color code with the chart on the wall. Try and become conversant with converting with ohms, kilohms (thousands of ohms) and megohms (millions of ohms).

B. OSCILLOSCOPES

1. Oscilloscope basics.

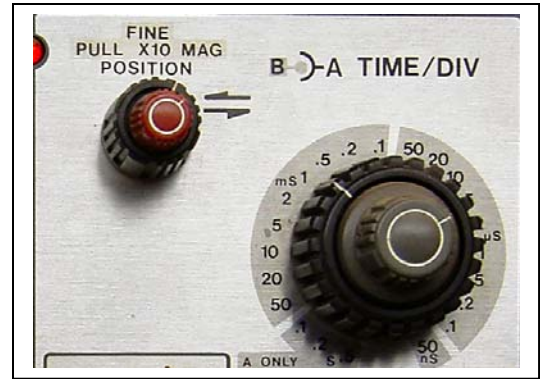
- a. An oscilloscope is basically nothing more than a voltmeter that measures voltage up and down as well as left to right simultaneously. It measures these voltages not with a pointer (which would be way too slow, as we shall soon see) but with a light beam that moves relatively instantaneously with applied voltage.
- b. It also has a whole bunch of doodads. It is the doodads that make the oscilloscope a lifetime job to learn properly.



- c. However, if we take the oscilloscope one section at a time, we may be able to get a fundamental understanding of how the 'scope works and may be able to use the major functions in a fairly short period of time.
- d. Let's go ahead and turn the oscilloscope on. Bring the intensity up until the "green worm" becomes visible and then use focus to make the line as sharp as possible. Note that if you have the intensity too high, you never will get good focus. Trace rotation makes the horizontal line parallel with the lines on the screen (it actually compensates for the earth's magnetic field). Scale Illumination is a backlight to make the grid easier to see.
- e. Normally, we use the Y (up and down) axis to measure our voltage and/or waveforms. We use the X (left and right) axis to measure time. While the voltage inputs are important, the time axis is the fundamental basis of the oscilloscope. Let's examine the time axis first.

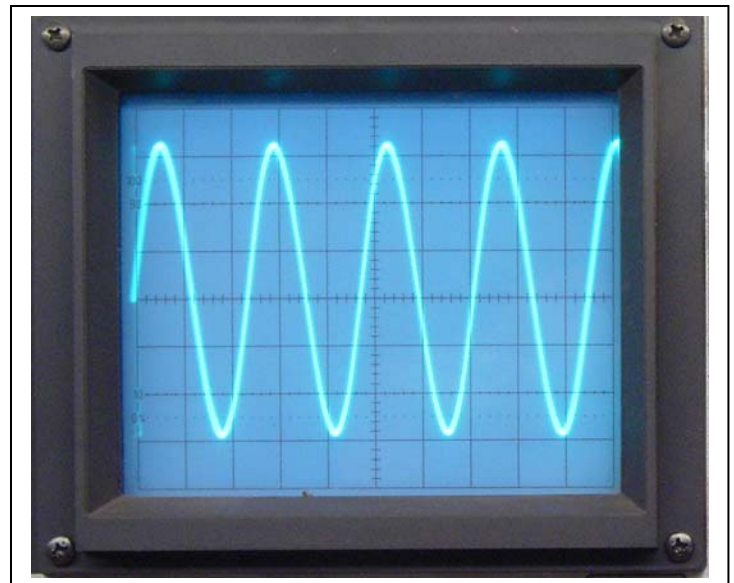


- f. Internal to the oscilloscope is a very well calibrated clock. This clock will measure from seconds to nanoseconds (a nanosecond is a billionth of a second) quite accurately. We use this clock to sweep the light beam across the screen in a precisely measured amount of time. For example, if I have the black outer knob turned to 1 mS then the beam will travel from left to right one millisecond for each of the little squares on the screen. This means that the beam travels from full left to full right on the screen in exactly ten milliseconds. We consider the return trace (the beam has to be sent back to the left side of the screen to start another "sweep" across the screen) to be instantaneous, or at least fast enough that we don't have to keep track of it.



The horizontal position (red and black knobs in the upper right hand corner of the picture) set the beam to start at the leftmost line on the screen.

- g. Now that you are an "expert" on the oscilloscope's "time base", let's do a little exercise. Here is a photo of the oscilloscope screen with the time base set as previously discussed. If the "period" of a waveform is when it goes from zero (the center line on the oscilloscope face) all the way positive, back through zero, all the way negative, and then back to exactly the same "zero" that it started from, what is this waveform's period in milliseconds?



If I give you further information

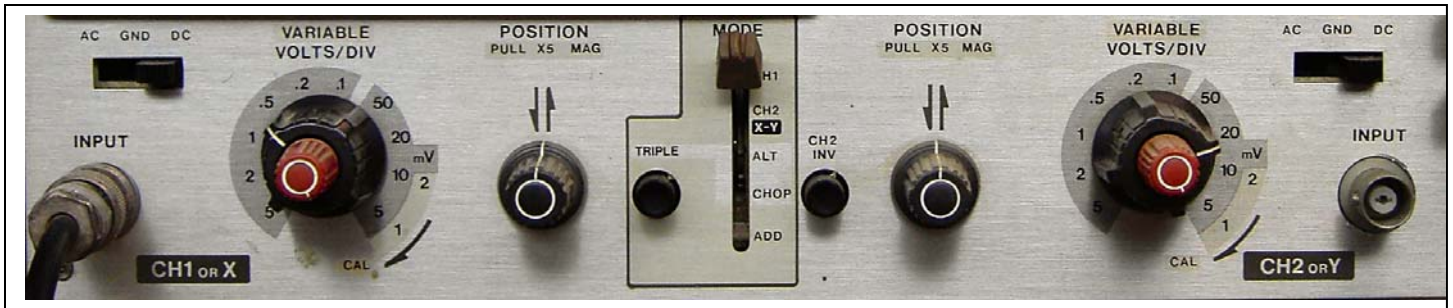
that a waveform's frequency is 1 divided by the period (or, in mathematical terms, $f = 1/t$ where t is the period in seconds) then what is this waveform's frequency?

- h. Let's move on to the vertical section. Most starter oscilloscopes are what are called "single channel" vertical input. Let's talk about this one first as though it were single channel. Moving from left to right we have the input -- sort of like the red and black leads of our multimeter, but for reasons that may not be apparent right now, we use shielded coaxial cable to bring our signals in to the vertical channel. Then we have a switch that allows us to measure DC or AC waveforms or to ground the input so that we can set the POSITION switch (far left) to where we want it on the screen. Finally we have the vertical calibration. In this case, I have

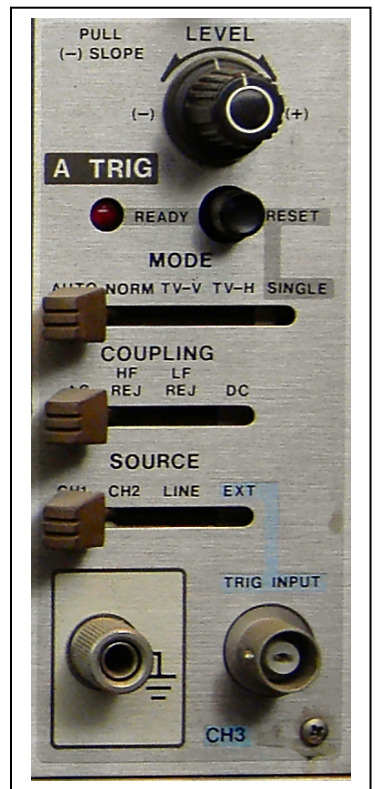


set the oscilloscope to 0.5 volts per division, and since the red knob is fully clockwise into the CAL position, each square on the face of the oscilloscope screen is exactly (well, pretty close anyway) 0.5 volts. Commonly referred to as half a volt per division. Question -- what is the voltage from the positive peak to the negative peak on this screen? If zero volts is the heavy vertical line from left to right halfway up the screen, what is the voltage from zero to the positive peak? To the negative peak (careful now)?

- i. A lot of oscilloscopes (especially those used for "real" in industry) have two or more vertical channels that you can use in various combinations. For right now, let's forget that channel two even exists and use the single channel that we have come to know and love



- j. Now the section that's gonna get you into trouble. Almost all modern oscilloscopes are what is called "triggered sweep". That is, they take a little sample of the signal that you are putting in and attempt to start the sweep from the left side of the screen at the same point on your waveform so that the screen isn't all jumbled up with traces starting at different times. If you will keep the MODE in AUTO, the COUPLING in AC, the SOURCE in whatever vertical channel you are measuring, and forget about external trigger, you are going to be just fine.



- k. Here's another gotcha-gonna-getcha. An oscilloscope PROBE is used to get those signals into the vertical input. A lot of probes have a 10:1 switch on them to enhance the frequency response of the probe. When the probe is in the 10x position, the frequency response is approximately ten times normal, but *the voltage is cut by a factor of ten*. You can use the "calibrated output" on most oscilloscopes to calibrate the vertical input AND check to see what kind of probe you have.



1. Oscilloscope Practice. Take half an hour to measure and observe the output of the function generator on the oscilloscope. See if you can correlate the frequency setting on the generator with the frequency that you calculate by measuring time on the oscilloscope. See what the various triggering options do for you. If there is one available, see what the sounds coming out of the function generator sound like (be respectful of others in the class who are doing the same thing).

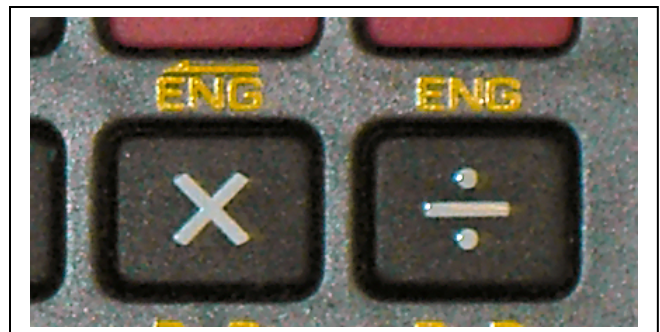
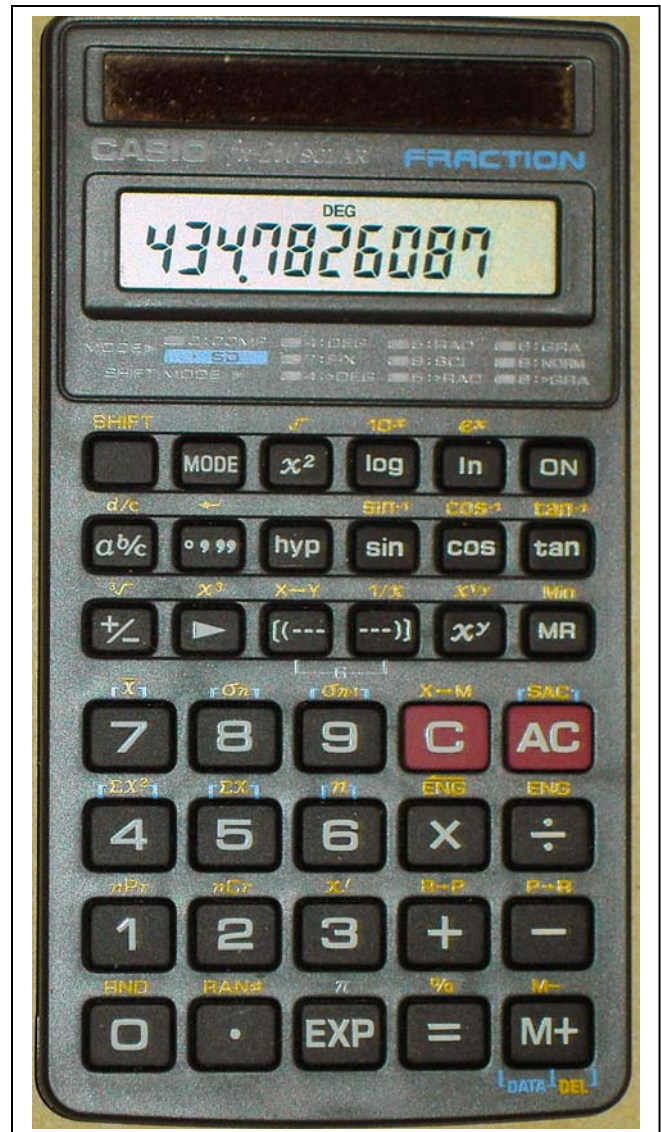
C. CALCULATORS AND THE ENGINEERING NOTATION

1. While a calculator isn't truly what we think of when we talk about "electrical instruments" it is enough of an instrument that we need to have some grasp of what it means to be able to use the calculator effectively. In essence, it is just another tool to the art of electronics.

I'd like to stop here for a brief moment if I may. I have absolutely no idea how many students I've counseled into buying the least expensive calculator that they could lay their hands on that will do the job. Most are willing to spend a semester's fees on a jolly whiz-bang box that will have the coffee perking when they get up in the morning. And, without a DOUBT they spend more time getting the wrong answers than the student that buys the \$8 calculator (shown at the right) that has just exactly what is needed, neither more nor less.

Nor is it possible to effectively use a calculator that has too FEW tools; the 99¢ Q-Mart special. In my opinion, and for the purposes of this class, your calculator should have:

- a. First and foremost, the ENG and reverse ENG functions so that you can go from kilosquirds to megasquirds to picosquirds with the push of a button. (More about this later.)



- b. Second, it's got to have the standard x^2 , square root, log, ln, 10^x , and e^x .

While it does NOT have to have the trig functions (sin, cos, tan, and all that) you will probably find those useful in your other electronics classes.



All together now, the trigonometry football team cheer:

Tan cos sec sin

3.14159

$e^x y^p$

Beer in the end zone, go team go. YAAAAAY.

- c. Third, the EXP function is absolutely mandatory. It is the EXP function that lets me input values in powers of ten. Thus, the only two ways I can enter a 1 Megohm resistor into a calculator is the long hard way 1000000 (which leaves tremendous margin for missing a zero) or 1EXP6 which gets it nailed exactly.
- d. The other normal functions (add, subtract, multiply, divide) come with any calculator. However, one sneaky function that I like about this parTICular calculator is that it is solar powered. No more running out of batteries in the middle of the exam.



2. Engineering notation and prefixes.

In the engineering world, we live on what are called "powers of three". That is, the difference between mega and kilo is a thousand, which is ten to the third power. The difference between giga and milli is a trillion, which (besides being a lot of zeroes) is simply ten to the twelfth (four times three). At some point in time, you will have this committed to memory. Don't worry if you have to look at a chart for a few months or years. You'll eventually get it.

By the way, this is what the ENG key on the calculator lets you do.

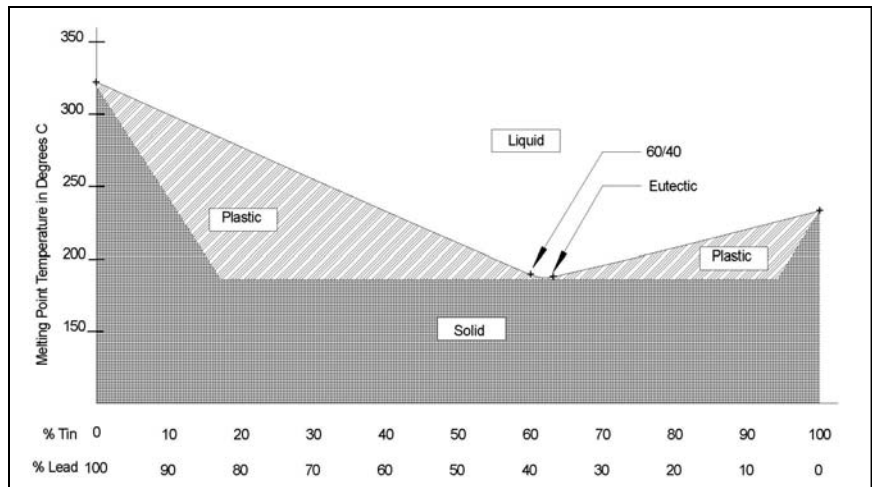
A "squird" is a purely made-up word for the purposes of illustration. Here you go:

	(THE LIST GOES ON)	
.0000000000000001	FEMTO- SQUIRD	10^{15}
.0000000000001	PICO-SQUIRD	10^{12}
.000000001	NANO-SQUIRD	10^9
.000001	MICRO-SQUIRD	10^6
.001	MILLI-SQUIRD	10^3
0	SQUIRD	10^0
1000	KILO- SQUIRD	10^3
1000000	MEGA- SQUIRD	10^6
1000000000	GIGA- SQUIRD	10^9
1000000000000	TERA- SQUIRD	10^{12}
	(THE LIST GOES ON)	

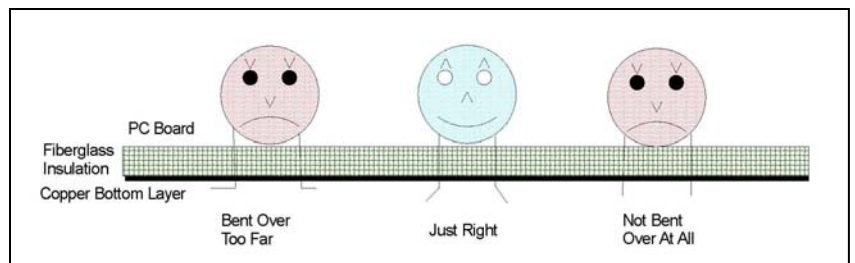
IV. ELECTRICAL ASSEMBLY TECHNIQUES

1. Soldering

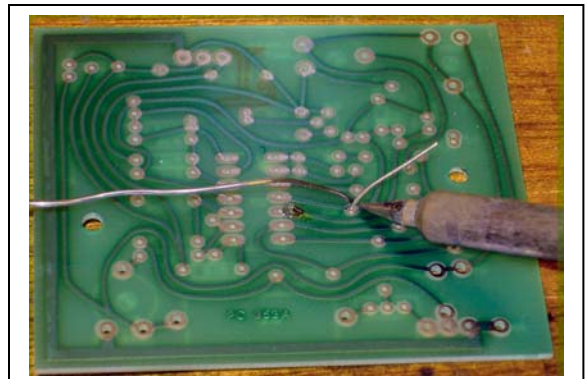
- a. Soldering is more of an art than a science. When we want to join two metal parts together electrically, we use an alloy of tin and lead called "solder". While pure lead melts at 320°C (620°F) and pure tin melts at 232° (450°F), the alloy melts at a much lower temperature. The "eutectic" point (lowest melting point of an alloy) happens at a tin/lead mixture of 63% tin, 37% lead at a temperature of 183°C (361°F), but most common solder is alloyed at a 60-40 mixture that melts at 188°C (370°F).



- b. Eutectic goes from the solid to the liquid state directly. All other alloys go through a "plastic" state where the solder is an unhappy blend of both solid and liquid. If the joint is allowed to move, even slightly, during this plastic state, when the solder moves to the solid state the joint will have cracks and other voids called a "cold" solder joint. Cold solder joints are responsible for a lot of unreliable equipment. Just remember that **solder makes a poor glue** and to make sure that the metal parts to be joined are mechanically rigid before trying to solder them.
- c. Once mechanically rigid and soldered together properly, the joint is usually difficult to take apart for repair. We can make it a little easier to repair during the construction phase by only bending the lead over on the pc board as far as is necessary to keep the part mechanically secure (generally a 45° bend of the component lead). Bending it over at right angles before soldering almost guarantees that the part AND the pc board pad come off as one.

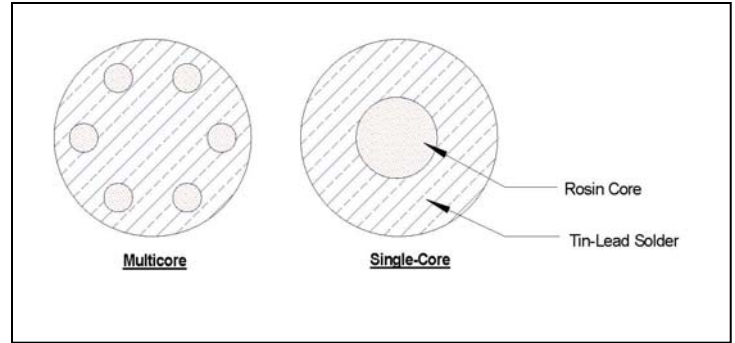


- d. To solder a joint well, it will be the joint itself that melts the solder. Put the soldering iron on one side of the joint and feed the solder in from the other side. When the joint gets hot enough to melt the solder, you can generally assume that the joint was properly made. Here you see a photo of the iron coming in from the right side of the picture and the solder feeding from the left side. The iron is



touching both the component lead (bent over at a 45° angle) and the pc board pad. Note the well made joint to the left of the joint being done.

- e. Electronic solder isn't just tin and lead. "Rosin core solder" is actually a tube of solder with a "flux" inside the tube, or core. This flux is made from a variety of pine sap called rosin. This rosin core chemically reacts with the metals that you are soldering together and (when heated) cleans any light corrosion or oxidation from the joint you are making. Do NOT expect the rosin to take off half an inch of accumulated grease and dirt; preclean the parts with a copper cleaner if they are filthy



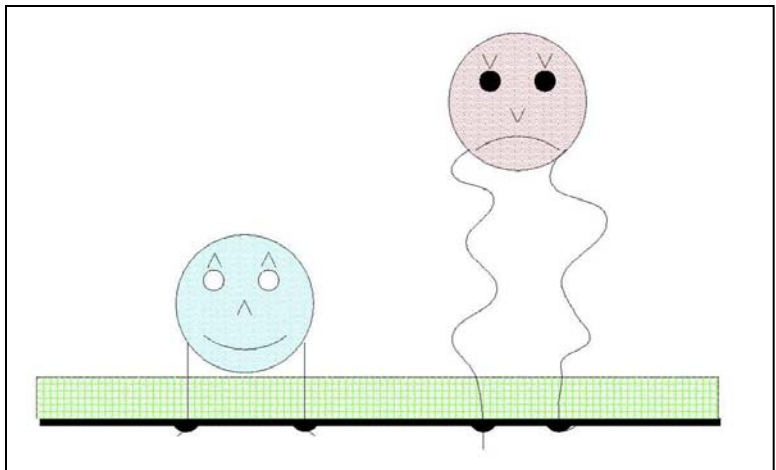
Some people prefer to take the rosin flux off of the board when the board has been soldered and some just let it be. There is argument that the used rosin is hygroscopic (takes water out of the air) and will eventually corrode the board. Other say that there are electronic devices made before World War II that are functioning just fine with the original rosin still on the joint.

Rosin is soluble in isopropanol (plain old rubbing alcohol) and ethanol and is also soluble in some halogenated hydrocarbons like freon -- but the hydrocarbons are not recommended for shop use.

Caution -- A lot of components are also soluble in the alcohols and hydrocarbons. Dipping the board in a flux stripper solution may take off the flux, but it may well take off some components as well.

- f. Here are the steps to making a successful joint:

- i. Get the soldering iron reasonably hot. 315°C (600°F) is a good place to start.
- ii. Once the iron is warm, brush it a couple of times on a water-wet sponge to remove the oxidized solder and flux
- iii. If the components or board are dirty, clean them with an abrasive cleanser, water, and a kitchen pot scrubber.
- iv. Put the component into the pc board and be sure that it is down as reasonably far on the board as it will go.
- v. Bend the component leads over at a 45° angle
- vi. Touch the soldering iron to both component lead and PC board pad



at the same time.

- vii. Feed the solder in from the other side and let the pad/component lead melt the solder
- viii. Do not let the joint move while the solder is cooling.
- g. Removing parts is far more difficult than putting the parts on in the first place. There are several tools that we can use, but in the small shop there are generally two tools that we have available to us for this purpose.
 - i. A "solder sucker" is a spring-loaded vacuum plunger with a teflon straw on the end of the vacuum chamber. The joint is heated to melt the solder, the teflon straw is placed close to the molten solder, and the "trigger" on the vacuum chamber is pressed. The resultant vacuuming action will suck the molten solder up into the vacuum chamber where it solidifies.
 - ii. Solder wick is nothing more than very clean copper braid impregnated with a good dose of rosin flux. In general, solder tends to have an affinity for small copper wire mesh and that affinity is helped along by the large dose of flux that the braid carries with it. We first use the solder sucker to take off the big lumps of solder and then clean up the joint with solder wick.
- h. Integrated circuits are somewhat difficult to remove from the board without removing pads and traces. In industry, special irons that heat all the pins of the IC at once and spring-loaded clips that put a constant tension on the package are used. While those special tools are generally not available in a small shop, judicious use of a small screwdriver, solder suckers, solder wick, and careful heating of pin by pin will sometimes let you save the board while still removing the IC. It depends very much on how well you have cleaned the solder from the joint and hole of the IC pin.
- i. Another technique that finds widespread practice is that of clipping the IC out of the circuit while leaving the pins attached to the board. This destroys the IC that you are trying to remove, but unless the IC is tremendously expensive, your time in removing the IC whole far exceeds the cost of most ICs. The replacement technique can be done sloppy and quick or neat and slow. You can solder the replacement IC pins directly to the leftover stubs of pins on the top of the board or you can remove the old pins one by one and then insert the replacement IC into the vacated pin holes.

2. Solder Practice

- a. Take a "junk" PC board and 6 "junk" parts. Be sure that one of the junk parts is an integrated circuit with at least 8 pins. Solder each of the 6 parts to the PC board.
- b. Now swap YOUR PC board with another student. Remove the components that the other student put on their board.

V. COMPUTER TOOLS

1) ENGINEERING DRAWINGS

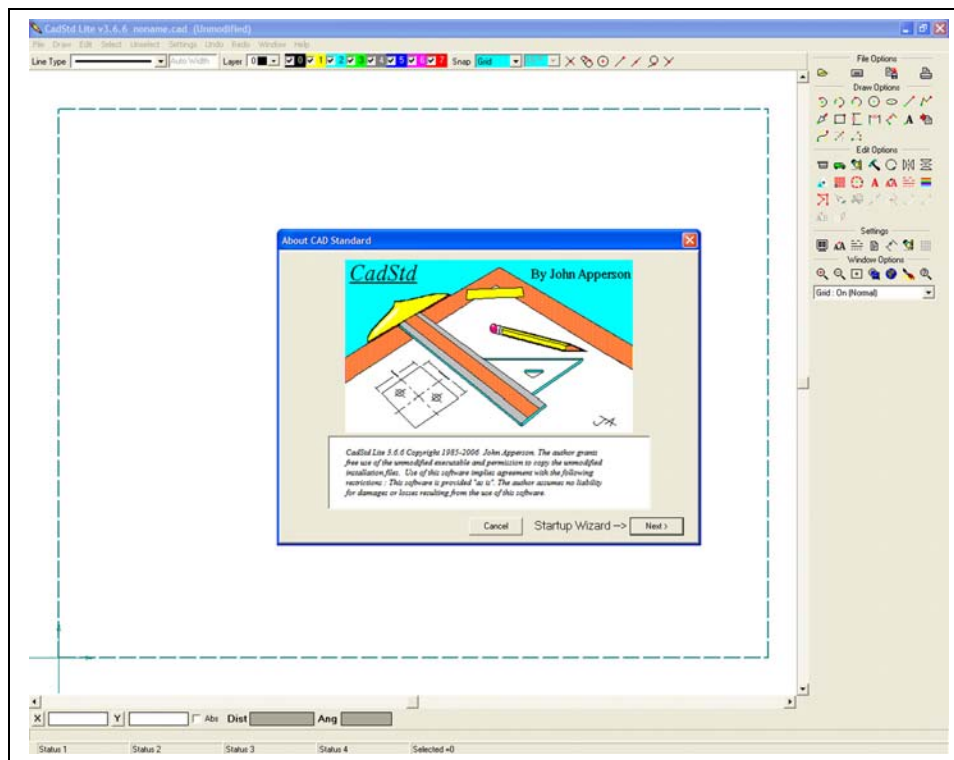
(1) Modifying Existing Drawings

- (a) Computer Program -- "CadStd", abbreviation for "Cad Standard".
- (b) If CadStd is not loaded, you will find the installation program in the "Software" folder on this CDROM. The program is cslte366.exe. It will load into Program Files on the default hard drive (C:).
- (c) If there is not already a shortcut on your computer's desktop to CadStd, drag one from the start menu onto your desktop. You may, if you wish, start your own personal drawings folder on your desktop.
- (d) The program should be on this CDROM in the Software folder . If it is not and you want one for your home computer, they are free for the download at www.cadstd.com . It is completely freeware; there are no timeouts or cripples to the program with the exception of a couple of drafting goodies that it might be nice to have someday. The pay-for version is only \$30 if you want the extra goodies.

→→ Do not, under any circumstances, leave original work on the classroom hard drive when you leave for the day. These drives are subject to being wiped clean, changed from computer to computer, and in general are absolutely unreliable from class session to class session. ←←

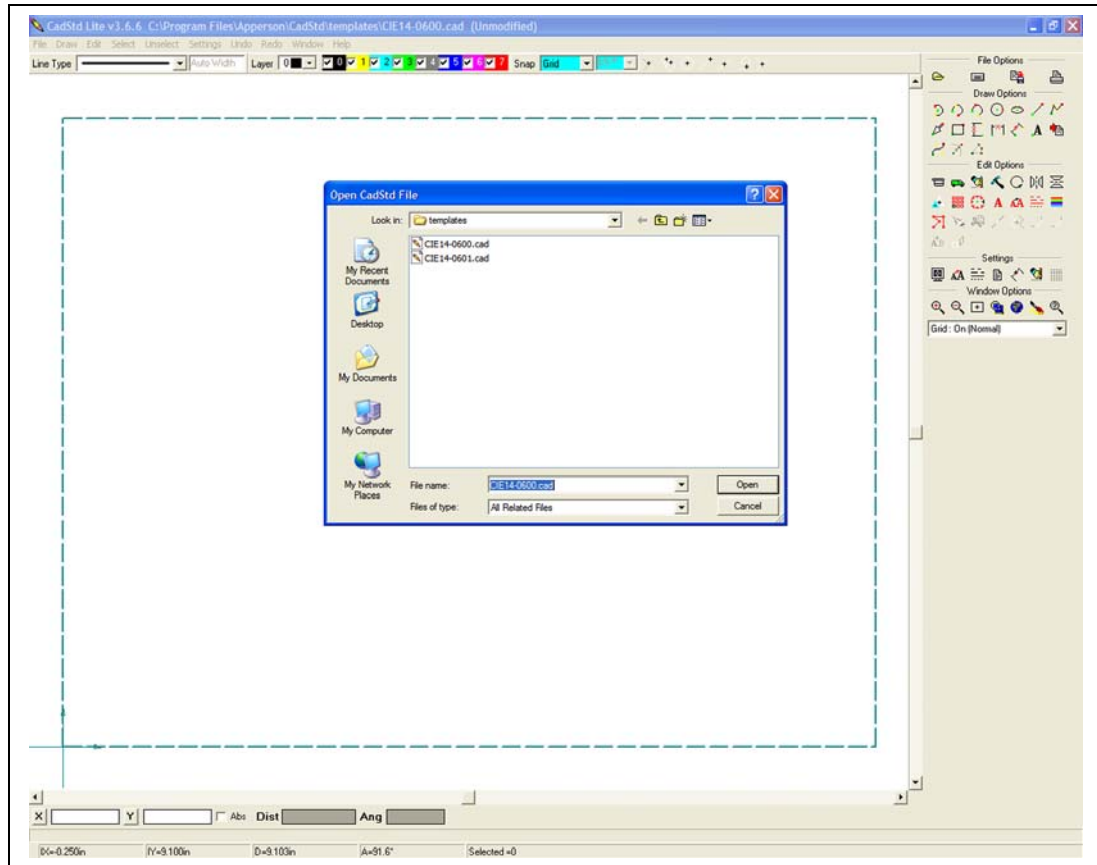
→→ Save your work on your own personal media - - flash drive, open-ended CDROM, or (as a last resort) floppy disk. ←←

- (a) Click on the CadStd shortcut icon on the desktop. It will pop up the opening screen as shown below:

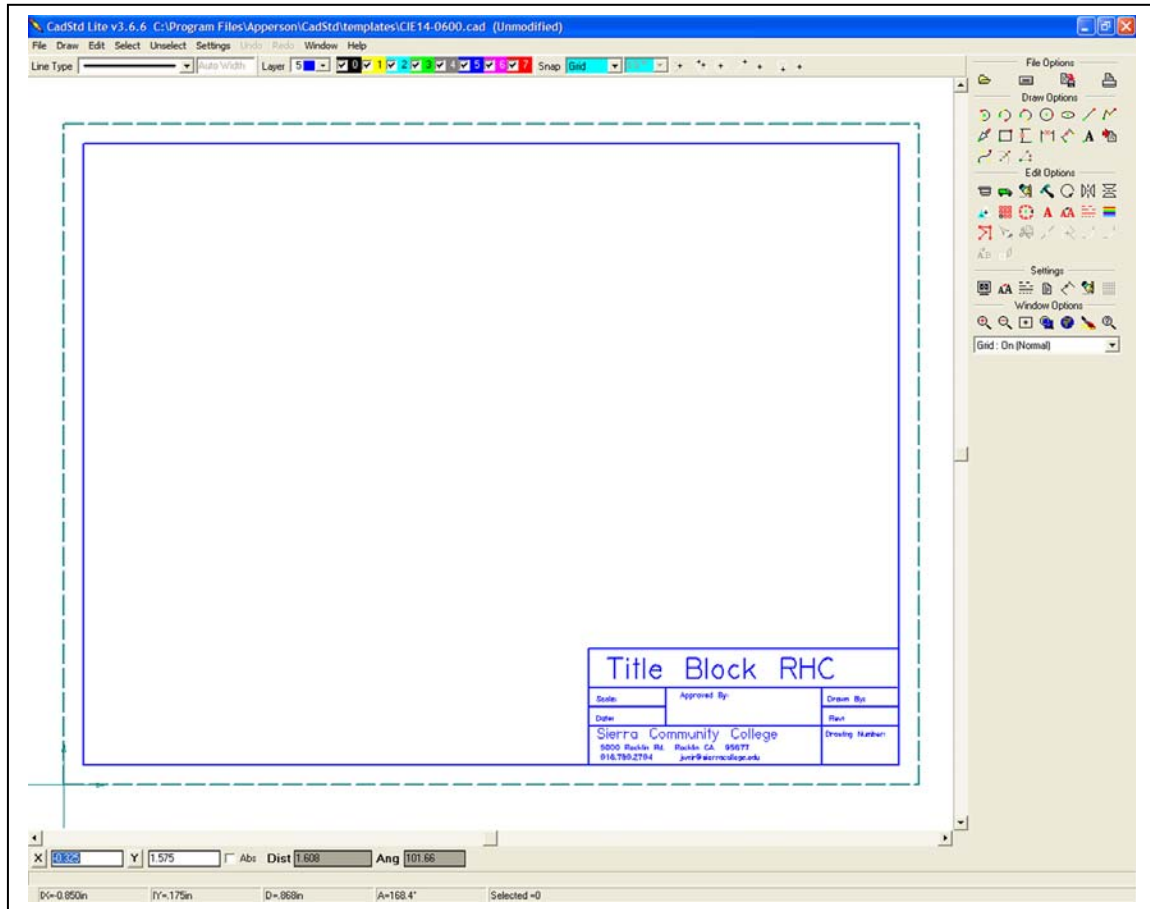


- (b) Choose the "Cancel" button.

- (c) From the "File" drop-down box in the upper left corner, choose "Open". It should take you to the Templates subdirectory. If it does not, arrow to the C:\Program Files \Apperson \CadStd \Templates subdirectory.



- (d) Select CIE14-0600.cad . It should load the standard drawing and title block "RHC" (Right Hand Corner".

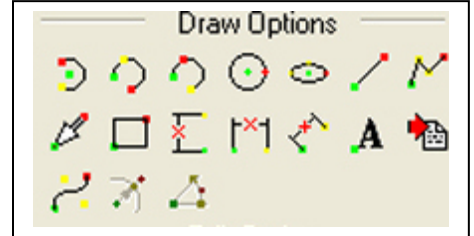


- (e) Before doing anything else, let's save this as your own personal drawing template. Go to File, Save As, and then "1234"-0600 where you replace "1234" with your own personal 4-digit code. This can be the last 4 of your SSN, the last 4 of your student ID, the last 4 of your telephone number, or any other 4-digit code you wish. For example, the last 4 digits of my phone number are 2203, so I'd save the drawing as 2203-0600

- (f) Now we can get to work. Look on the right side of the screen. You will find a group of tools called "Edit Options". Among them is a single red A. Click on the red A and you can change any of the preprinted text on the screen. Change everything in the Sierra Community College to your own name/business and address - phone - email entries.

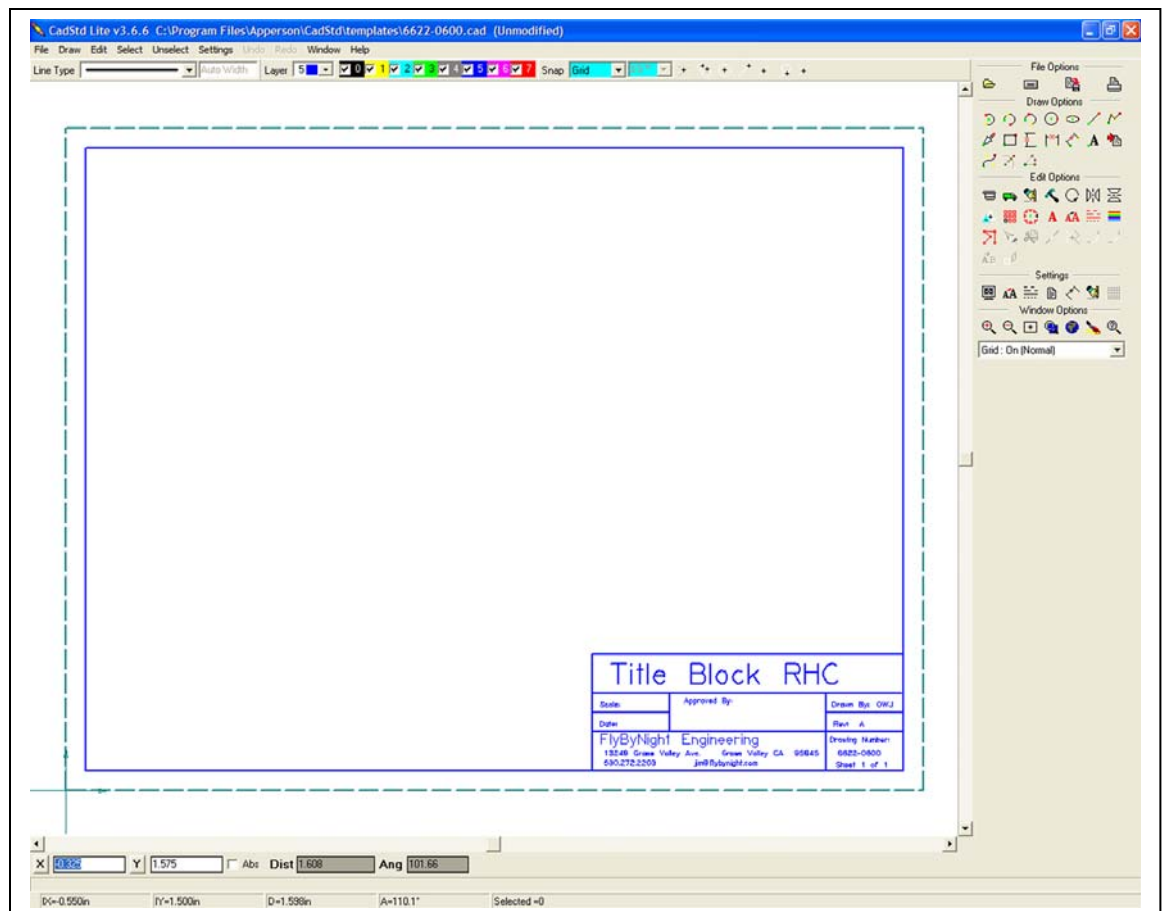


- (g) Now we want to INSERT some text. From the Draw Options tools, pick the black A with the green dot in the lower left hand corner. Insert your drawing initials into the "Drawn By", the letter A into the "Rev" and "1234"-0600 Sheet 1 of 1 into the Drawing Number. Please note that there is a particularly nasty glitch in CadStd when doing lettering in that you have to reset the "angle" measurement manually to zero with each text entry.



Dist 1.608 Ang 101.66

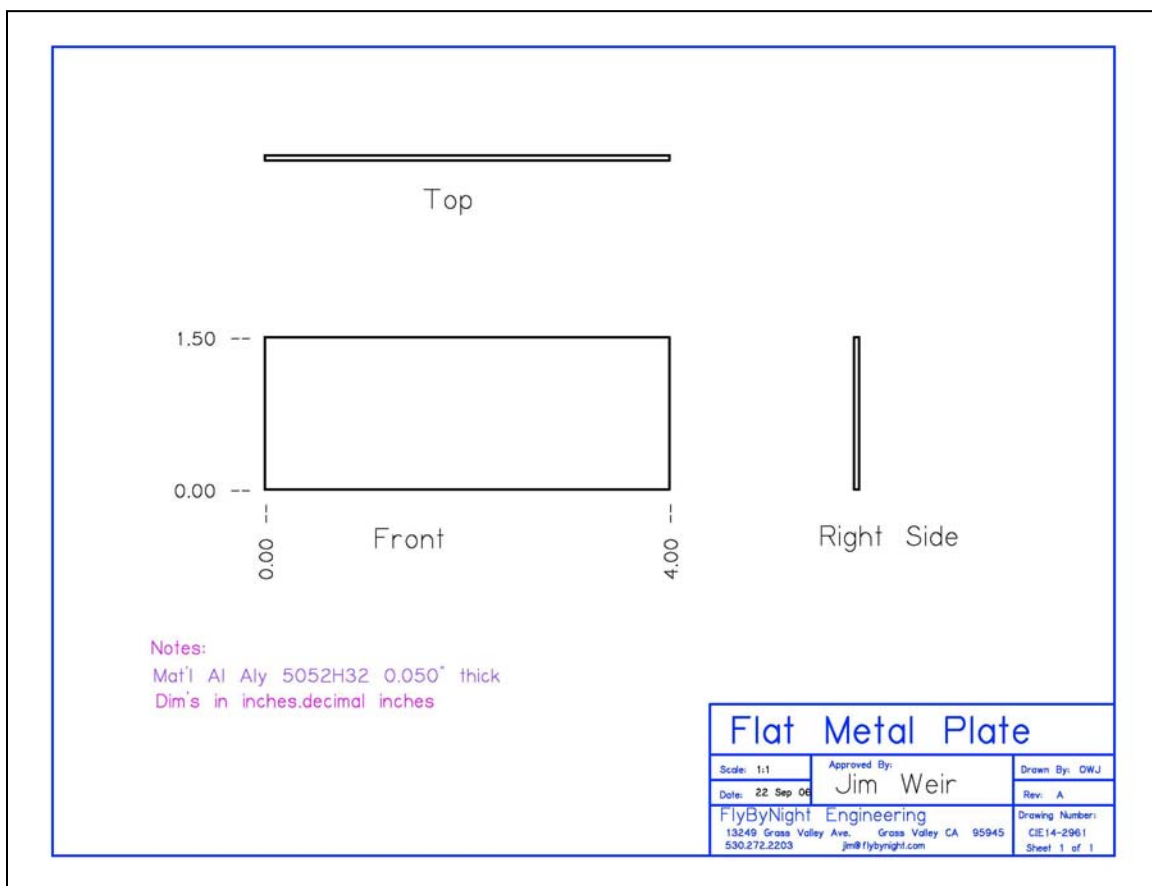
- (h) The final drawing you should see looks a lot like this (Don't forget to save your drawing a LOT during the drawing. There is no "autosave" in this program.)



- (i) Now that you know how to change an existing drawing, remake the CIE14-0601 and turn it into your own drawing "1234"-0601. When you are satisfied with your drawings, save them one last time and then go to MyComputer, find the Templates file (C:\Program Files\Apperson\CadStd\Templates) and make both of your personal drawings READ-ONLY. This will keep you from inadvertently over-writing your templates when you draw on them.

(2) Doing Original Drawings

- (a) When doing an original drawing, most often you will use the Draw Options and only use the Edit Options when you've drawn something you want to change. Let's use our "1234"-0600 template to make an old familiar drawing.
- (b) Open CadStd. Load up your template and then let's draw a piece of aluminum 1.5" x 4.0" (sound familiar?).



I'll bet you never thought you were going to see this little rascal again, right? Bzzzt.

- (c) We need to know a few things before we begin our drawing. First, each different part of the drawing needs to be on its own LAYER. LAYERS are color coded so that we can tell what part of the drawing is telling us what. Layers are also numbered from 0 through 7 The assignments for layers are as follows:
 - (i) Layer 0 - Mechanical (the actual drawing of the part)

- (ii) *Layer 1 - Construction (lines and figures that keep the drawing "straight")*
 - (iii) *Layer 2 - Unassigned*
 - (iv) *Layer 3 - Unassigned*
 - (v) *Layer 4 - Dimensions*
 - (vi) *Layer 5 - Outline & Title*
 - (vii) *Layer 6 - Notes*
 - (viii) *Layer 7 - Faceplate Screen Artwork*
-
- (d) Note in the Draw Options that there is an option to draw a rectangle. First draw the large 1.5" x 4" rectangle "Front" at some convenient spot on the drawing. Note that in the lower left corner there is a "counter" telling you how big the rectangle is.
 - (e) Now draw the two smaller rectangles (Right Side and Top) so that the edges of this rectangle are in line with the edges of the large rectangle. Don't worry if the small rectangles are close to the big rectangle; we can move the smaller rectangles with the MOVE command in the Draw Options.
 - (f) As a matter of fact, it might be a good idea for you to open the HELP file and click on Help. Here you have a list of all the commands that exist in CadStd. Some of them (like DIMENSION) you won't worry about except for the LEADER tool. Some of them, like move, copy, rotate, and all of the DRAW options you will be using on a regular basis.
 - (g) When you have the rectangles done, it is time to dimension the drawing. First use the text tool to get the 0.00-- dimension placed accurately at the lower left corner of the drawing. Then use the COPY tool to copy this 0.00-- to the upper left corner of the plate. Now choose TEXT EDIT to change 0.00-- to 1.50--. Use the copy command again to get another copy of 0.00--. Now use the ROTATE tool to rotate the text by 90° and place it at the lower left corner of the rectangle. Copy it to the lower right corner of the rectangle. Edit the text to read 4.00--.
 - (h) Now (he says with an evil grin) there are several drawings you did by hand in MechTools for the little bracket and the small chassis box. Redo them using CadStd.
 - (i) The goal is not to slap drawings together and finish them all in one sitting. Rather the goal is to get one drawing done perfectly and then let your lab partner(s)

finish one drawing perfectly. If we don't get them all done in one class, we don't get them all done in one class and that's the way it goes.

This is not a race against time; this is a race against perfect drawings. Finish one drawing show it to me on the computer screen, print it out on the color printer, then finish another one. If everybody gets ONE perfect drawing done today I will consider the class a success.

Complete the full set of drawings over the next couple of weeks when you get spare class time.

- (j) Here are the drawings to begin your work with (located in the MechTools Images directory):

(i) *CIE14-2701s1*

(ii) *CIE14-2701s2*

(iii) *CIE14-2201s2*

(iv) *CIE14-2201s3*

(v) *CIE14-2201s4*

(vi) *CIE14-2201s5*

2) Circuitmaker & Traxmaker -- Printed Circuit Boards

- a)* A “printed circuit board” (“pcb”) starts out life as a thin sheet of epoxy-fiberglass to which is laminated a thin foil of copper. If this copper is only on one side of the board, the material is referred to as “single sided”. If there are two sheets of foil, one on the topside and one on the bottom side, the material is referred to as “double sided”.



- b)* A pattern of conductors (called “traces” or “tracks”) connect the various components in the circuit. The components are placed on the pcb in holes drilled through the copper-fiberglass sandwich and soldered to the traces. This is called “through-hole” construction. Another technique called “surface mount” solders the components directly to the copper traces without the need to drill holes.

- c)* The pcb process proceeds like this:

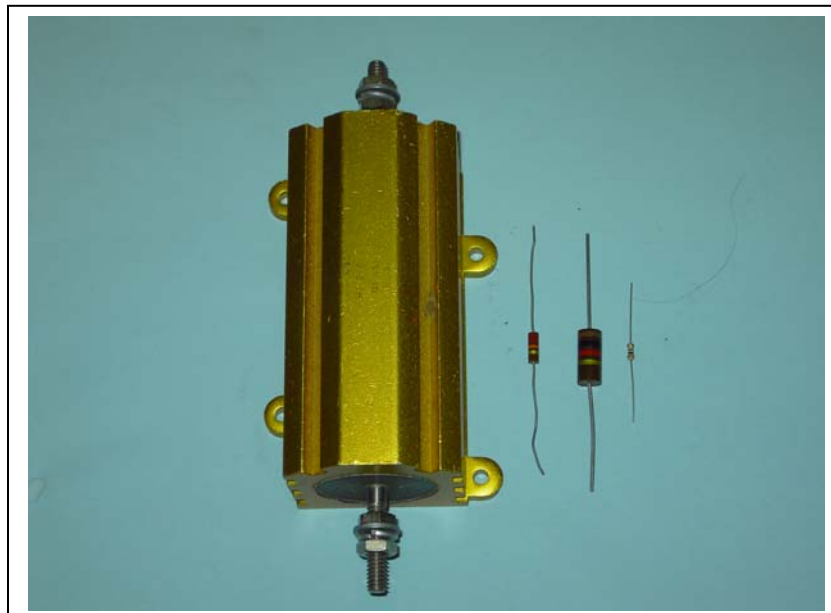
- (1) The electronic circuit diagram (“schematic”) is drawn using a special computer program called a “schematic capture” program. The program that this course uses is called “Circuitmaker”. Unfortunately, Circuitmaker is no longer available to purchase, but the demo version included with this CDROM will be sufficient to produce the small pcb for the class project.
- (2) If Circuitmaker/Traxmaker have not been installed on your computer, go into the "SOFTWARE" folder, then the "SCHEMATIC-PCB" folder in this section. Execute DSDemo.exe. It will create two folders on your hard drive. One will be Circuitmaker and one will be Traxmaker. To get the special files we need for this class, copy all the EF-CM6Student files into the Circuitmaker folder and replace any files that are there with the files in the Student section. Likewise, copy/replace the Traxmaker files with the files in EF-TM3Student.
- (3) Prior to using Circuitmaker to lay out the schematic, a “library” of electronic components was generated that assigned each component a physical outline and component “pin” assignment. This is called the “footprint” of the component.
- (4) After the schematic diagram is generated and checked for accuracy, a “netlist” is generated in Circuitmaker and saved in the project file with the suffix .net. The schematic is saved with the suffix .ckt.
- (5) The pcb layout is produced in a companion program named “Traxmaker”. Traxmaker, too, has a library linked to the Circuitmaker library by means of the footprint assignment in Circuitmaker. When, for example, Circuitmaker uses the LM317 voltage regulator, it assigns it package TO-220V. When the netlist from Circuitmaker is

opened in Traxmaker, the TO-220V package is loaded along with the LM317 regulator. Traxmaker opens its library to see exactly what pins on the part are used and what the part layout should be.

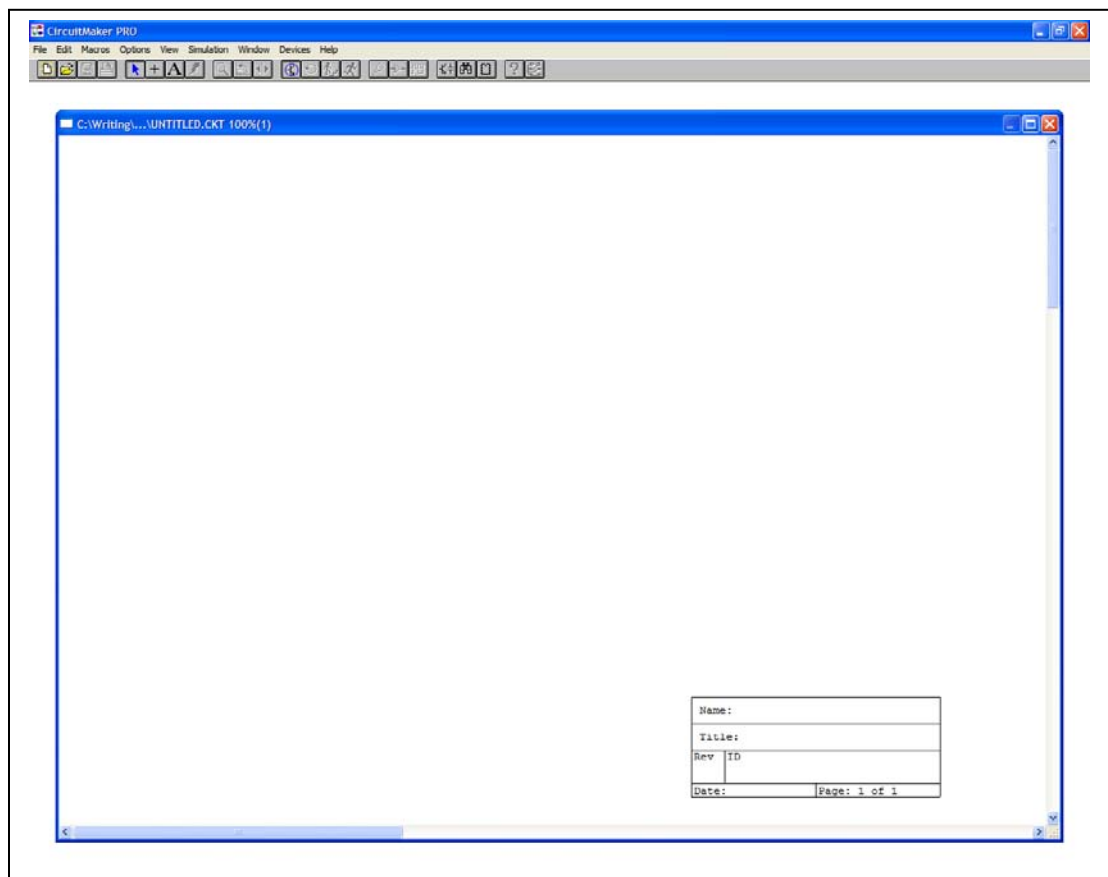
- (6) Not only is the footprint loaded along with the component, all the connections to and from the component are loaded in text format. ("Pin 1 of the regulator is connected to the cathode of D1 and pin 2 of R4.") These are called NETS (hence the name "netlist") and each net is one set of connections in the schematic.
- (7) While Traxmaker has a feature called "autorouter", for purposes of this class, we will manually route the board using the mouse and the computer to show the traces. Also, while the pcb material may be double-sided, the simple circuits in this class will all be routed on one side of the board.
- (8) Once the board has been routed, in production we would make a set of computer files called "Gerbers" and send them to our board manufacturer. The board manufacturer would use a photographic process to take these gerbers and make a pc board using a light-sensitive resist material and a camera.

d) Circuitmaker -- Drawing Schematics

- (a) A "schematic" is nothing more than a graphical representation of an electronic circuit. While a "pictorial" diagram, showing an actual picture or drawing of the part to scale, is one way of drawing a circuit, you can imagine drawing the pictorial of a complex circuit (like a motherboard) with all sorts of ratsnest wires connecting pins to pins and curling around each other. It would be impossible to read. For very simple circuits (like the battery, resistor, and LED light shown here) it isn't too difficult, but think about how big a simple hand calculator pictorial would be.
- (b) The schematic diagram does not take into account the physical size of the part. For example, the resistors shown here vary from a 200 watt 1 ohm resistor to a tenth of a watt 1 million ohm (1 MEGohm) resistor. However, they all have exactly the same schematic shape.

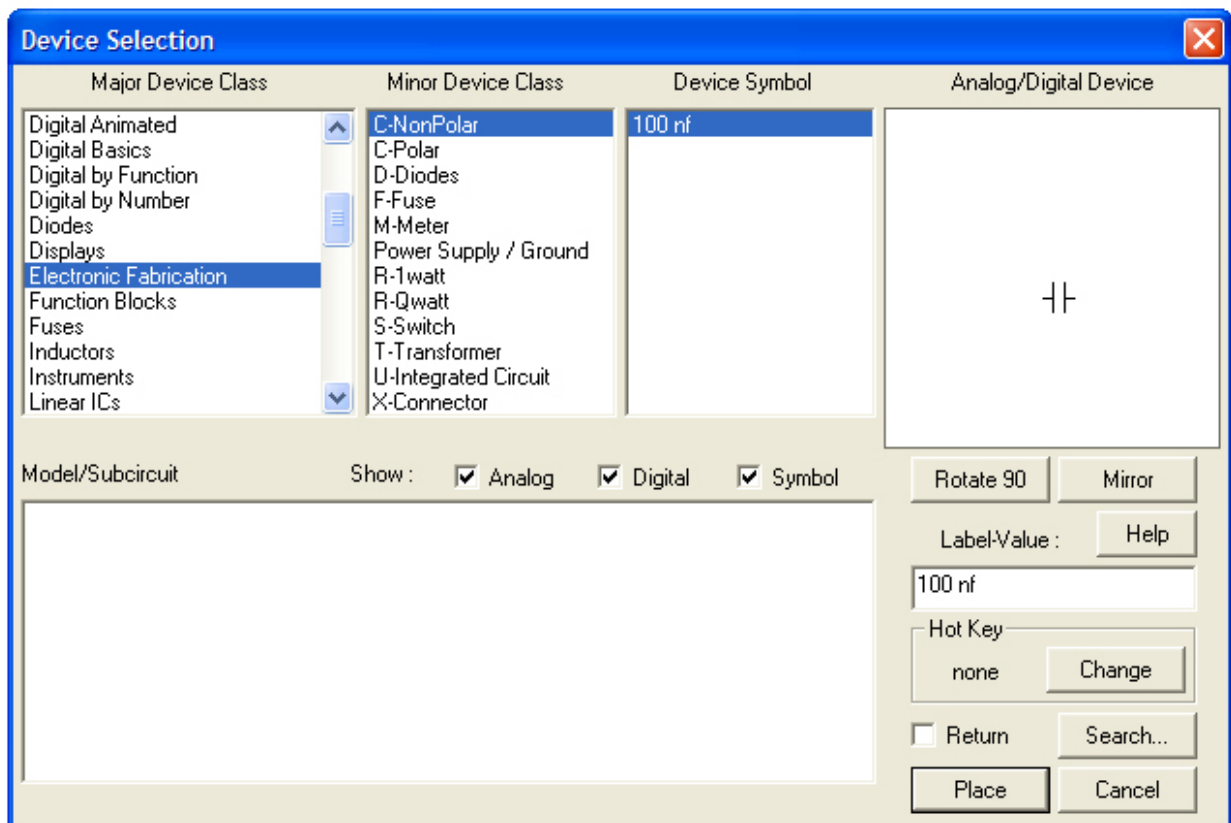


- (c) Let's do a very simple schematic diagram. Using the example from above, let's make a schematic diagram of a battery, resistor, and LED light that we will later transform into a small pc board, export the netlist to Traxmaker, lay out the pc board traces, print to regular printer paper the board, draw the traces by hand onto a piece of pc board material, etch the board, install the components, and then light the LED.
- (d) Circuitmaker should have been installed on your computer in the directory `c:/program files/microcode engineering/Circuitmaker Pro/cirmaker.exe`. If the installer was very kind to you, there will be a Circuitmaker icon on your desktop.
- (e) Open up Circuitmaker and you should be presented with a screen that looks like this.

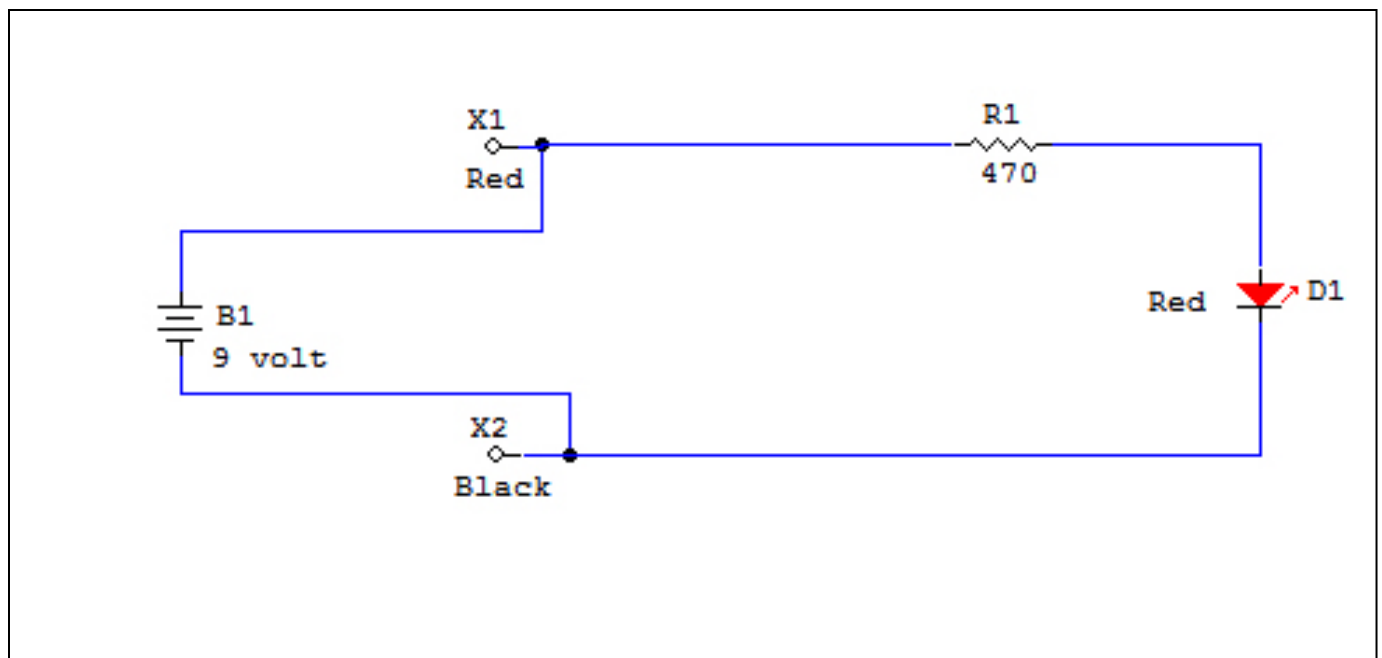


- (f) Up at the top of the screen there are 24 small button boxes, starting at the left with a sheet of paper with yellow marks around it and ending up on the right with a stylized pcb layout next to a question mark. Circuitmaker is sort of fussy, in that it will not save a blank file, so find the large letter "A" 7th button from the left and click on it. Now click in the UNTITLED.CKT and a shadow box will appear. Type your first name into the shadow box, then click on the arrow box two boxes to the left of the A. Your name should appear on the screen in the UNTITLED.CKT area.

- (g) The first thing to do, right off the crack of the bat, is to save this blank file UNTITLED.CKT to your own Circuitmaker file. Please use the file number 1111-5803.ckt where 1111 is your personal 4-digit identification code. Save it TWO places. Circuitmaker will default to MY DOCUMENTS. Make a folder under MY DOCUMENTS called PC BOARD 1111 (where 1111 is your personal number) and save this file into PC BOARD 1111 folder. Also save it to your removable media (flash drive or floppy disk) under the same number.
- (h) Let's get to drawing the circuit of our battery, resistor, and LED. There is a PARTS library button fifth from the right, just to the left of the binoculars. Click on that button and it will drop down a parts selection box. It will default to a Major Device Class ("parts library") called .General. Scroll down until you come to Electronic Fabrication. This is a library made especially for this class, containing only the components we are going to use in this class.
- (i) Note that there are twelve Minor Device Class categories starting with C-NonPolar to X-Connector. All of the components we are going to use this semester will be in one of these twelve parts bins. In particular, open up the parts bin marked R-Qwatt and you will find four resistor values, all of the Quarter Watt size. Select each of the resistors and notice that the schematic symbol does not change with value.

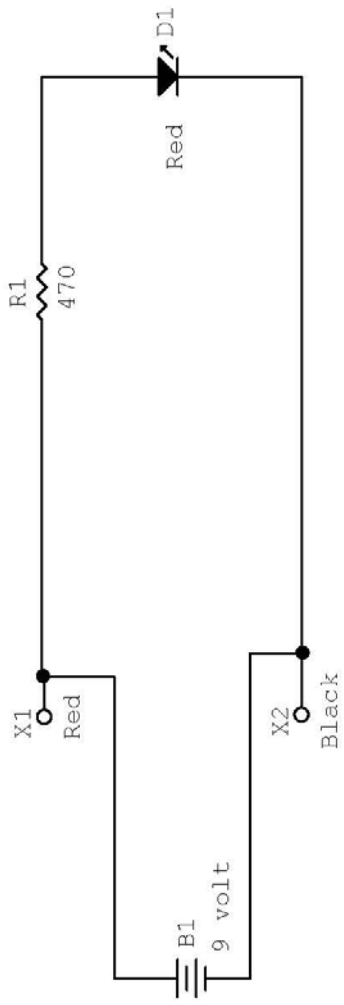


- (j) Select the 470 ohm resistor and then click on PLACE in the lower right hand corner. Notice that the resistor schematic symbol was placed on the schematic page but can be easily moved with the mouse. RIGHT click on the symbol and notice how it rotates from horizontal to vertical. Make the resistor horizontal and LEFT click to put it into the diagram permanently.
- (k) Save your .ckt file on the hard drive after this and every step from now on while we are using Circuitmaker.
- (l) Notice that it automatically gave it the designation R1. This is called the REFERENCE DESIGNATOR, and is one way we have of identifying this part. Now LEFT CLICK and HOLD on R1 and notice that we can drag it around the screen if we wish. Drag it and release the left button. Single click on it and notice that it turns red. This means that the part is in the edit mode. Up at the top of the screen in the button row notice that there is a rotating arrow symbol 10th from the left. Notice that you can rotate the resistor in steps of 90° for each click on the rotate button.
- (m) Double left click on R1 and it brings up an information box. Notice in the top it gives the label value (parts value) of 470 ohms (470Ω) for R1. It also gives the reference designator of R1. VERY important to us in a few pages is the Package (AXIAL0.5) because when we export the netlist to Traxmaker, this tells Traxmaker what shape to put on the PC board for R1. Because this is a relatively simple board we are going to make, checkbox by label value and the part value will be added to the symbol on the schematic. Click OK at the bottom.
- (n) Oops, it put the part value right on top of the reference designator. Not a problem. Select (left click) the part and drag the 470 down below the resistor.
- (o) Similarly, out of the D-Diode parts bin drag a LED 5mm onto the schematic and leave it vertical. Double left click on the LED and note that the value is missing. Type in Red or Green or whatever color we have in the stockroom into this location and check the box, then OK, then drag the color to the opposite side of the part.
- (p) Now things get a little more difficult. We aren't going to be able to fit a moosey 9 volt battery onto our small PC board, so we are going to have to wire a connector onto the board. We do this by leaving small circles of copper, drilling a hole in them, and running the wires (red and black) from the battery snap connector to the pc board. Place two X-Connector Wirepin symbols at the top and at the bottom of the schematic as shown. Put value Red in for the one at the top and value Black in for the one at the bottom. (these are the wire colors from the snap connector) -- disregard the message that Circuitmaker couldn't find a subfile. We don't use subfiles in this project.
- (q) Go to Power Supply/Ground and place a Battery on the page. Go into Edit Device Data (double left click) on the battery and be sure that Exclude From PC Board is checked at the bottom of the box.



- (r) This is almost starting to look like a real circuit, isn't it? Let's wire it up. In OPTIONS (at the very top of the page) be sure that Quick Connect is checked.
- (s) Put the mouse cursor right at the right end of connector pin and notice that there is a little red box over the end of the pin. This is telling you that Circuitmaker has found a place to put a wire. Left drag a wire from the right end of X1 over to the left end of R1. It is sort of sloppy looking, but let's leave it for the moment.
- (t) Similarly, connect the right end of R1 to the ANODE (arrow) end of D1. Connect the CATHODE (stripe or bar) end of D1 to X2.
- (u) Connect the (+) positive end of the battery to the blue wire connected to X1. Connect the (-) negative end of the battery to the blue wire connected to X2. Note that in both cases the program put a small "connection dot" where the wire from the battery connected to the wire in the circuit.
- (v) Now let's make it look pretty. Move the parts and wires around until all the wires are straight with no kinks in them and the wires don't run through any of the words by the parts.
- (w) Almost done, at least with the schematic. Go to Options (top of the screen), Title Block. Check "Display On All Pages". Give it name "Small LED Light". Erase "Title:". Give it ID 1111-5802, revision A, and today's date.
- (x) Go to File, Print Setup, Printer, and make it Landscape, Fit To Page, Black & White.

Jim



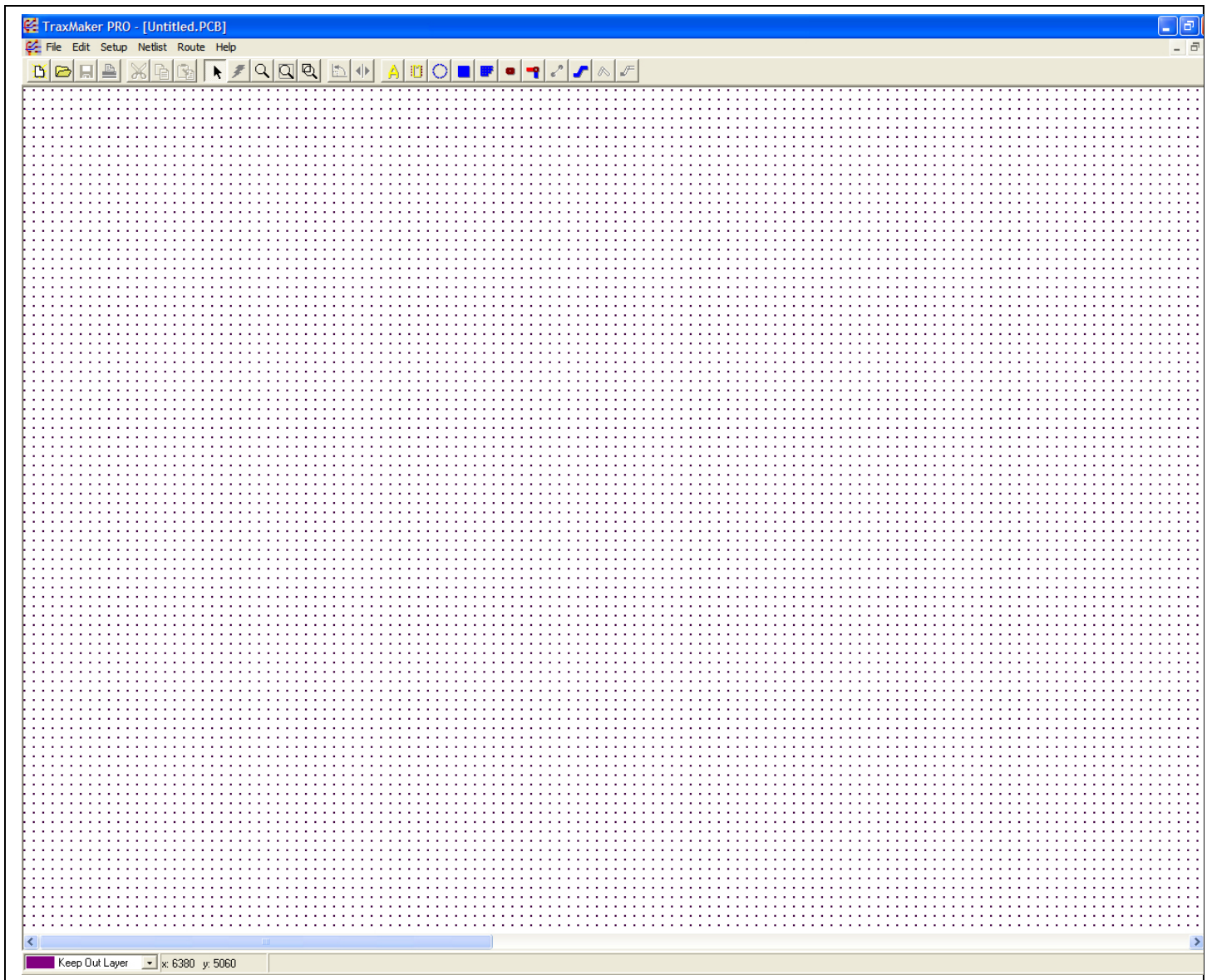
Small LED Light		
Rev	ID	
A	2203- 5803	
Date: 10 Oct 06		Page: 1 of 1

- (y) Select your name that you entered onto the screen when you first started this routine. Hit the delete key and it will disappear. Do a final save and print out your schematic.
- (z) Now let's prepare to export this circuit to Traxmaker. File, Export, PCB Netlist. This will bring up a screen that lets you export the netlist, make the PCB size, and bring up Traxmaker automatically. For the time being, Uncheck the box that says Run Traxmaker and load netlist. Then OK. It will ask you if you want to save the file under the same name as the Circuitmaker file, only as a .net (netlist) file. Save.
- (aa) Now save both the .ckt file and your new .net file to your flash drive or floppy disk.

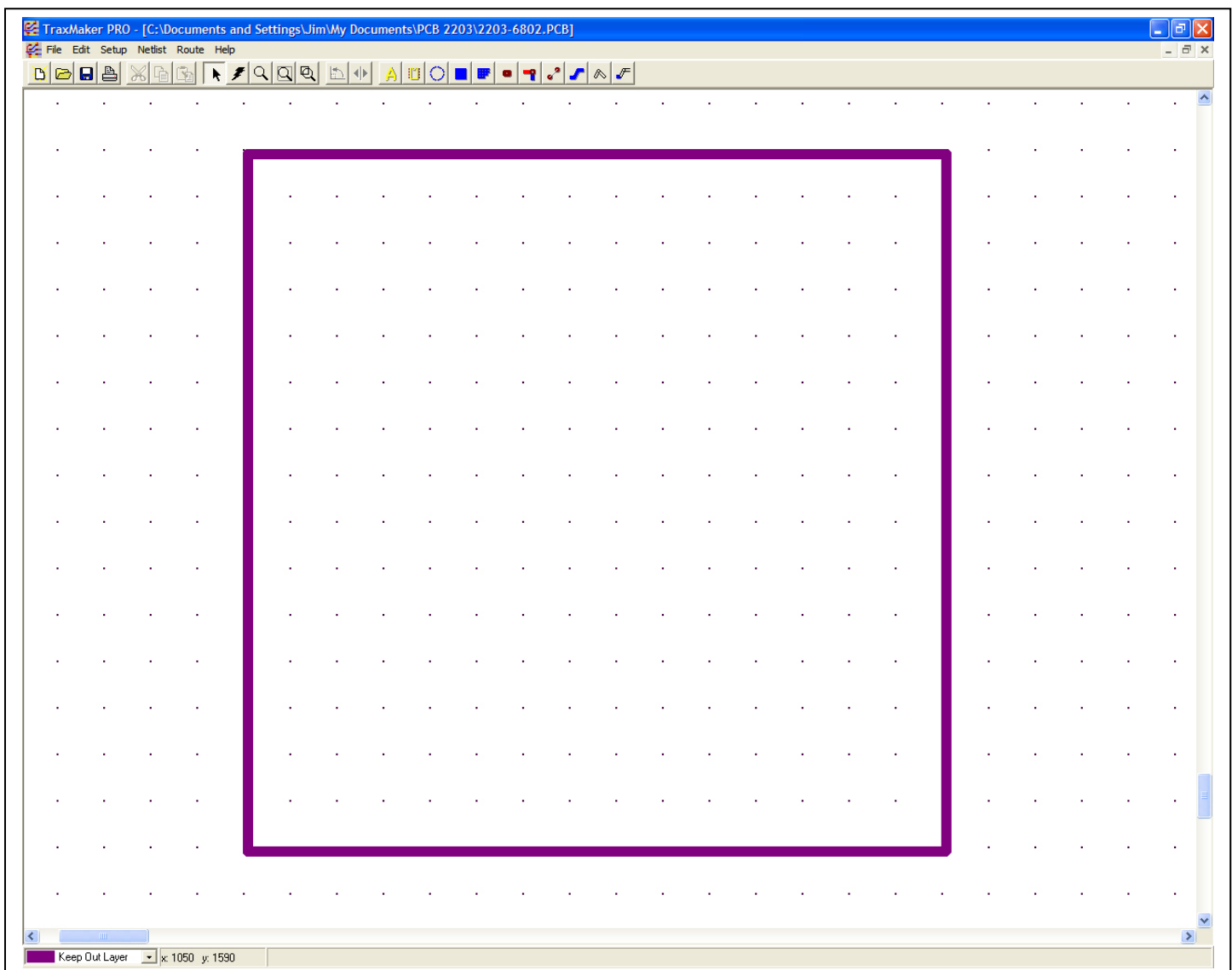
e) Traxmaker -- Making the PC board.

Traxmaker is going to allow you to take that schematic drawing you just created and turn it into a real live PC board.

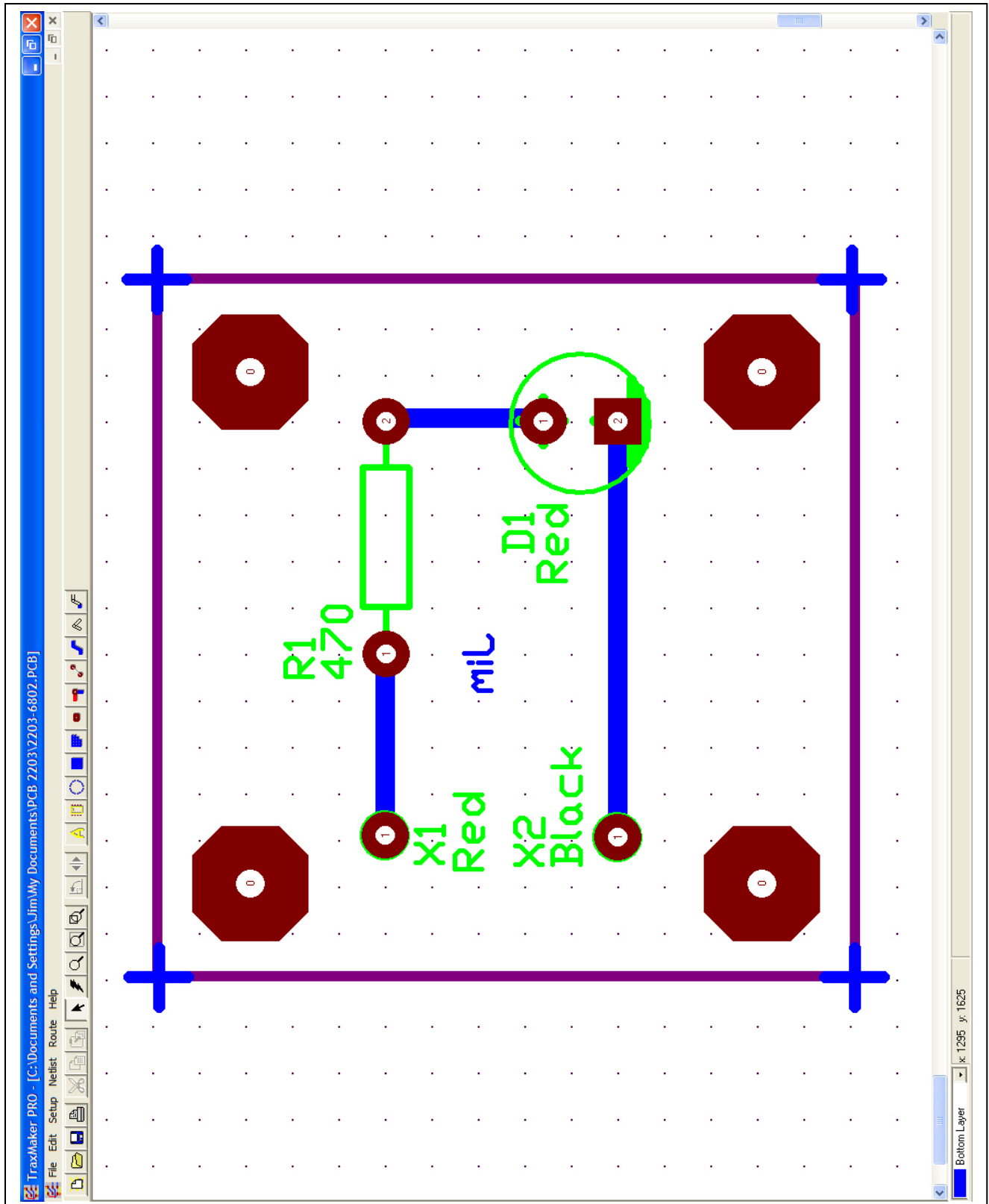
- (1) From the desktop, click on the Traxmaker icon and open Traxmaker.
- (2) In the lower left hand corner, there is a drop down box with a colored block in it. Open that drop down box and choose "Keep Out Layer" (purple color).



- (3) In the "Setup" routine at the top of the screen, choose "Setup Cursor", Relative, Set Floating Origin. Put the cursor about an inch away from the lower left hand corner of the screen and directly on one of the black grid dots. This is now your "zero-zero" corner from which we will measure everything
- (4) Note that in the little x-y box in the lower left hand corner, the numbers keep track of where your cursor is on the screen relative to that zero-zero corner you just set.
- (5) Now we are going to make the outline of the pc board that we are going to etch. Notice at the top of the screen that there are 26 small boxes to choose from. The third from the right is a blue "S" shape. Click on that box (this lets you draw a line, or a "trace" as we call it).
- (6) From this zero-zero corner run a trace 1500 mils to the right, 1500 mils up, 1500 mils left and 1500 mils down. This makes a square box 1.5 x 1.5 inches. Now go to Edit, Zoom, Fit Board To Window and the purple box should fill the screen.



- (7) Go to Netlist, Load Netlist, Traxmaker, Netlist and Components. Navigate to My Documents, then your personal PCB file, and the netlist inside of it. Click anywhere inside of the box outline and the parts will all be loaded onto the board.
- (8) Save this board file in the same PCB file. This file should be 1111-6803.pcb (note that a 5803 schematic builds a 6803 pc board). Do NOT give the board the extension .pcb -- the program does that automatically for you.



- (9) Go to the PADS button (brown oval, 6th from the right). Select an OCT250 pad and place one pad 200 mils vertically and horizontally from each corner (four total pads).
- (10) Arrange the components in roughly the same shape that they were in on your schematic -- input pins on the left, horizontal resistor along the top and ... oops. The program put the LED in horizontally too. Not a problem. Left click on D1 and then use the ROTATE tool (button with a curved arrow in the button bar at the top of the screen) and then drag D1 with a left button drag until the SQUARE hole is at the bottom of the screen and lined up with X2.
- (11) Oops. The program rotated the value and the reference designator too. Not a problem. Simply change (lower left corner of the screen) from the Keep Out Layer to make the Top Overlay (green) the active layer. Now select the value and the designator (Shift-click) and rotate them individually. Left drag them into a convenient position on the board.
- (12) Now go to Netlist, Show Nets, Show All Nets. There should be very narrow lines connecting X1 with R1, R1 with D1, and D1 with X2. These are where the copper TRACES are going to go.
- (13) You might want to take some time right now and move the parts around until those narrow lines are as straight as they can be.
- (14) Now (lower left hand corner) make the BOTTOM LAYER (blue) the active layer. In SETUP, TRACKS, make the default ORTHOGONAL with a width of 40 mils.
- (15) Using the TRACK button (blue, third from the right) draw blue (copper) tracks along all the net lines from X1 to R1 to D1 back to X2.
- (16) Use the TEXT button (yellow A) to type in your first name on the bottom layer and MIRRORED.
- (17) Use the text button to place a + sign (size 200 mils , line 25 mils), one at each corner. You can place the first one and then use the Windows copy function to place the rest of them.
- (18) Let's step back a pace and see what we have done. The components, which are going to be placed on the TOP (component) side of the board are in green. The traces, which are going to be placed on the BOTTOM (trace) side of the board are in blue. The pads (which are going to get holes for the parts to go through) are also going to be on the bottom layer. Your name is mirrored because you are looking at the bottom layer THROUGH the top layer and if you could do this in real life, you would see the text as mirrored (this prevents us from etching the board backwards). The plus signs on the corners are where we are going to shear the final board.
- (19) Let's print out our little masterpiece. File, Print, and a whole bunch of options. For this first go around, let's make the scale 2.000 x and 2.000 y. In Print Options, the only checks on the left column should be Check Print, Bottom Layer, and Top Overlay. In the right column, Pads, Vias, Text Strings, Single Layer Pad Holes, Pad Guide Size 30 mils, Color Mode B&W. OKPrint, OK.

- (20) Now, the same thing, only 1.000x and 1.000y. This will give you a 1:1 (exact size) to let you make the board exactly the size you asked for.
- (21) Since we are not going to PRINT resist onto this board, but draw it on by hand, let's print one more sheet, only this time in Print Options check the Mirrored option and uncheck the Top Overlay box. Now you have an exact roadmap of what you will need to draw on your PCB material.

f) Fabricating the PC Board

- (1) Cut a piece of pcb material an inch bigger than the final pcb. That is, for a pcb 1.5 x 1.5", cut the board material 2.5 x 2.5". Lay the MIRRORED copy from step (22) above, onto the pcb material in roughly the center of the material, face up.
- (2) Using an automatic center punch, punch the center of each of the 10 holes in the pads. Punch the center of the four (+) plus shear marks also.
- (3) Drill the 4 octagonal hole centers with a 0.120 bit (#4 clear). Clean out any burrs.
- (4) Drill the 6 component holes with a 0.040 (#60) bit.
- (5) Clean both sides of the board using a kitchen potscrubber, water, and a scouring or copper cleaning cleanser. From this point on, do not touch the board with your fingers. Dry it with a clean cloth and inspect it to be sure that it is bright and smooth. If there are any discolored or dull areas, clean it again.
- (6) Use a permanent marker ("Sharpie®") to draw all the traces on the copper surface of the PC board EXCEPT the four large octagonal traces in the corners. Make sure that the "pads" completely encircle the holes that you drilled.
- (7) Etch the board using standard ferric chloride or copper chloride techniques (see Section VI if you want to look ahead and see how etching is done).
- (8) Remove the ink resist with light sandpaper and tin the copper traces in the tin plating bath.
- (9) Install the battery connector wires to X1 (red) and X2 (black). Solder the wires and clip any excess lead length.
- (10) Install R1 (R1 does not have a "polarity" or direction in which it has to be installed.) Solder and clip.
- (11) Install D1. D1 does have a polarity. D1 should have a "flat" or flat spot on the bottom of the case of the diode. The "flat" (cathode lead) is noted by the black line on the top screen drawing and by the square pad on the trace drawing. If your diode does not have a flat, see the instructor to locate the cathode side of the diode.
- (12) Plug in the battery and light the LED up.

VI. Metal Fabrication

A. A Short Discussion of Various Metals

1. Steel is not used for a large amount of fabrication work in electronics. Basically, it is too heavy for many of the products that we fabricate. It also rusts, but rust can be prevented by a number of different techniques -- painting, plating, and hot dip galvanizing are three of the methods used.
2. Copper and brass are used where it is necessary to solder electronic components to the chassis for shielding or other electrical purposes. It is rare to find copper or brass used for any other purpose.
3. Other more exotic metals find limited use in electronic fabrication.

B. Aluminum

1. By far and away, aluminum is the metal of choice for electronic fabrication. It is lightweight, easily machined, relatively unaffected by corrosion, and inexpensive. Aluminum comes in hundreds of versions and strengths. Here are a few comments on the various alloys of aluminum:

Basic Aluminum Alloys

1xxx series---Aluminum of 99 percent or higher purity has many applications, especially in electrical and chemical fields. These compositions are characterized by excellent corrosion resistance, high thermal and electrical conductivity, low mechanical properties and excellent workability. Moderate increases in strength may be obtained by strain-hardening. Iron and silicon are the major impurities.

2xxx series---Copper is the principal alloying element in this group. These alloys require solution heat-treatment to obtain optimum properties; in the heat-treated condition mechanical properties are similar to, and sometimes exceed, those of mild steel. In some instances, artificial aging is employed to further increase the mechanical properties. This treatment materially increases yield strength, with attendant loss in elongation; its effect on tensile (ultimate) strength is not so great. The alloys in the 2xxx series do not have as good corrosion resistance as most other aluminum alloys, and under certain conditions they may be subject to intergranular corrosion. Therefore, these alloys in the form of sheet are usually clad with a high-purity alloy or a magnesium-silicon alloy of the 6xxx series, which provides galvanic protection to the core material and thus greatly increases resistance to corrosion. Alloy 2024 is perhaps the best known and most widely used aircraft alloy.

3xxx series---Manganese is the major alloying element of alloys in this group, which are generally non-heat-treatable. Because only a limited percentage of manganese, up to about 1.5 percent, can be effectively added to aluminum, it is used as a major element in only a few instances. One of these, however, is the popular 3003, which

is widely used as a general purpose alloy for moderate-strength applications requiring good workability.

4xxx series---The major alloying element of this group is silicon, which can be added in sufficient quantities to cause substantial lowering of the melting point without producing brittleness in the resulting alloys. For these reasons, aluminum-silicon alloys are used in welding wire and brazing alloys where a lower melting point than that of the parent metal is required. Most alloys in this series are non-heat-treatable, but when used in welding heat-treatable alloys, they will pick up some of the alloying constituents of the latter and so respond to heat treatment to a limited extent. The alloys containing appreciable amounts of silicon become dark gray when anodic oxide finishes are applied and are in great demand for architectural applications.

5xxx series---Magnesium is one of the most effective and widely used alloying elements for aluminum. When it is used as the major alloying element or with manganese, the result is a moderate to high strength non-heat-treatable alloy. Magnesium is considered more effective than manganese as a hardener--about 0.8 percent magnesium being equal to 1.25 percent manganese--and it can be added in considerably higher quantities. Alloys in this series possess good welding characteristics and good resistance to corrosion in marine atmosphere. However, certain limitations should be placed on the amount of cold work and on the safe operating temperatures permissible for the higher magnesium content alloys (over about 3.5 percent for operating temperatures above about 150 degrees F) to avoid susceptibility to stress corrosion.

6xxx series---Alloys in this group contain silicone and magnesium in approximate proportions to form magnesium silicide, thus making them heat-treatable. The major alloy in this series is 6061, one of the most versatile of the heat-treatable alloys. Though not as strong as most of the 2xxx or 7xxx alloys, the magnesium-silicon (or magnesium-silicide) alloys possess good formability and corrosion resistance with medium strength. Alloys in this heat-treatable group may be formed in the T4 temper (solution heat-treated but not artificially aged) and then reach full T6 properties by artificial aging.

7xxx series---Zinc is the major alloying element in this group, and when coupled with a smaller percentage of magnesium results in heat-treatable alloys of very high strength. Usually other elements such as copper and chromium are also added in small quantities. The outstanding member of this group is 7075, which is among the highest strength alloys available and is used in airframe structures and for highly stressed parts.

(Data courtesy of Lusk Metals. www.luskmetals.com/reference.html)

2. Here are the chemical makeups of several popular aluminum alloys. Pay particular attention to 5052 alloy.

Alloy	Si	Fe	Cu	Mn	Mg	Cr	Zi	Ti	Aluminum
1100	.095 Si + Fe		0.05-0.20	0.05	0.10	..	99.00
2011	0.40	0.7	5.0-6.0	0.30	..	Remainder
2024	0.50	0.50	3.8-4.9	0.30-0.9	1.2-1.8	0.10	0.25	0.15	Remainder
3003	0.6	0.7	0.05-0.20	1.0-1.5	0.10	..	Remainder
5005	0.30	0.7	0.20	0.20	0.50-1.1	0.10	0.25	..	Remainder
5052	0.25	0.40	0.10	0.10	2.2-2.8	0.15-0.35	0.10	..	Remainder
5083	0.40	0.40	0.10	0.40-1.0	4.0-4.9	0.05-0.25	0.25	0.15	Remainder
5086	0.40	0.50	0.10	0.20-0.7	3.5-4.5	0.05-0.25	0.25	0.15	Remainder
6061	0.40-0.8	0.7	0.15-0.40	0.15	0.8-1.2	0.04-0.35	0.25	0.15	Remainder
6063	0.20-0.6	0.35	0.10	0.10	0.45-0.9	0.10	0.10	0.10	Remainder
7075	0.40	0.50	1.2-2.0	0.30	2.1-2.9	0.18-0.28	5.1-6.1	0.20	Remainder

- The aluminum alloy of choice for this class is 5052-H32 which is 0.050" (50 mils) thick. Here are some comments on this alloy:

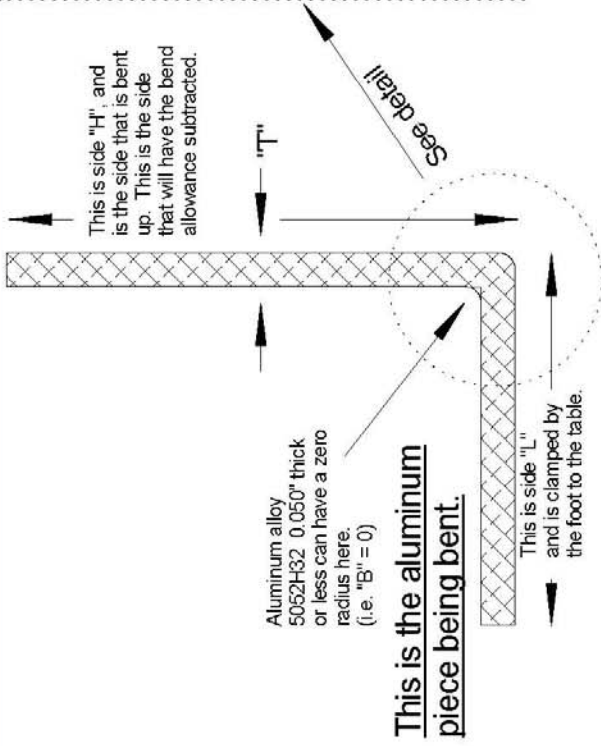
5052-H32 (QQ-A-250/8d)

Main alloy is magnesium. Forms well with reasonable inside bend radii. Corrosion resistance and weldability is very good. Better salt water corrosion resistance than 1100. Used for electronic chassis, tanks, pressure vessels and any number of parts requiring considerable strength and formability at reasonable cost. Anodizing may be slightly yellowish.

- Note well that the comment is that this alloy "forms well with reasonable inside bend radii". This is the engineerspeak for "it makes nice tight bends". As a matter of fact, the aircraft bend specification for 5052-H32 is that it will take a zero radius (i.e. no curvature in the bend) for metal up to 0.2" (200 mils) thick. Our process in making aluminum chassis out of 50 mil aluminum will take advantage of this specification for zero-radius bends.

C. Bend Allowance

- When any metal, including aluminum, is bent, there is a squeezing and stretching phenomenon going on that will "grow" the aluminum longer than the original flat piece that was cut to be bent. We will have to take this growth into account when we make the bends for the power supply chassis. As a matter of fact, as the next two pages show, 50 mil aluminum will grow 0.037" (37 mils) past what you would expect without the growth. Also do not forget that you gain an additional amount of growth because of the thickness of the metal. On a right angle bend, therefore, we will have to allow (50+37) for a growth of 87 mils in our chassis layout.

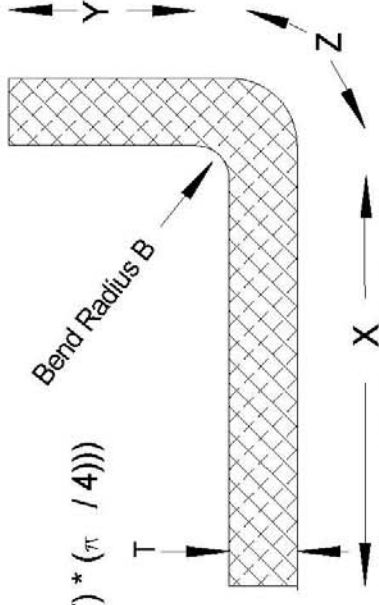


To lay out a flat piece to be bent, it should be $X + Y - Z$ long, where

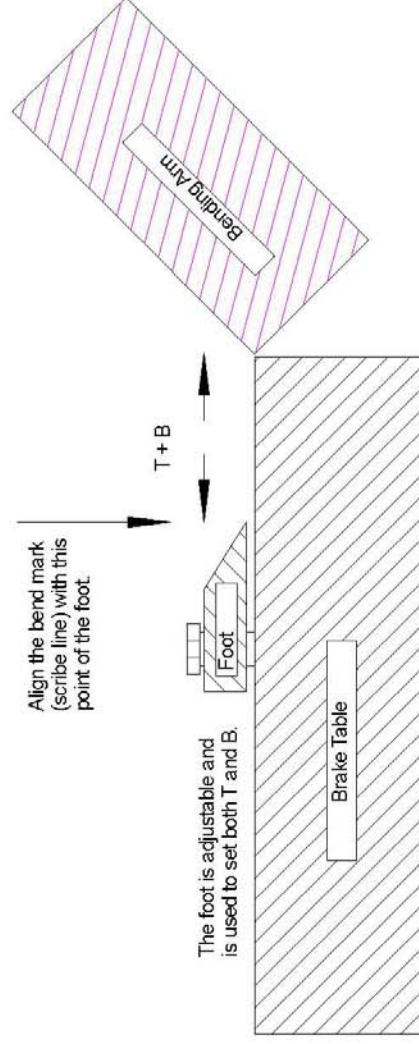
$$X = L - T - B$$

$$Y = H - B$$

$$Z = 2 * (B + ((0.90T) * (\pi / 4)))$$



- NOTES:
1. All dims in inches.
 2. The equation for Z contains a $\pi / 4$ term. This is for a right angle bend. The general form of the term is $\pi / (360 / \theta)$ where θ is the angle of the bend in degrees from horizontal flat plate.



Bending Brake, Bending Radius, and Bend Allowance

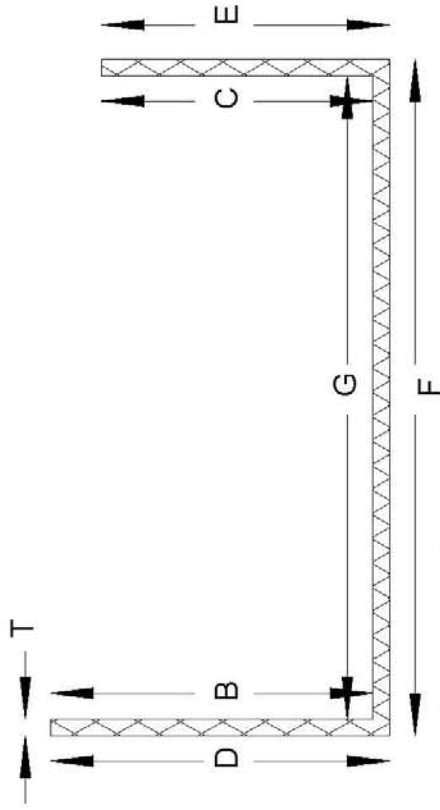
Scale: NTS
Date: 28 Aug 01

Drawn by: OMU
Rev: C

Approved by: *Jim Weir*

RST Engineering
13993 Downwind Court
530.272.2203
tech@rst-engr.com

Drawing #
6622-0620
Sheet 1 of 1



$$D=B+T$$

$$B=D-T$$

$$E=C+T$$

$$C=E-T$$

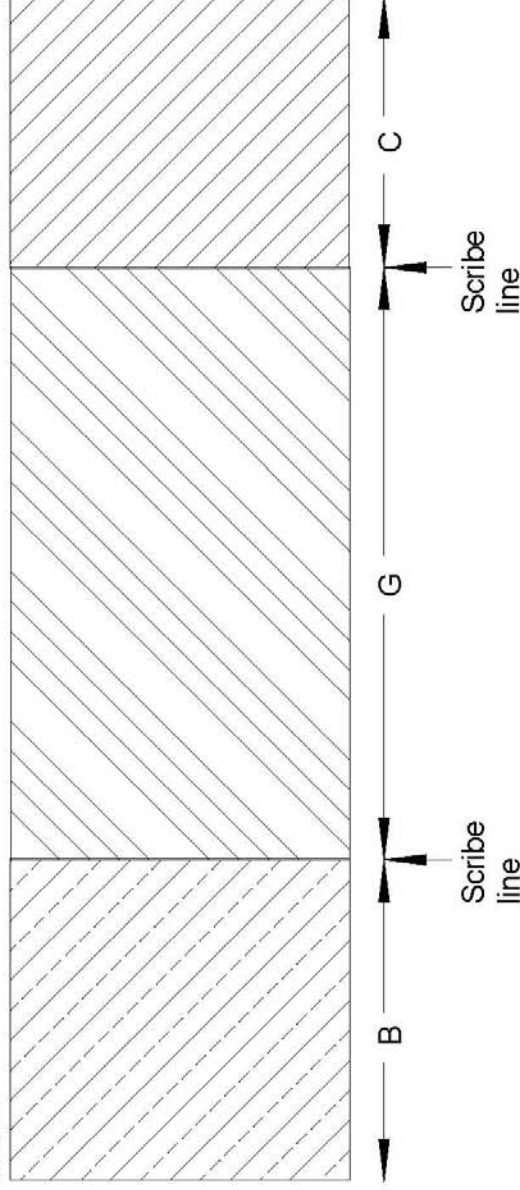
$$F= G+ (2*(T*1.7))$$

$$G= F- (2*(T*1.7))$$

Note – the "2" in these equations is because there are 2 bends. If the second bend does not exist, reduce this "2" allowance to 1 allowance.

This section gets held under the foot of the brake.

This section gets held under the foot of the brake.



Metal Bending Allowances

Scale: NTS
Date: 28 Aug 01

Drawn by: OMJ
Rev: C

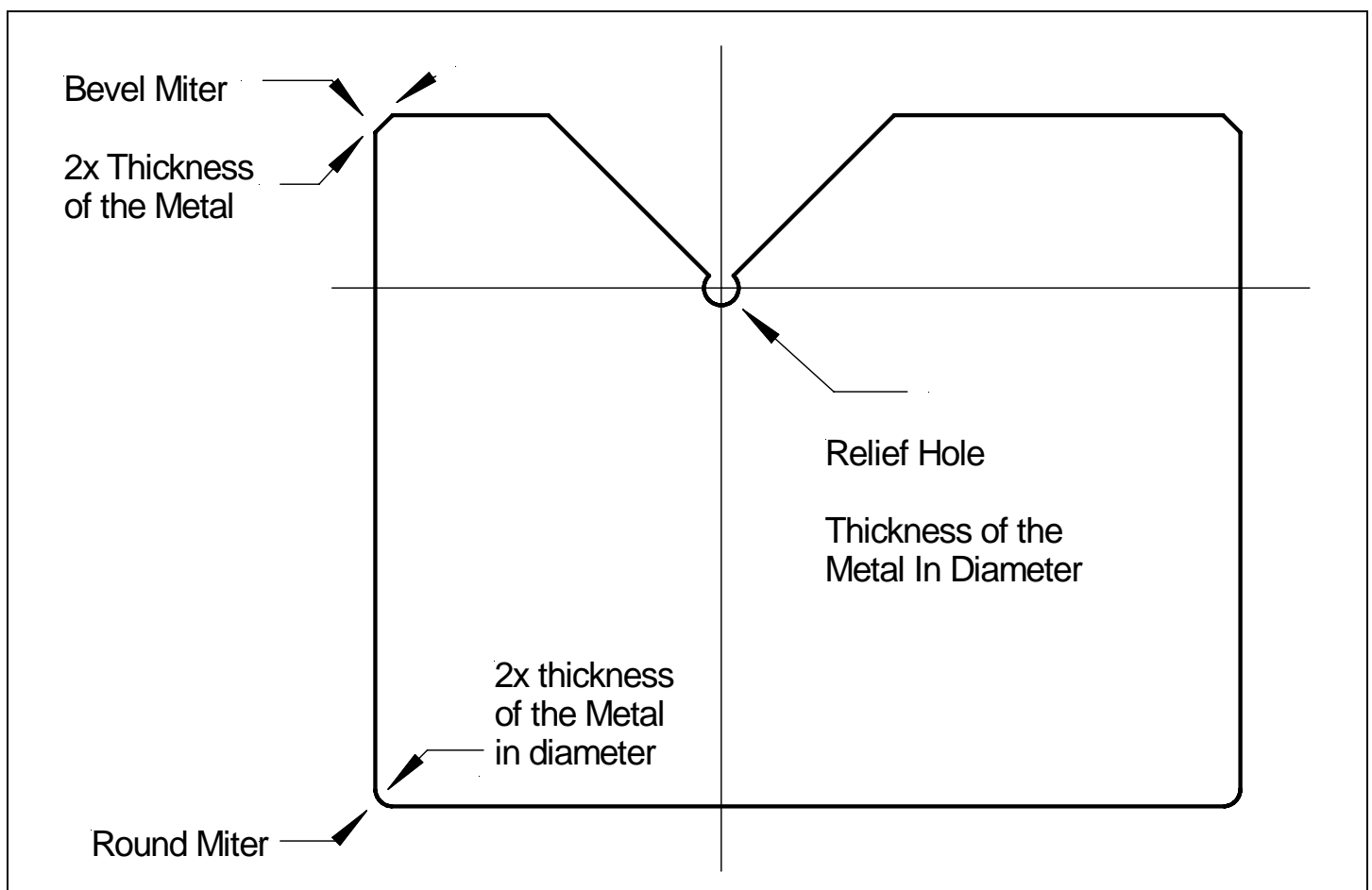
Approved by:
Jim Weir

RST Engineering
13993 Downwind Court Grass Valley CA 95945
530.272.2203
tech@rst-engr.com

Drawing #
6622-0621
sheet 1 of 1

C. *Mitering and Relief Holes*

1. In general, sharp corners and tight box bends are things that we will need to address from a safety and stress relief point of view.
2. Mitering, or the "rounding off" of corners may be done in two general ways. One way is to take a little tiny bit (roughly twice the thickness of the metal wide) 45° cut across the corner ("bevel miter") and the other way is to round the corner with a file or rotary tool. Corners mitered in this manner have much less of a chance of causing damage to skin or finished surfaces than a plain sharp corner.
3. When a piece of metal comes to a sharp inside corner (like a V-notch for a box corner) it sets up stress lines into the metal from that corner. The sharper the corner, the more stress lines are set up. If, however, we have a small hole at that corner, the stress lines are greatly reduced. In general, a hole about the thickness of the metal will reduce the stress lines due to the corner to nearly zero .



D. Finishing

The finishing procedure for aluminum is a two-step process. First, we need to coat the bare aluminum with some sort of protective layer (paint or anodize) and we also need to put some sort of legend or lettering on the panel.

1. Priming and Painting

- a. Aluminum does not readily take paint because of the growth of an impervious oxide layer on the surface of the aluminum. In general, it is necessary to PRIME the aluminum surface before proceeding to paint..
- b. The very BEST primer for aluminum is zinc chromate. Unfortunately, zinc chromate is somewhat toxic to humans and animals and is generally relegated to commercial uses. It is widely used in the aviation industry and can be purchased without restriction from any aviation parts supply store. There are several dozen aviation mail order stores in the country that will ship small quantities of zinc chromate spray cans. (Note -- ground shipping is no problem; air shipping incurs a significant hazmat fee .
- c. Whichever version of primer you use, be sure to do a warm water and soap rinse of the chassis and rough the surface up just a bit using a green "tuffly" type nylon pot scrubber. Do ***NOT*** use steel wool on the aluminum surface; you will be years picking the little tiny steel fibers out of the aluminum. Dry the aluminum thoroughly (a toaster oven works well) before painting.
- d. Primer should go on thin enough so that you can just barely see the surface of the aluminum below the primer. Do NOT put it on thick.
- e. A second, and nearly as good a primer is the "rattle can" automotive primer found in any good home improvement store. For light duty such as that found by a lab bench power supply, spray cans of primer will do quite nicely.

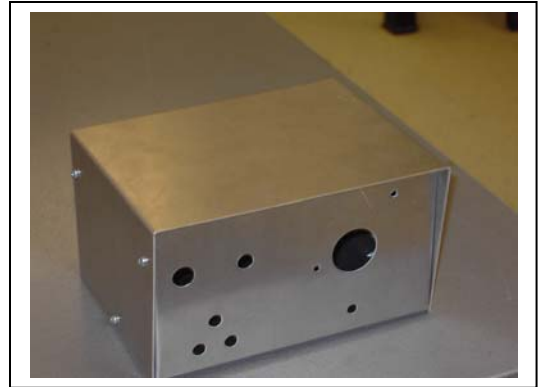


- f. When it comes to the final finish, a good spray can of professional quality enamel is hard to beat. Keep the following in mind:
1. The piece to be painted should be at a warm room temperature ... not freezing or baking.
 2. There should be no hand contact with the aluminum after priming. Use vinyl gloves or use a wire pickup loop through a hole in the chassis for moving the chassis.
 3. A few thin coats are better than one thick coat. Let the coats dry thoroughly between application (the legend on the can should give an approximate time between coats).
 4. Choosing a color is an art. I personally prefer a light beige for the chassis and either a dark brown or black for the cover. Use satin rather than gloss or flat -- satin hides a multitude of toolmark sins.



2. Anodizing

- a. It used to be that anodizing could only be done by the plating shop down in the industrial park. Then somebody posted that they tried doing it at home and it worked...sort of. Then a few more of us started experimenting and we've gotten quite good at it.
- b. **CAUTION. YOU ARE WORKING WITH STRONG CHEMICALS THAT CAN BURN SKIN, CAUSE BLINDNESS, AND IN GENERAL NOT BE PARTICULARLY GOOD FOR YOUR HEALTH.** On the other hand, millions of people work with drain cleaner, battery acid, and fabric dye every day and don't hurt themselves. Just take precautions.
- c. What we start out with is the raw aluminum chassis. Do not put ANY other metal into the anodizing baths. In particular, don't forget to take out the steel chassis screws before beginning the process.

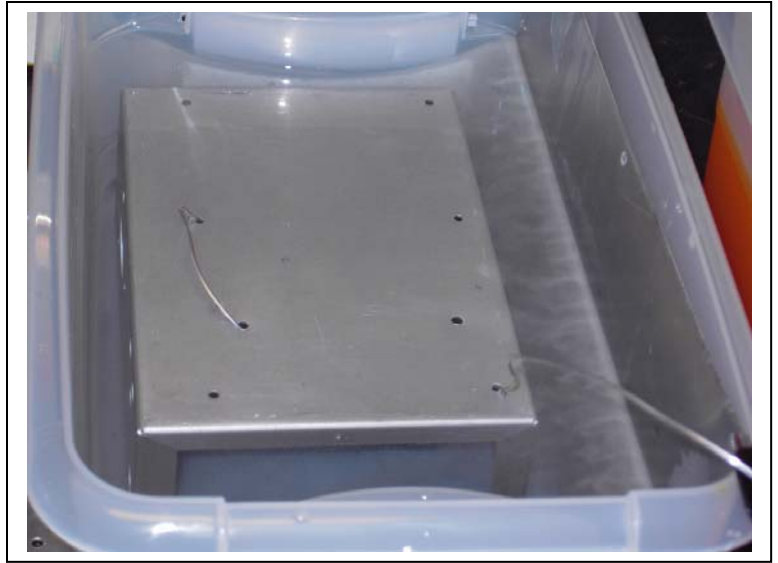


- d. Here is the process for anodizing and color coating:

1. The chassis is thoroughly washed with warm soapy water and a green "tuffy" pot scrubbing pad. Any tool marks, nicks, and the like should be polished out with 400 grit sandpaper before washing. Thoroughly rinse the chassis with clear water.
2. The etch bath is drain cleaner ("Drano") mixed somewhere between 4:1 and 8:1 with water (pour Drano into water, not the other way around). I prefer the milder solution but it takes a little longer. The chassis is completely immersed in the drain solution for half an hour to an hour. You may see some slight bubbling, but if the bubbling becomes heavy or apparent, you have too strong or too warm a solution. Rinse well after the etch bath with warm water.
3. The anodize bath is battery acid mixed in the same proportions as the etch bath (first water, then acid). A "cathode" (negative plate) is immersed in the solution along with the chassis to which has been attached an aluminum wire. (DO NOT USE STEEL OR COPPER WIRE or allow the cathode to touch the chassis.) A power supply is attached to the cathode (-) and chassis



(+). The current of the power supply is adjusted so that you have approximately one ampere of current for each square foot of surface area on BOTH sides of the



chassis. The cathode will become quite full and "boiling" with bubbles, but the chassis should have relatively few bubbles on the surface. If you are using thin aluminum wire, be prepared to change it several times during the anodizing process. The anodizing process itself should take between half an hour to an hour, depending on how thick you want the anodize layer. Note in this photo the aluminum contact wire serpentine through holes on the bottom and lots of bubbles coming from the cathode plate in the right side of the tray.

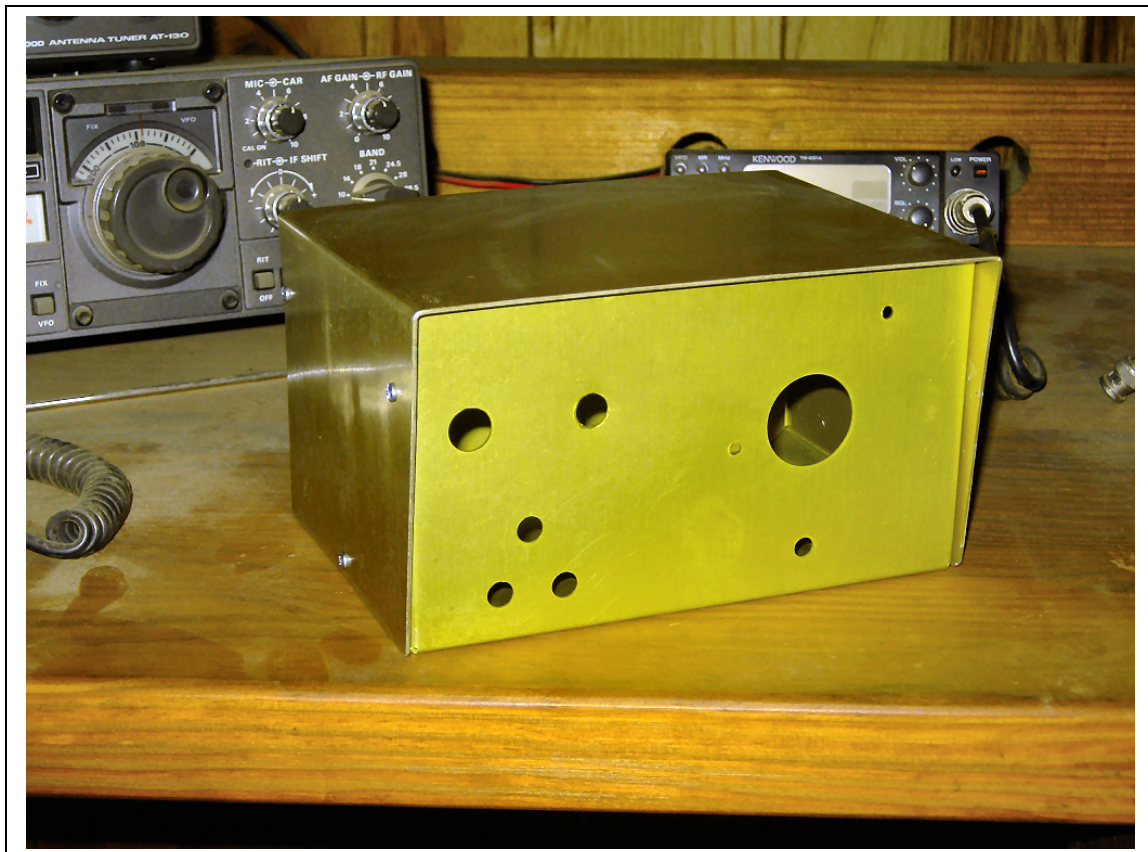


4. After the anodize bath, and rather quickly, rinse well with warm water and immerse the chassis in the fabric dye bath. For light colors, two packages of dye per gallon works well; it may take double or triple of the dye plus some heating of the solution for darker colors. Again, depending on the depth of the color, it will take between half an hour to an hour in the dye bath.

5. After the dye bath, to completely harden the surface (it actually turns the surface into aluminum hydrate), steam the chassis and other parts that you have anodized for an hour in a vegetable steamer.



6. Here is the finished product:



3. Lettering & Legend

1. (tbd)

E. Hardware & Fasteners

1. When joining two pieces of metal together, there are a wide range of fasteners used. Here are the most common in the electronics industry.
2. By far and away, the most common are machine screws. Machine screws come in sizes from tiny (holding a wristwatch together) to huge (holding a spacecraft together). Rarely in electronics do we go outside the range (right to left) of #4, #6, or #8.



Note that a machine screw has a constant and uniform thread from top to bottom.

3. To fasten that screw down we need a little more hardware. Here we see (left to right) a flatwasher, an internal tooth lockwasher, and a machine nut. The flatwasher is only necessary if you are absolutely sure that the metal surface that you are fastening down on is flat, otherwise the lockwasher is useless.



4. The screws can have several types of heads on them. Some of the most common (left to right) are the flathead, the pan or binder head, and the brazier head. Note that the pan and brazier only need a through-hole for mounting, but the flathead needs a countersink in the hole also so that it will lie flush with the top of the metal.



5. The screws can also have several styles of drive. On the left is the slot type of drive and on the right is the so-called crosspoint commonly called a phillips head.



6. Where components will be mounted at some distance above the chassis, we can use spacers to put some clearance between the component and the chassis. From left to right we have the relatively inexpensive rolled aluminum spacer, a nylon insulating spacer, and a solid aluminum spacer (least to most expensive).



7. Where we want to feed wires through a hole in the chassis we can use a rigid plastic or flexible vinyl (shown here) grommet. A grommet simply puts an insulating layer between the sharp edge of a chassis and the relatively soft insulation of a wire.



8. Now we come to what I consider one of the greatest inventions of all time. This little plug goes by several names. The formal name is "hole plug" but in the real world it is more likely to be called a "botch button" or a "butch plug". Its sole function in life is to conceal the fact that you drilled or punched a hole where it should not have gone and you don't want to have to remake the chassis. When asked what it is for, you simply smile and say that you were making provisions

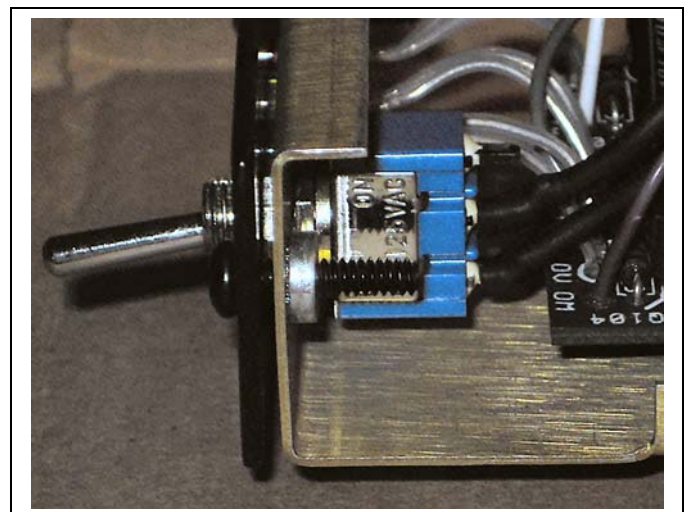
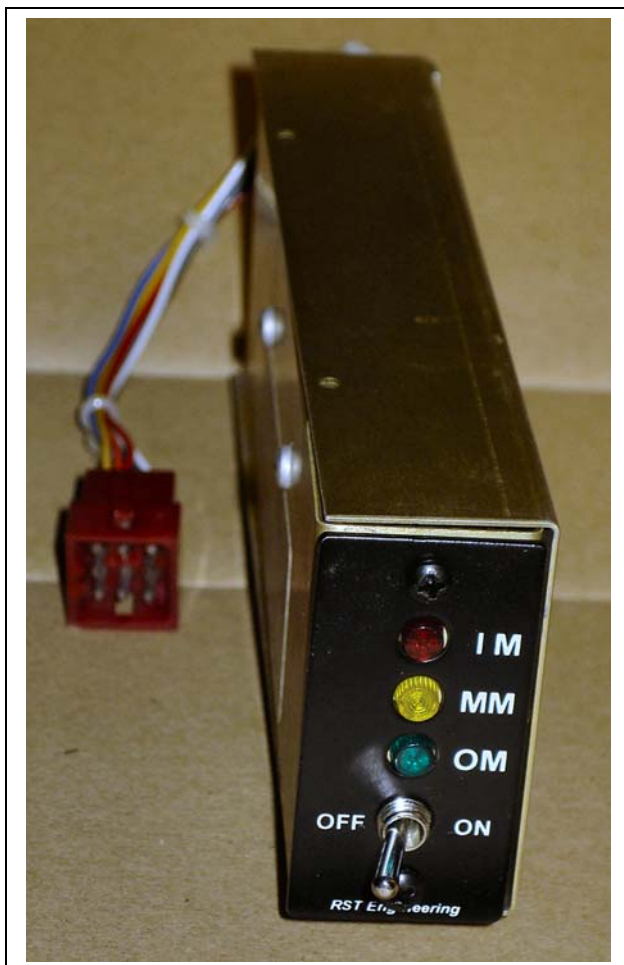


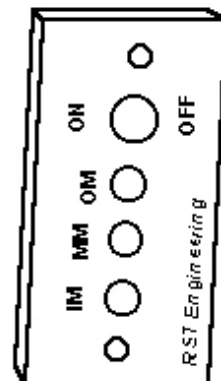
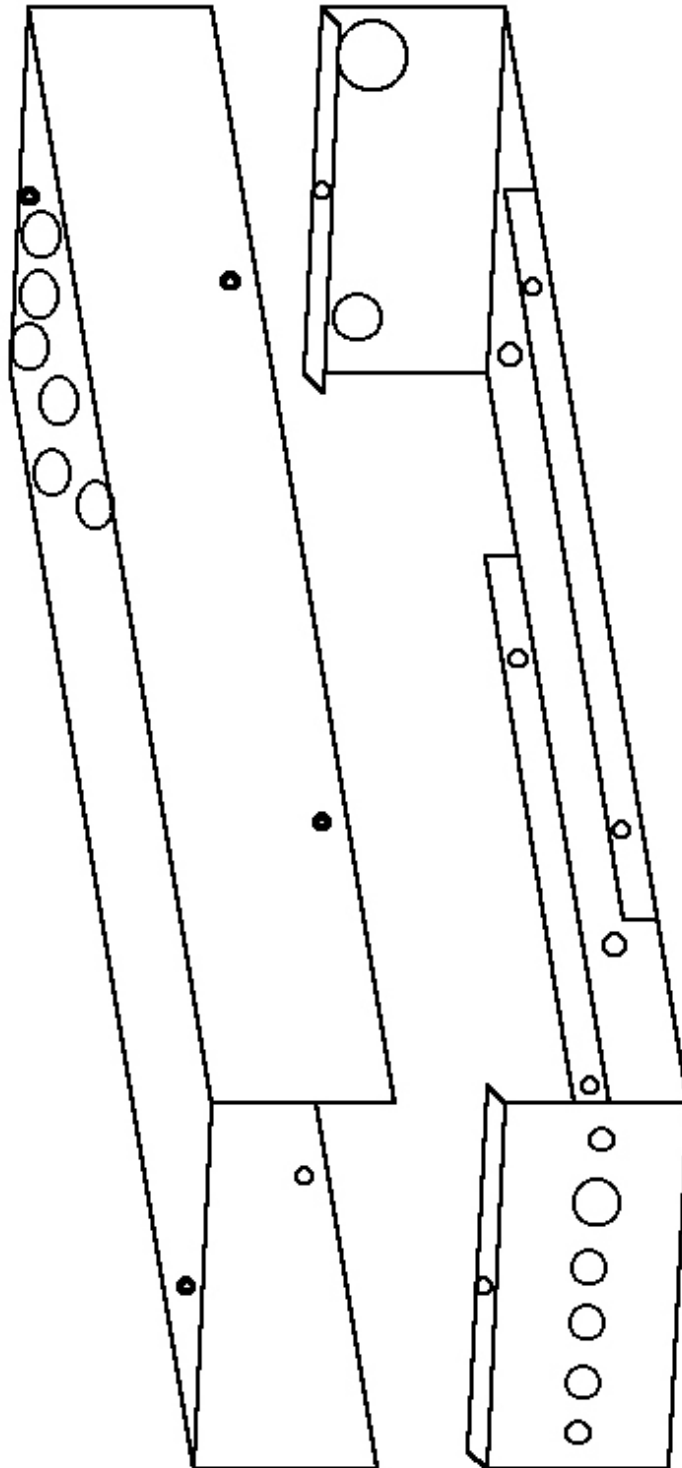
for additional peripherals to be attached to your product at some future point in time. No need to spill the beans that you botched up the drawing.

Speaking of botching up the drawing, I want you all to be aware that all of us, newcomers to old timers, all of us work to a single MIL-SPEC when making prototypes. That specification is MIL-TDD-41 (Make It Like The Drawing For Once).

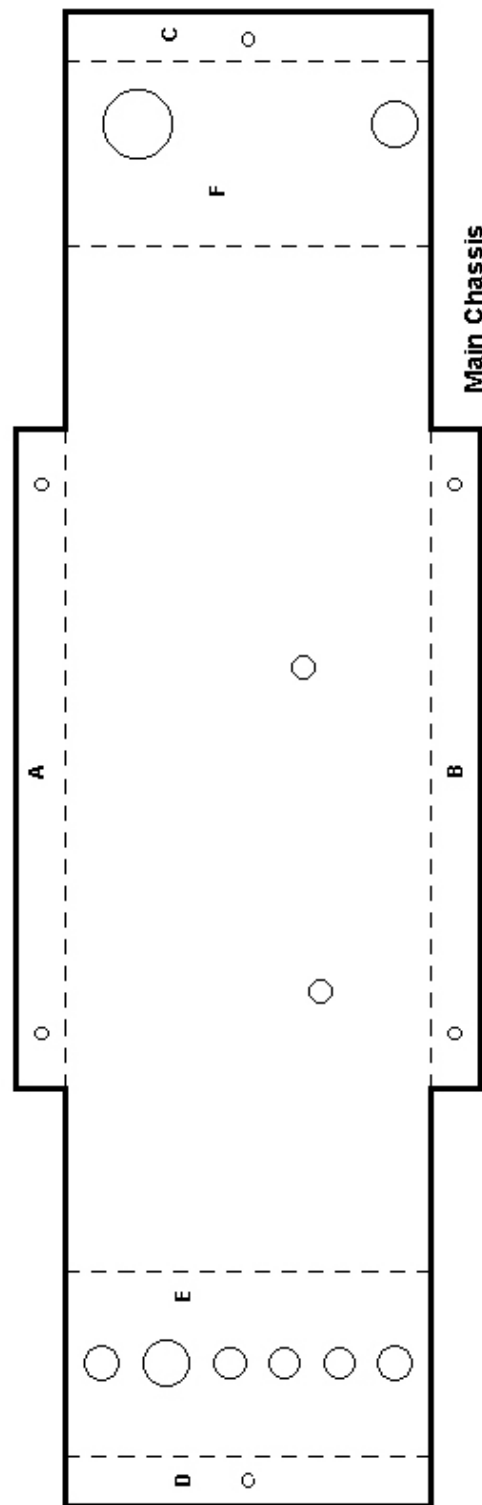
F. Paper Dolls

1. Before beginning the prototype construction of any project and committing the design to bent up metal, a cardboard cutout of the design, with all the parts simulated in cardboard, is a good way to "proof" the design.
2. "Cardboard" as used in this sense means any stiff paper. The most common "stiff paper" found around the average shop or office is a manila file folder. These file folders come in letter size for small projects and legal size for slightly larger projects, and they are cheap.
3. The next pages contain a 3-d drawing and a full size print of a paper doll for a small navigation receiver. This navigation receiver is made up of a chassis bottom, chassis cover, faceplate, pc board, and small mechanical components. It is **VITALLY IMPORTANT** to make paper dolls of all the parts so that (for example) the front panel switch doesn't hit the PC board when the unit is assembled.
4. It is certainly possible rather than making paper dolls for ALL the components to use the components themselves if the structure will support the components. For example, we made a paper doll for the PC board in this receiver, but we used an actual switch to see if there was clearance. (There was NOT enough clearance in the original design; we had to move the PC board back a couple of hundred mils to clear the switch terminals.)
5. Here are photographs of an assembled receiver. Notice how close the switch terminals (blue switch) are to the (black) PC board.

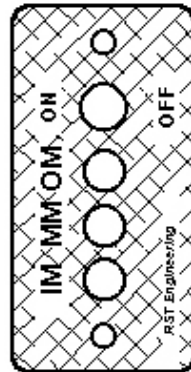
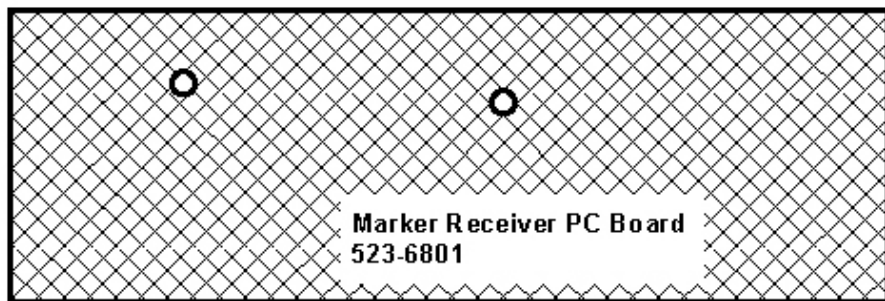




RST-523 3-d Conceptual Views			
Scale: NTS	Approved by:	Drawn by: OWJ	
Date: 15 Aug 96	<i>Jim Wein</i>	Rev: A	
RST Engineering		Drawing #	
13249 Grass Valley Ave Grass Valley CA 95945			
916/272-2203 (voice & fax) rst-engr@oro.net			
			523-0600

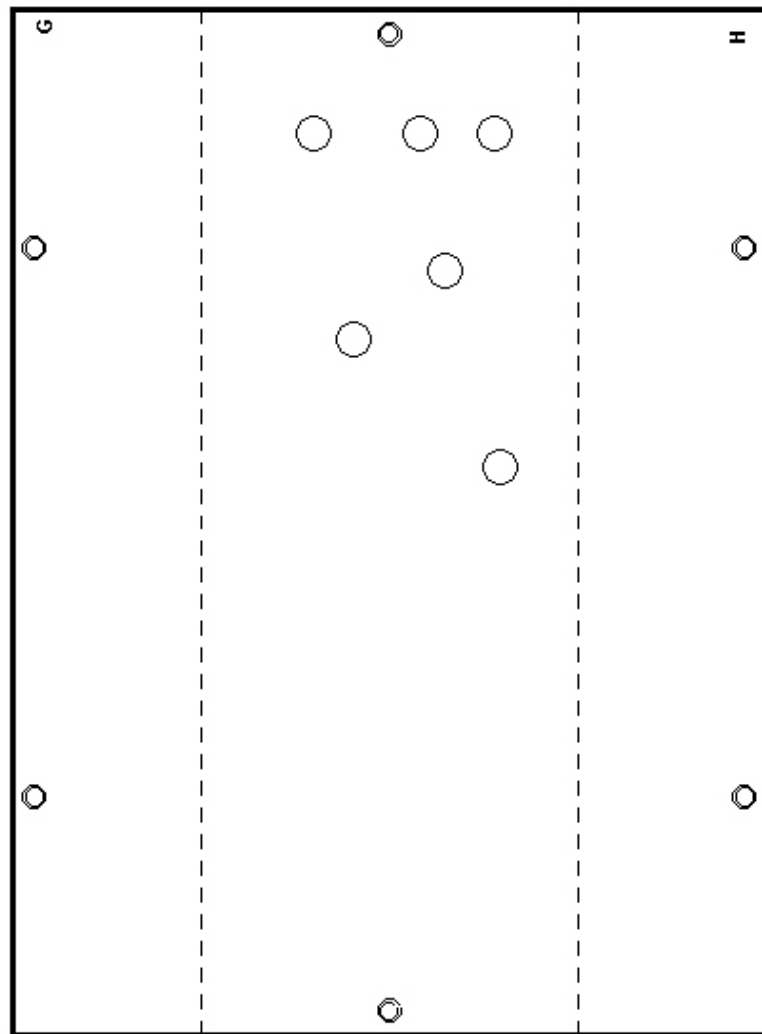
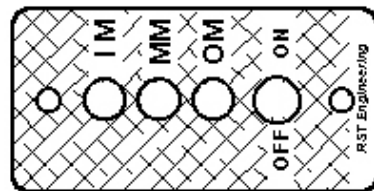


Main Chassis



Faceplate horizontal mount

Faceplate
Vertical
mount



Cover

RST-523 Marker Beacon Receiver Printer Prototype Cutout Drawing

Scale: 1:1

Approved by:

Gene Weir

Drawn by: OWJ

Date: 17 Aug 96

Rev: A

RST Engineering

13249 Grass Valley Ave Grass Valley CA 95945
916/272-2203 (voice & fax) rst-engr@oro.net

Drawing #

523-0610

6. Note on the drawing 523-0610 that there is a single dimension of 5.60" for the length of the top cover. This is given so that if you have been given a drawing that is not an original copy of the drawing that you can check to see if there has been some sort of printing distortion (bigger or smaller) and that if you use the drawing as-is that there will be size differences between the real part and your paper doll.

G. Power Supply Chassis Design & Fabrication

1. In the design of any product, one of the primary things to take into account is how the product is going to be used. In the particular instance of the power supply, the end product is intended to be used as a laboratory utility bench power supply, not intended for rough-and-tumble use outside, nor for high-precision work inside.
2. For lab instruments, we generally try and:
 - a. Put large heavy components on the bottom for mechanical stability
 - b. Put some sort of shield over expensive delicate components.
 - c. Since 90% of the world is right-handed, put the most used controls as far to the RIGHT as possible and indicators or lesser used controls on the LEFT side of the chassis front.
 - d. Almost-never-used controls and parts on the rear of the chassis.
3. Let's see if we can achieve these design goals with our power supply:
 - a. The transformer is the heaviest component in the power supply and the PC board with all its components is second heaviest. Mount them both on the chassis bottom.
 - b. Make a U-shaped chassis with lips for mounting screws and an "eyebrow" cover so that if the power supply gets tipped over, the eyebrow will keep the expensive delicate meter from being trashed.
 - c. Put the on-off switch to the far left, the voltage control near the middle and the (optional) voltage-current switch on the far right. Mount the meter on the far right in the "corner" of the eyebrow to achieve maximum protection.
 - d. Mount the power cord and fuse on the back of the chassis.
4. The drawings on the following pages are illustrations of how such a supply might be fabricated. Please feel free to choose another form factor (mechanical design) for your supply if you wish.



Lab Regulated Power Supply
0-15 volts DC @ 1.5 amperes
CIE-14 Summer '98
20 July 1998 jw

Fuse
Type 3AG
0.25 amp
Slo-Blo



110 VAC
60 Hz
15 w
1 ph



Power Supply Rear Panel Screen

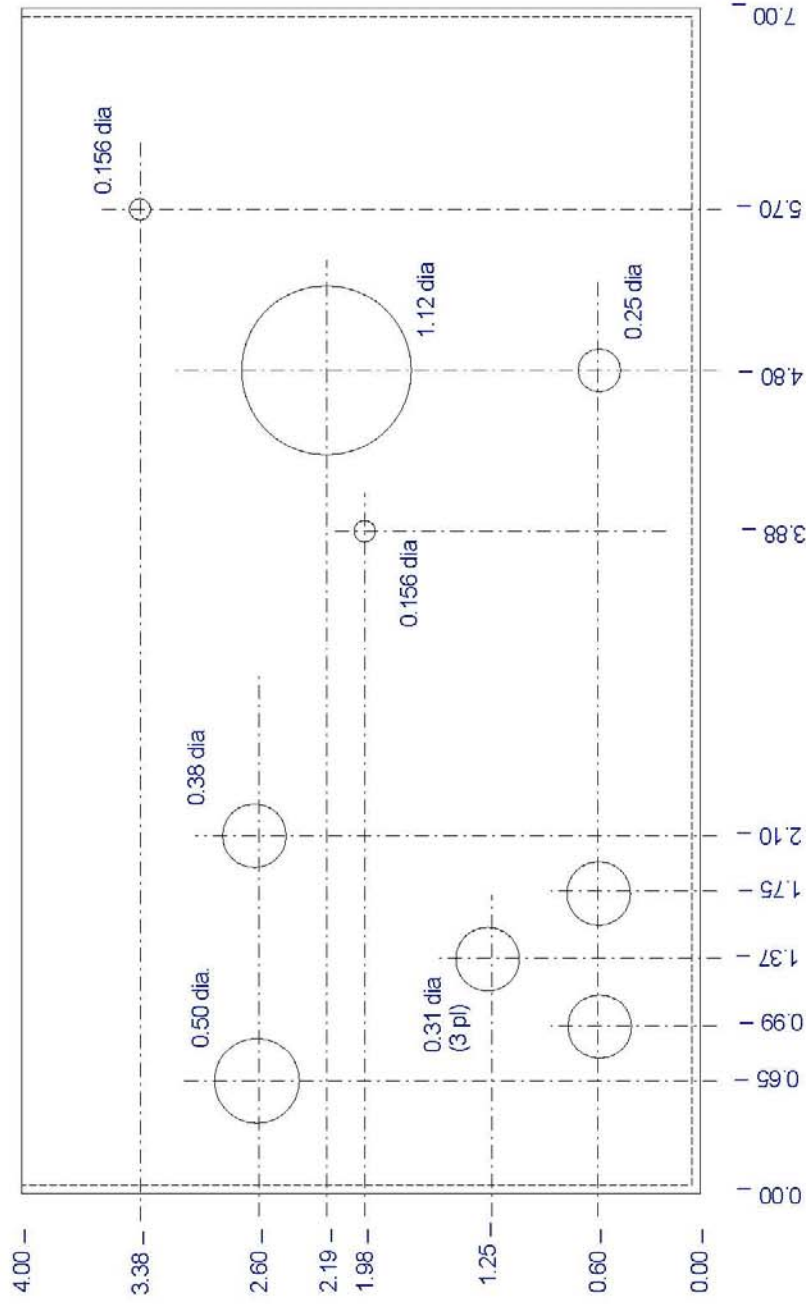
Scale: 1:1
Date: 30 Jun 98

Drawn by: OWJ
Rev: A

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley CA 95945
916.272.2203 tech@rst-engr.com

Drawing #
6622-1200
Sheet 1 of 1



Front View Basic Version

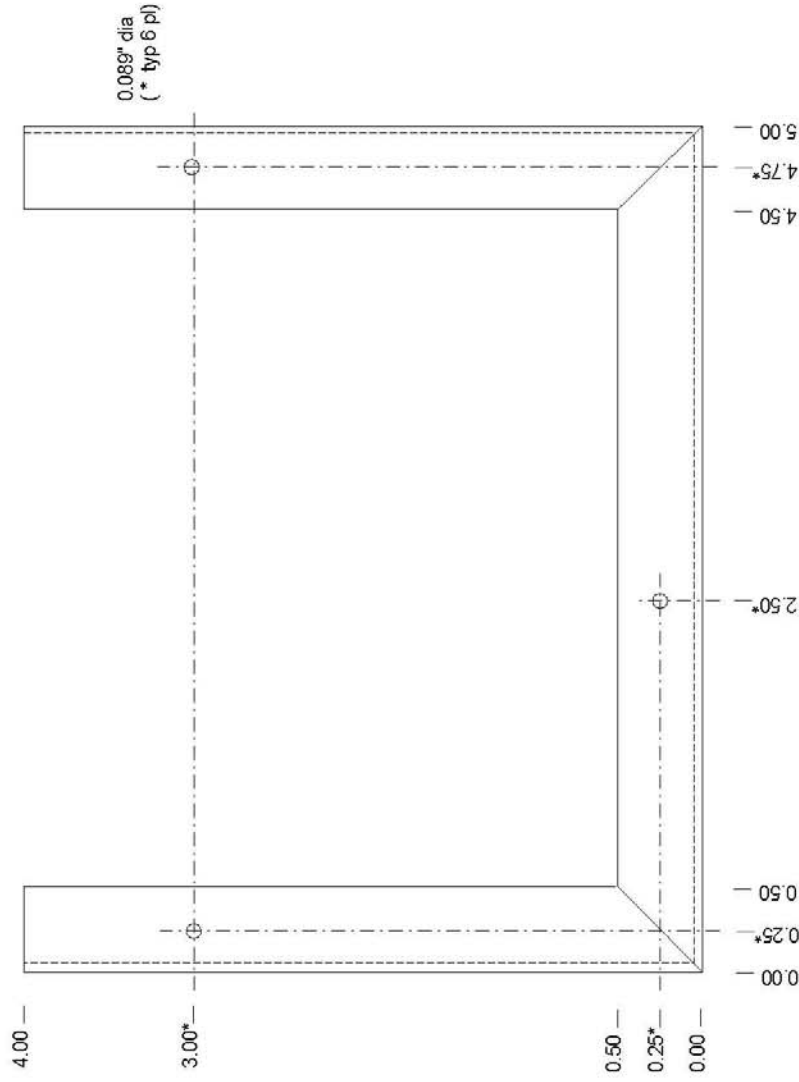
Power Supply Main Chassis

Scale: 1:1
Date: 31 Oct 06

Drawn by: OWJ
Rev: F

Approved by: *Jimm Mader*
RST Engineering
13249 Grass Valley Ave Grass Valley CA 95945
530.272.2203 tech@rst-engr.com

Drawing #
6622-2200
Sheet 1 of 5



NOTE: Hole dimensions marked with an asterisk (*) will be drilled using the cover as a template.

Right Side View

Power Supply Main Chassis

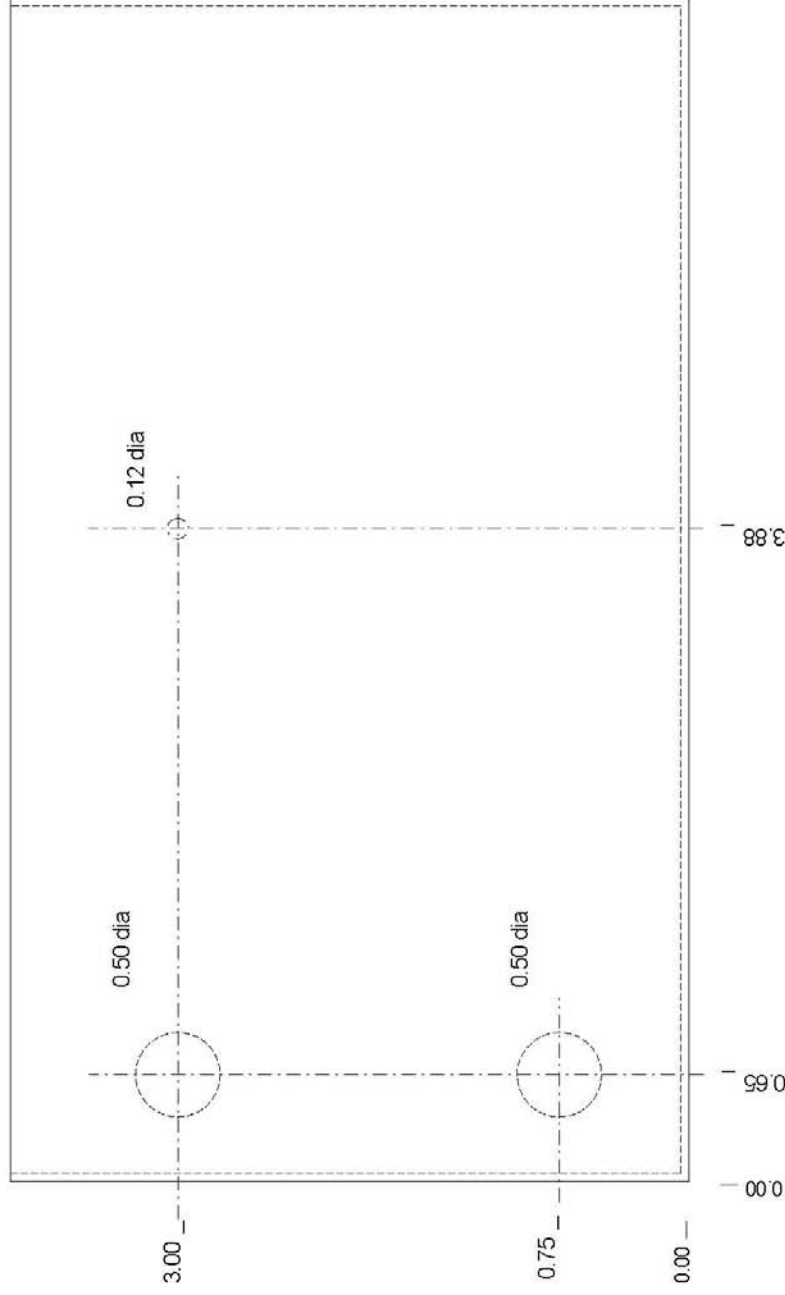
Scale: 1:1

Drawn by: OWJ

Approved by: Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley CA 95945
916.272.2203 tech@rst-engr.com

Drawing #
6622-2200
Sheet 2 of 5



Power Supply Main Chassis
Back Seen Through Front

Power Supply Main Chassis

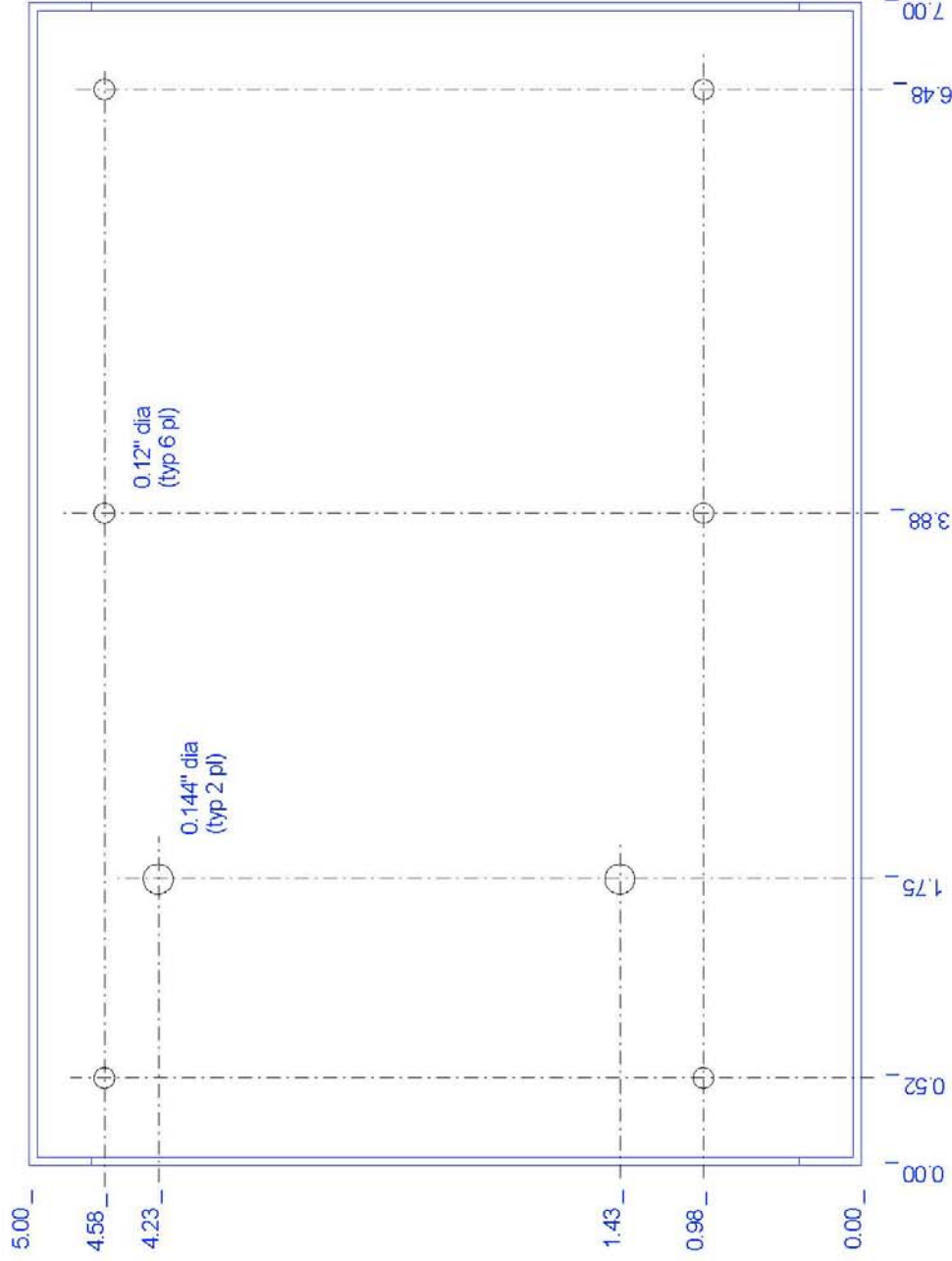
Scale: 1:1
Date: 02 Oct 04

Drawn by: OWJ
Rev: C

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley CA 95945
530.272.2203 tech@rst-engr.com

Drawing #
6622-2200
Sheet 3 of 5



Bottom Seen Through Top View

Power Supply Main Chassis

Scale: 1:1
Date: 22 Nov 01

Drawn by: OWJ
Rev: B

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley CA 95945
530.272.2203 tech@rst-engr.com

Drawing #
6622-2200
Sheet 4 of 5



Front View

Power Supply Cover

Scale: 1:1
Date: 28 Aug 05

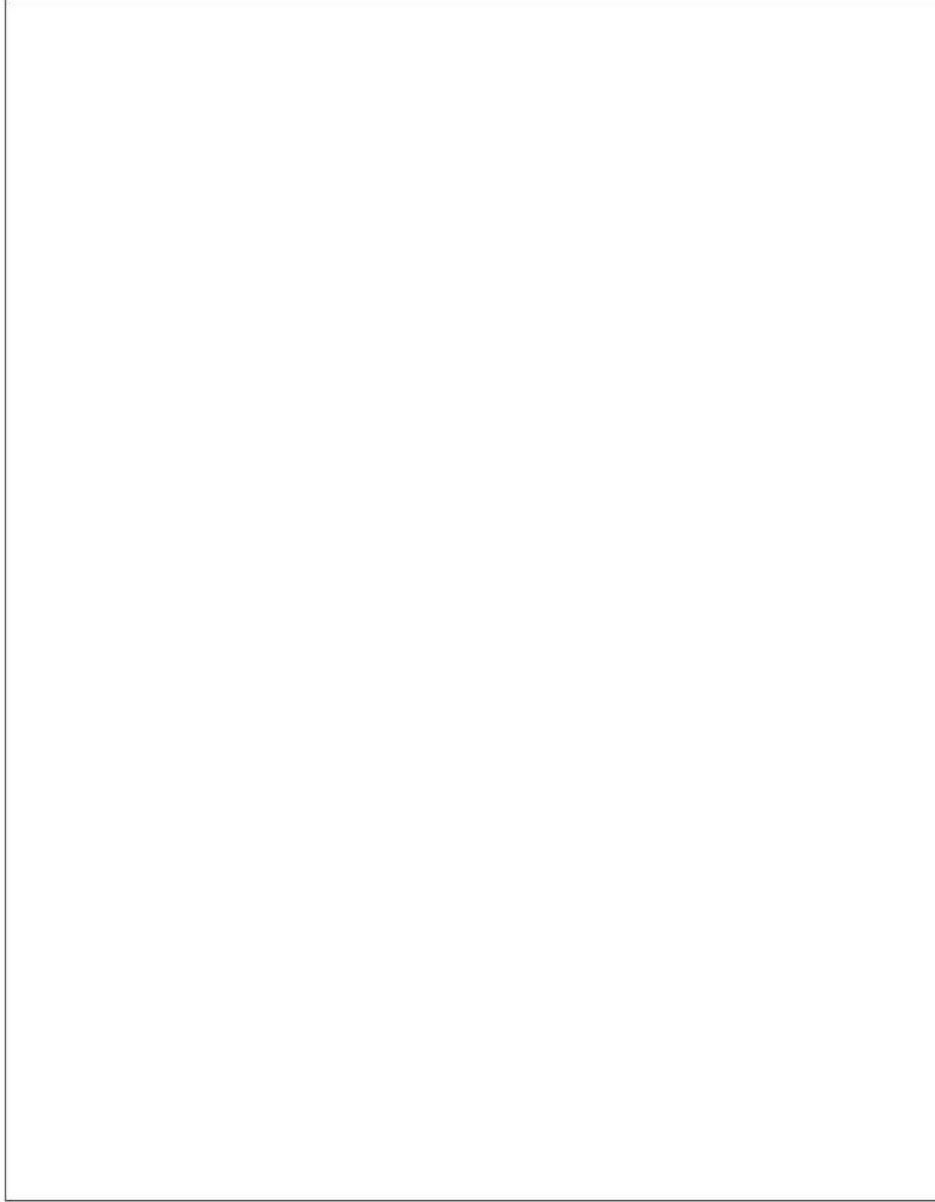
Drawn by: OWJ
Rev: C

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley CA 95945
530.272.2203 tech@rst-engr.com

Drawing #
6622-2201
Sheet 1 of 3

5.50 —



0.00 —

— 00'0

— 7.14

Top View

Power Supply Cover

Scale: 1:1

Drawn by: OWJ

Date: 28 Aug 05

Rev: C

Approved by:

Jim Weir

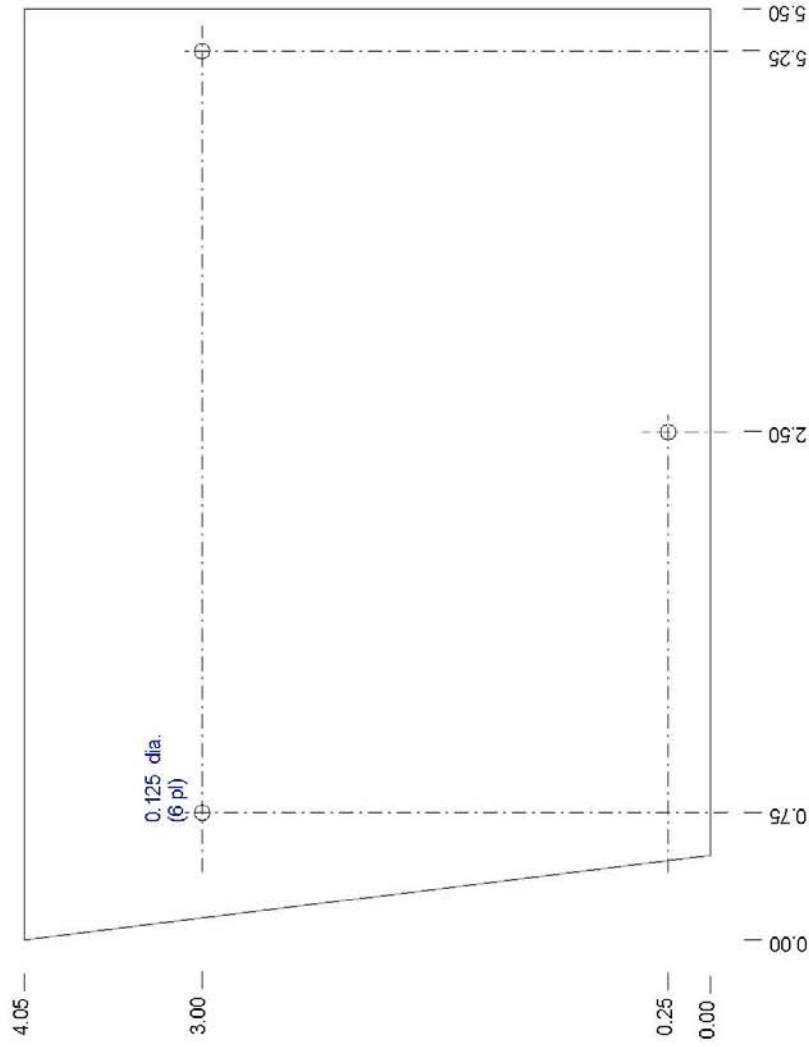
RST Engineering

13249 Grass Valley Ave Grass Valley CA 95945
530.272.2203 tech@rst-engr.com

Drawing #

6622-2201

Sheet 2 of 3



Power Supply Cover

Scale: 1:1

Drawn by: OWJ

Approved by:

Rev: C

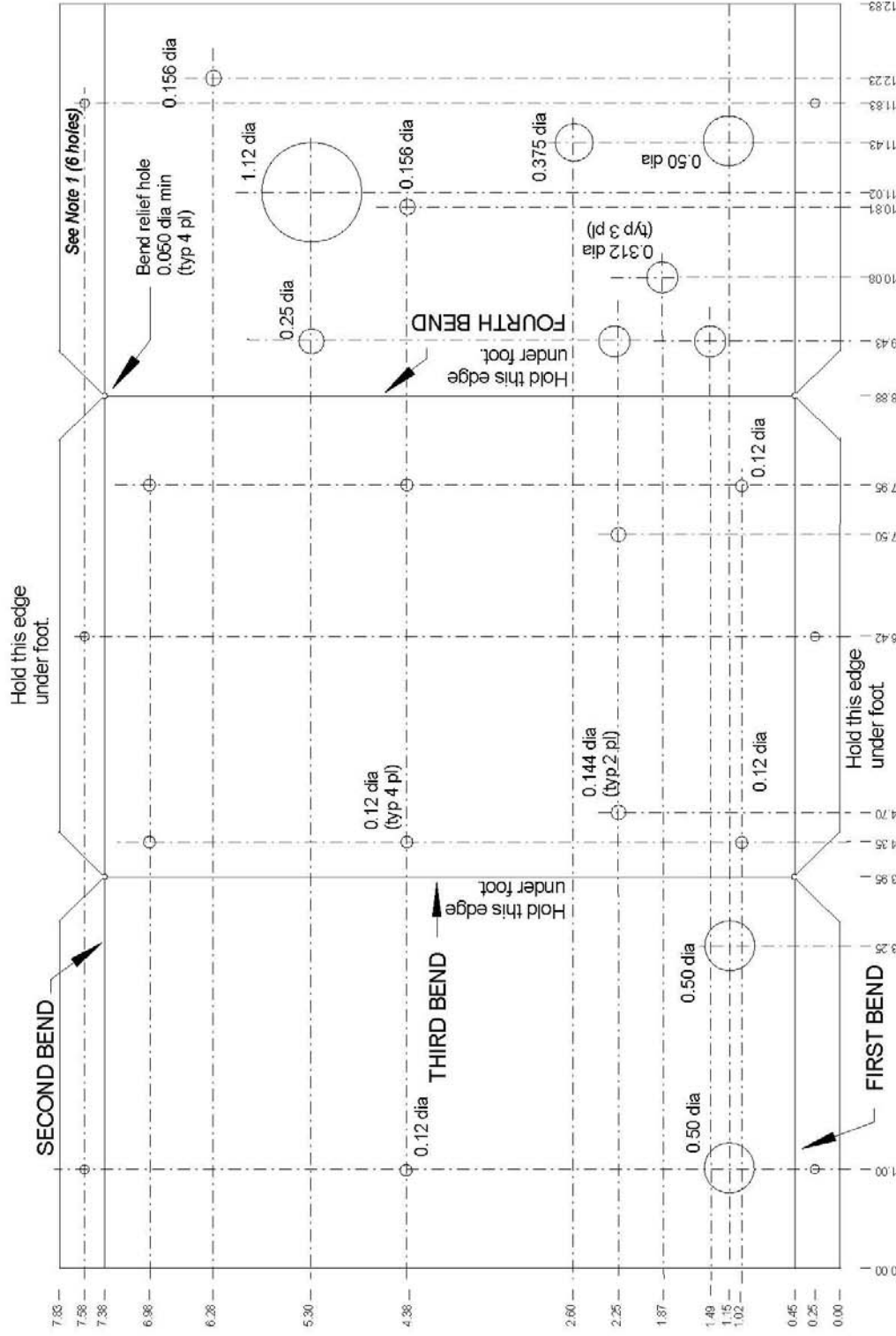
Date: 02 Oct 04

Jim Weir

RST Engineering

13249 Grass Valley Ave Grass Valley CA 95945
530.272.2203 tech@rst-engr.com

Drawing #
6622-2201
Sheet 3 of 3



NOTES:

1. Before drilling the 6 cover mounting holes 0.089 diameter, fabricate the cover and then mark these holes using the cover as the template.
2. Material 5052H32 0.050 thick.
3. Drill/punch all holes FIRST, then cut the notches SECOND, then bend THIRD. Note order of bends. All bends are UP (away from sheet).

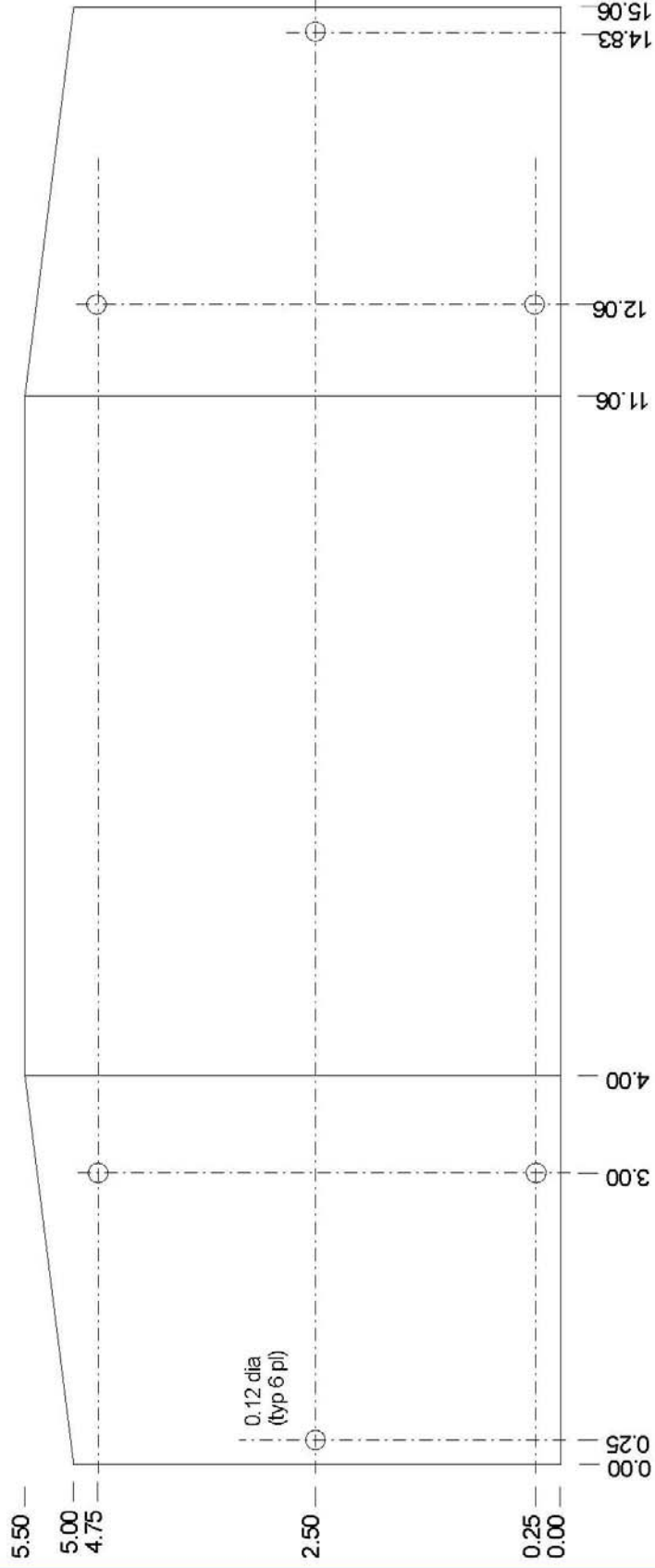
BEND DRAWING BASIC CHASSIS

Scale: 2.3 approx
Date: 30 Oct 06
Drawn by: OWJ
Rev: F

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley CA 95945
530.272.2203
tech@rst-engr.com

Drawing #
6022-2250
sheet 1 of 1



BEND DRAWING - COVER

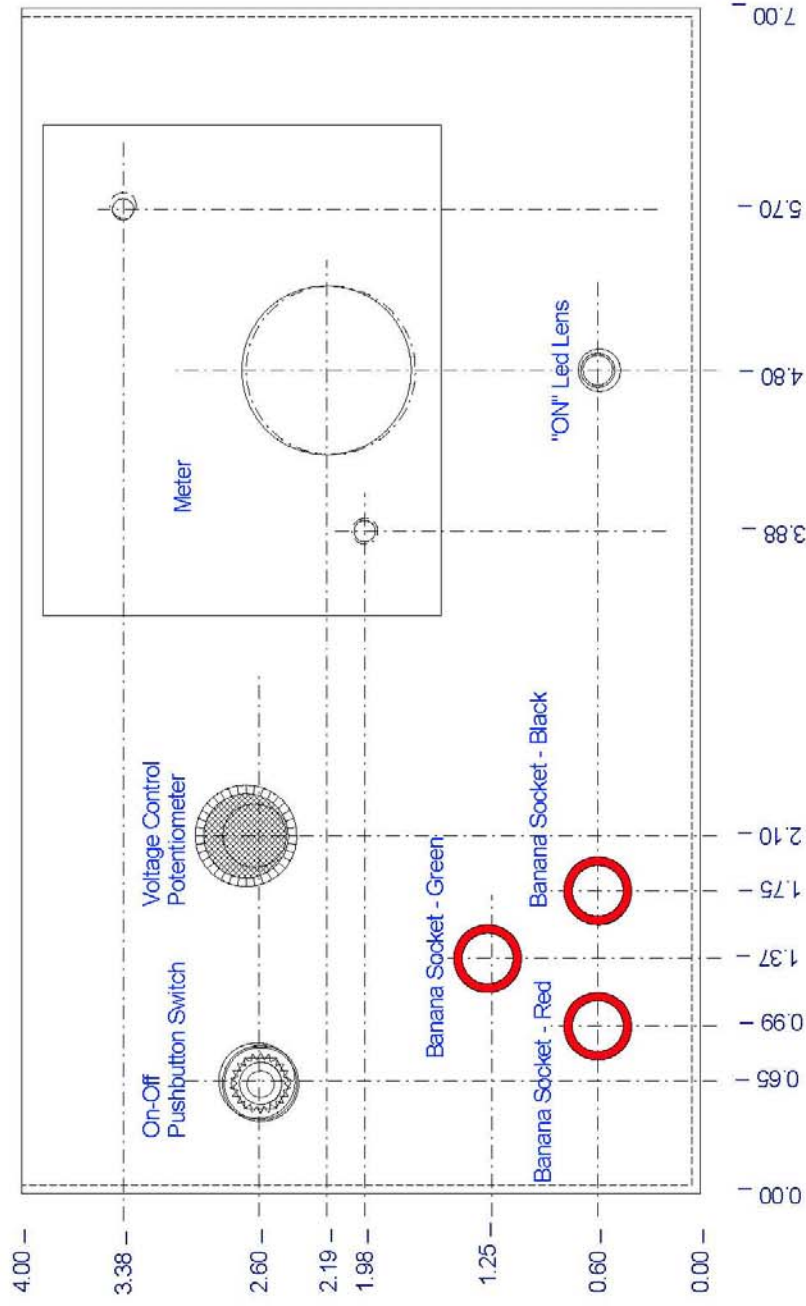
Scale: 2:3 approx
Date: 28 Aug 05

Drawn by: OMJ
Rev: B

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley CA 95945
530.272.2203 tech@rst-engr.com

Drawing #
6622-2251
sheet 1 of 1



Assembly Drawing Basic Version

Power Supply Main Chassis

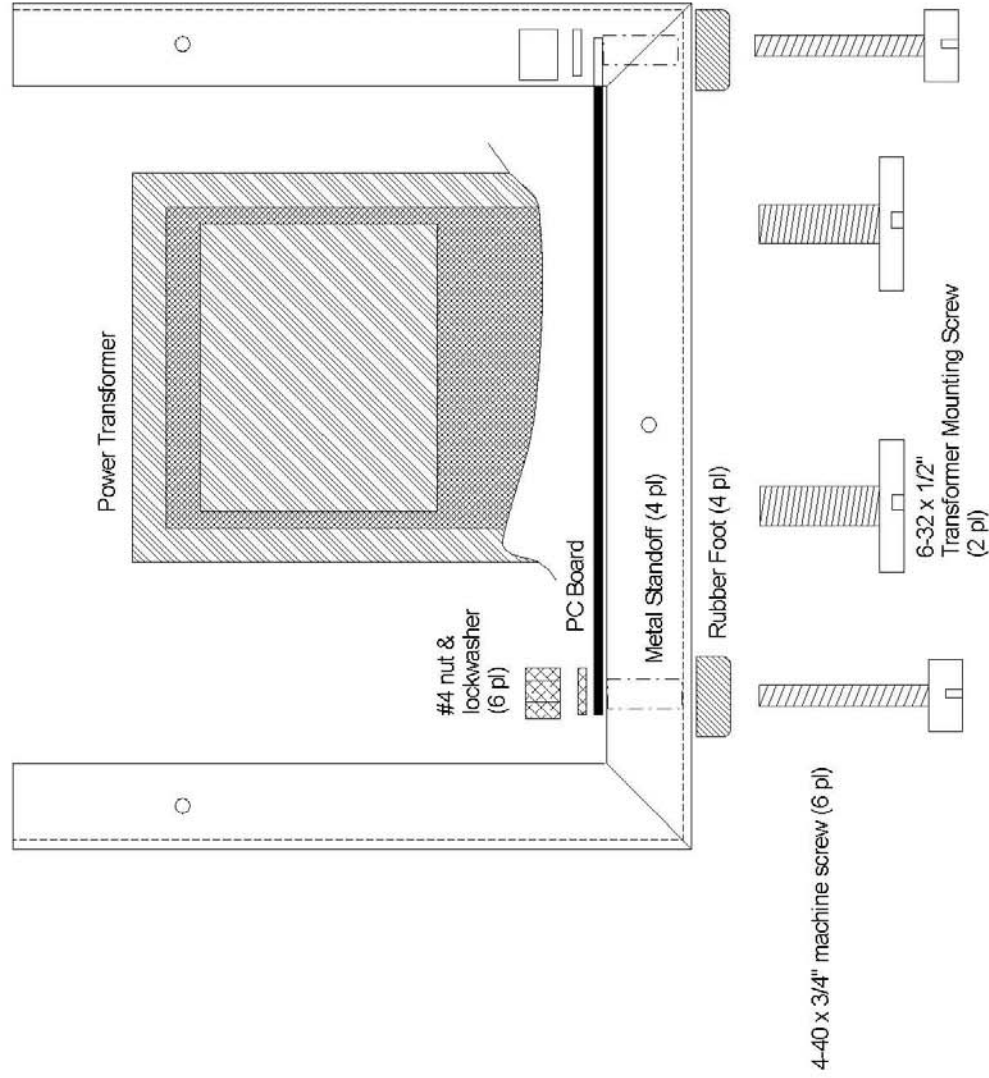
Scale: 1:1
Date: 02 Oct 04

Drawn by: OWJ
Rev: C

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley, CA 95945
530.272.2203
tech@rst-engr.com

Drawing #
6622-4200
Sheet 1 of 5



Assembly Drawing Right Side View

Power Supply Main Chassis

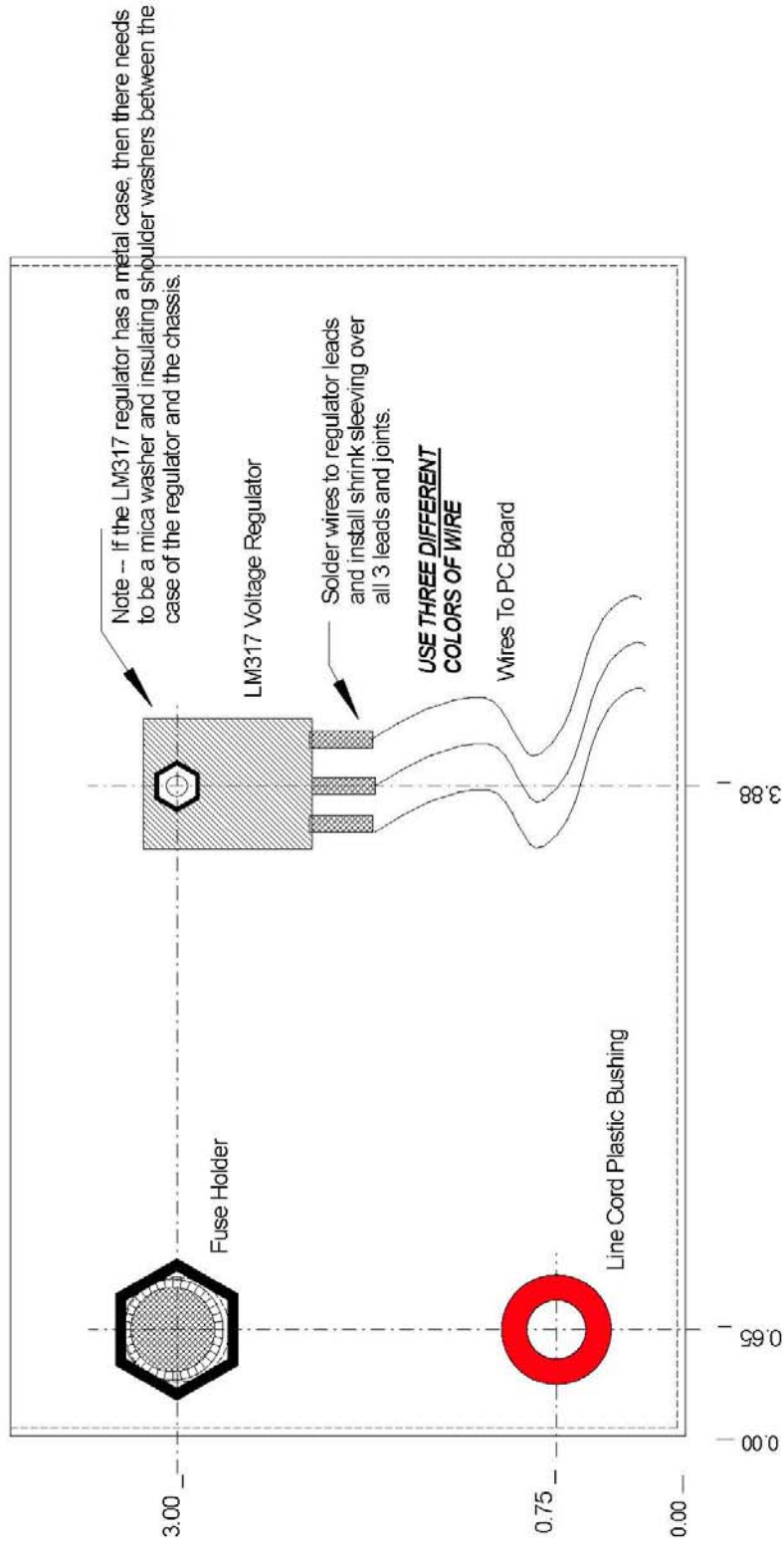
Scale: 1:1
Date: 20 Nov 01

Drawn by: O/WJ
Rev: B

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley, CA 95945
916.272.2203 tech@rst-engr.com

Drawing #
6622-4200
Sheet 2 of 5



Assembly Drawing

Back Seen Through Front

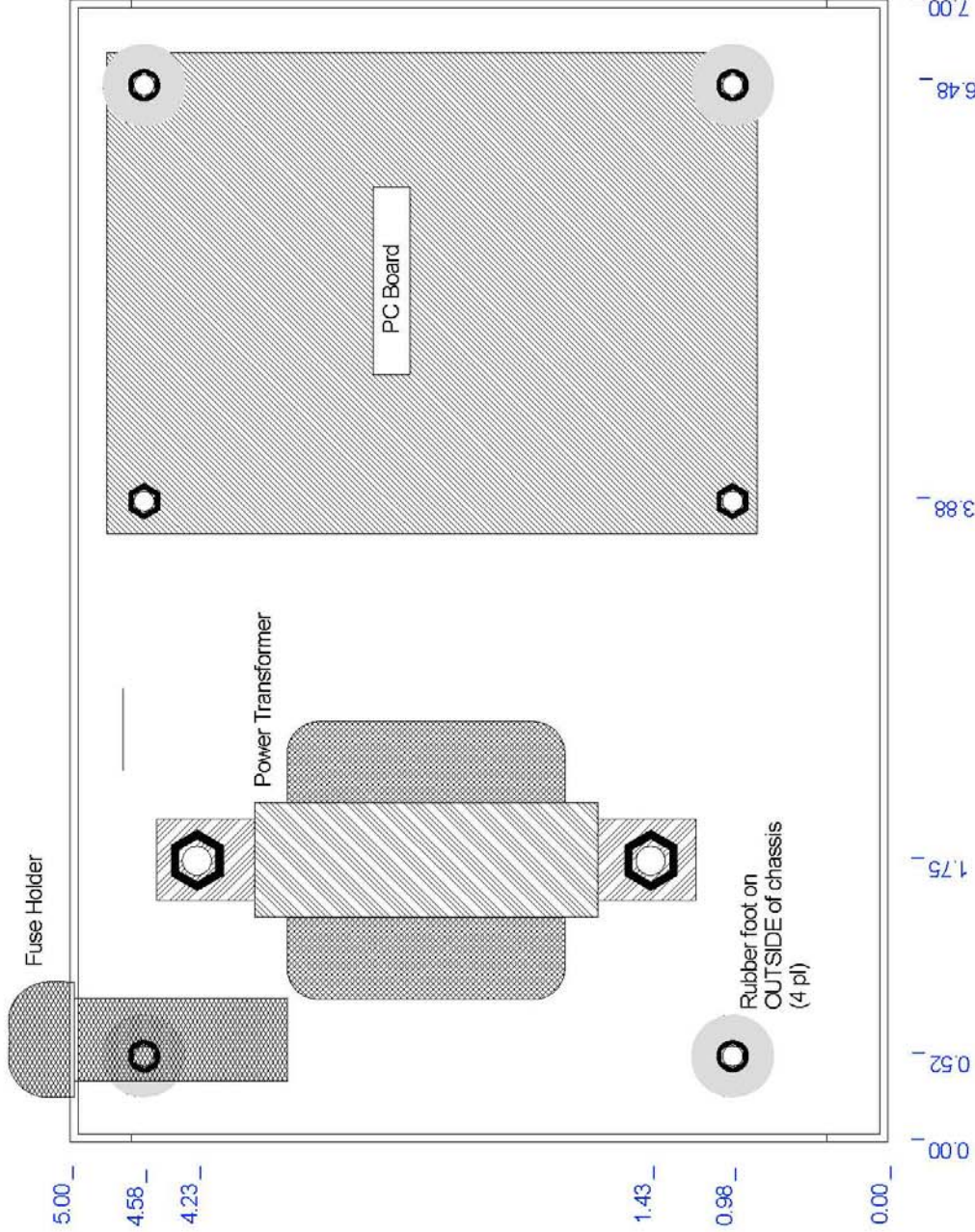
Power Supply Main Chassis

Scale: 1:1
Date: 30 Aug 04

Drawn by: OWJ
Rev: D

Approved by: Jim Weir
RST Engineering
13249 Grass Valley Ave Grass Valley, CA 95945
530.272.2203 tech@rst-engr.com

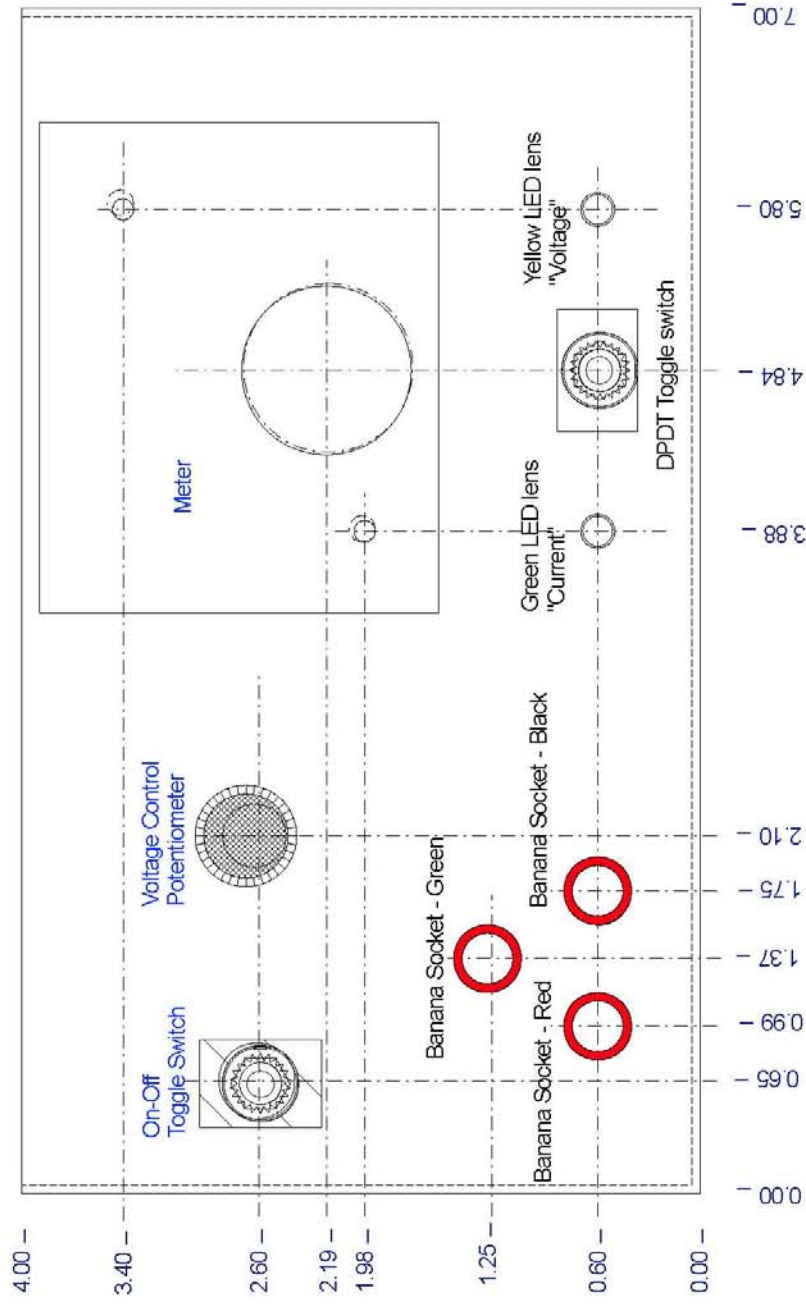
Drawing #
6622-4200
Sheet 3 of 5



Note: Use shrink sleeving on both white and black power cord wires to the MAXIMUM extent possible. Do not expose ANY metal objects to which white or black wires are connected.

Assembly Drawing Bottom View

Power Supply Main Chassis			RST Engineering 13249 Grass Valley Ave Grass Valley CA 95945 530.272.2203 tech@rst-engr.com		Drawing # 6622-4200 Sheet 4 of 5
Scale: 1:1	Drawn by: OWJ	Approved by:			
Date: 24 Oct 06	Rev: C	Jim Weir			



Assembly Drawing Advanced Version

Power Supply Main Chassis

Scale: 1:1
Date: 22 Nov 01

Drawn by: OWJ
Rev: B

Approved by:
Jim Weir
13249 Grass Valley Ave Grass Valley, CA 95945
530.272.2203 tech@rst-engr.com

Drawing #
6622-4200
Sheet 5 of 5

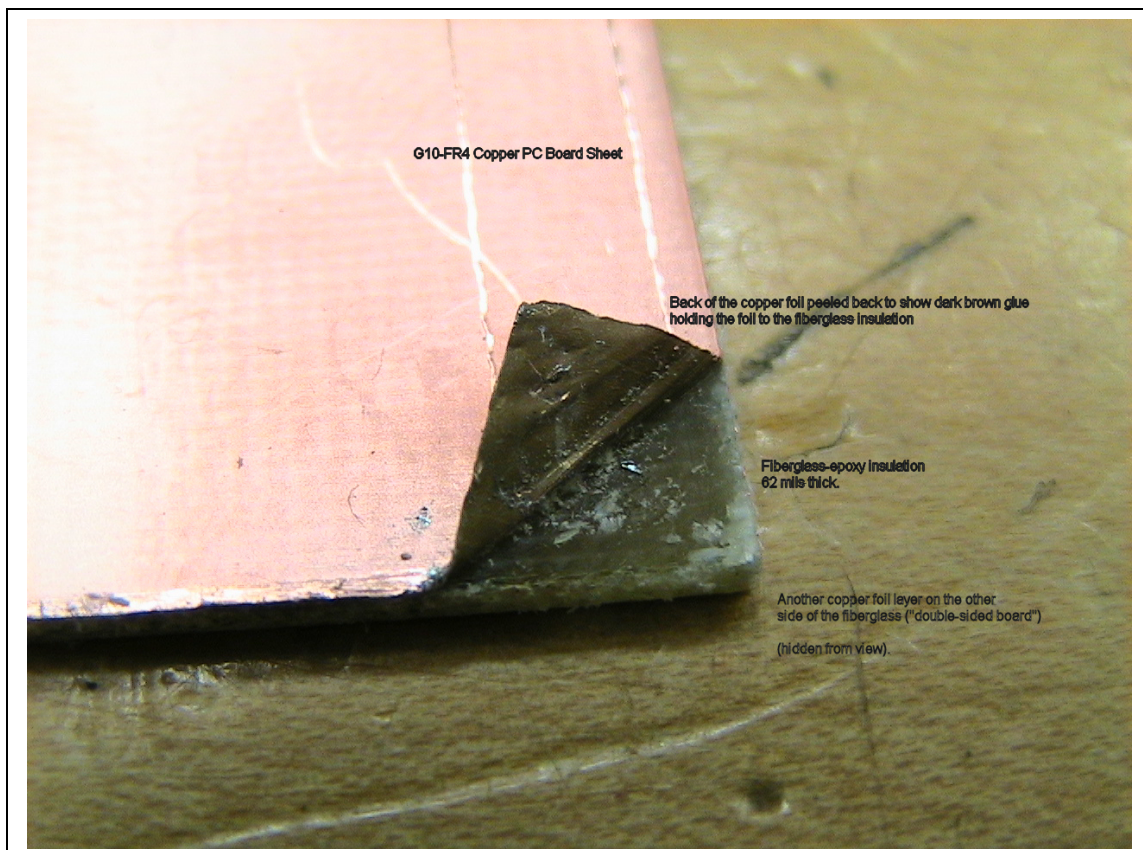
G. Chassis Fabrication

1. Fabricate the paper doll for the marker receiver (RST-521) and compare it to the photographs in this section.
2. Fabricate a paper doll model of the CIE-14 power supply using the bend drawing layout 6622-2250s1 and 6622-2251s1 drawings in this section.
3. Fabricate the power supply chassis from aluminum. You may wish to have a "class collaboration" to make a template for the whole class that all students can use in lieu of everybody making individual measurements on the chassis.
4. Finish the power supply chassis using either paint or anodize and letter the chassis using either iron-on (for anodize) or mechanical tape for paint.
5. Replicate ALL the drawings in this section for fabrication of the power supply. Note that the bend drawings are too large to fit 1:1 on an "A" size drawing (8½ x 11") sheet of paper. Use "B" size paper to do the 1:1 drawing and then use the SCALE command to reduce the entire drawing to 66% (2:3) to fit onto an A size sheet of paper.

VII PCB Fabrication

A. PCB Material

1. Very early in the electronic business, we tried literally to make printed circuit boards. That is, we took an insulator (bakelite was all we had at the time) and tried to print copper wires ("traces") onto the bakelite plastic using metal-loaded ink. The problem was that the ink wasn't rigid enough to survive the slightest bit of flexing of the bakelite, nor did it take well to vibration.
2. The next step in this process was to take that same sheet of bakelite and glue a very thin metal foil to the plastic surface. The desired circuit was drawn on a sheet of paper and a photographic silk screen process made from that paper artwork screened acid-resistant paint onto the metal (almost always copper) foil in the pattern drawn on the paper. The whole plastic-foil-paint combination was then dropped into an acid bath and the unwanted copper was dissolved in the acid. Although technically an "etched board" the original "printed circuit" board name was retained.
3. Over the years refinements have been made to the process but the basic material remains the same -- an insulator covered with copper foil on one side of the insulator (single sided board) or on both sides of the insulator (double sided board).
4. While many insulators have been used over the years for pc board material (phenolic, paper, rubber, Teflon®, Mylar® and other exotic materials) for most boards made today the insulator of choice is fiberglass-epoxy resin. We call this material G-10 or FR-4 interchangeably, although FR-4 is the Fire Retardant grade of G-10 if you want to be technically correct. G-10 comes in many thicknesses, but 62 mils is the "standard" pcb

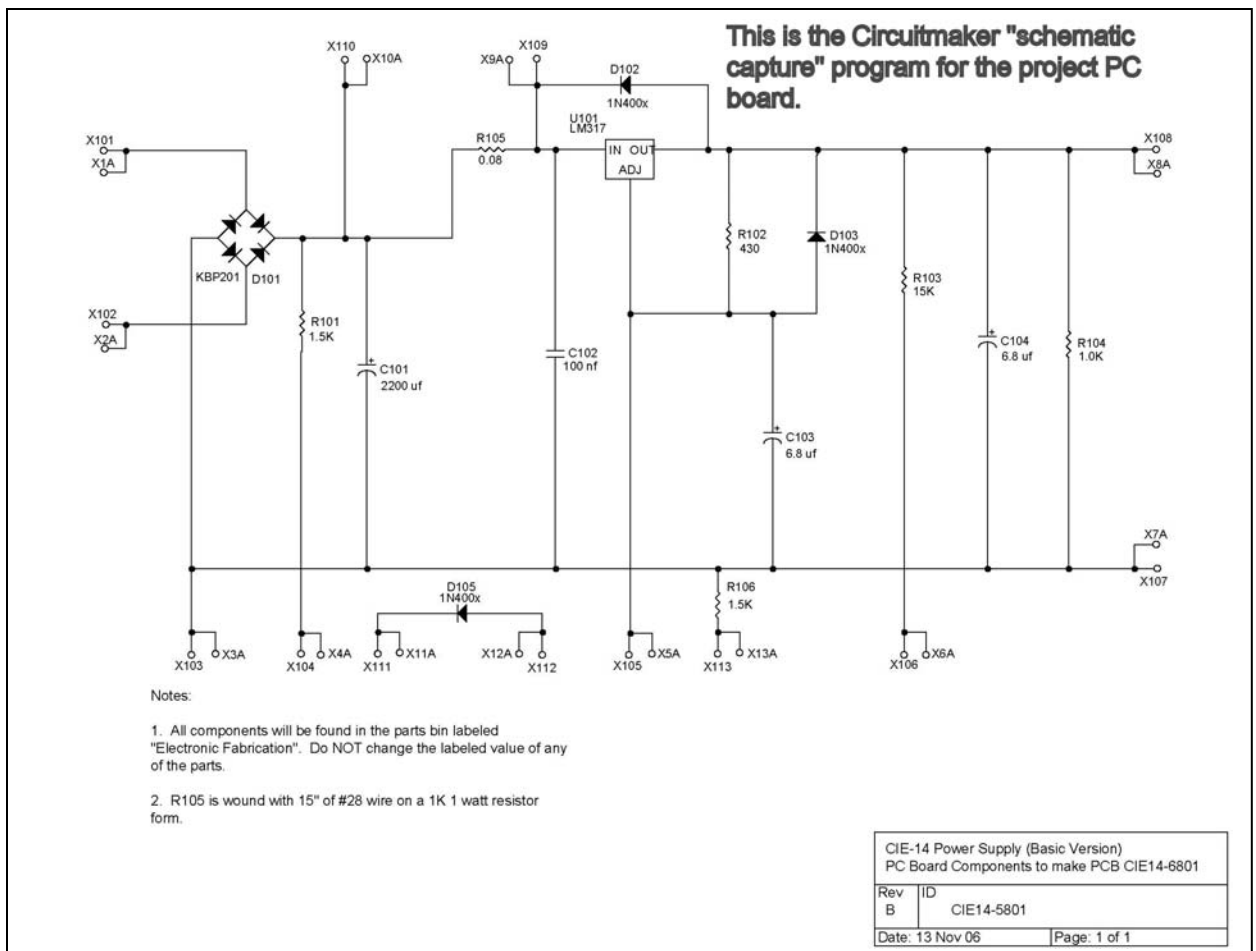


material.

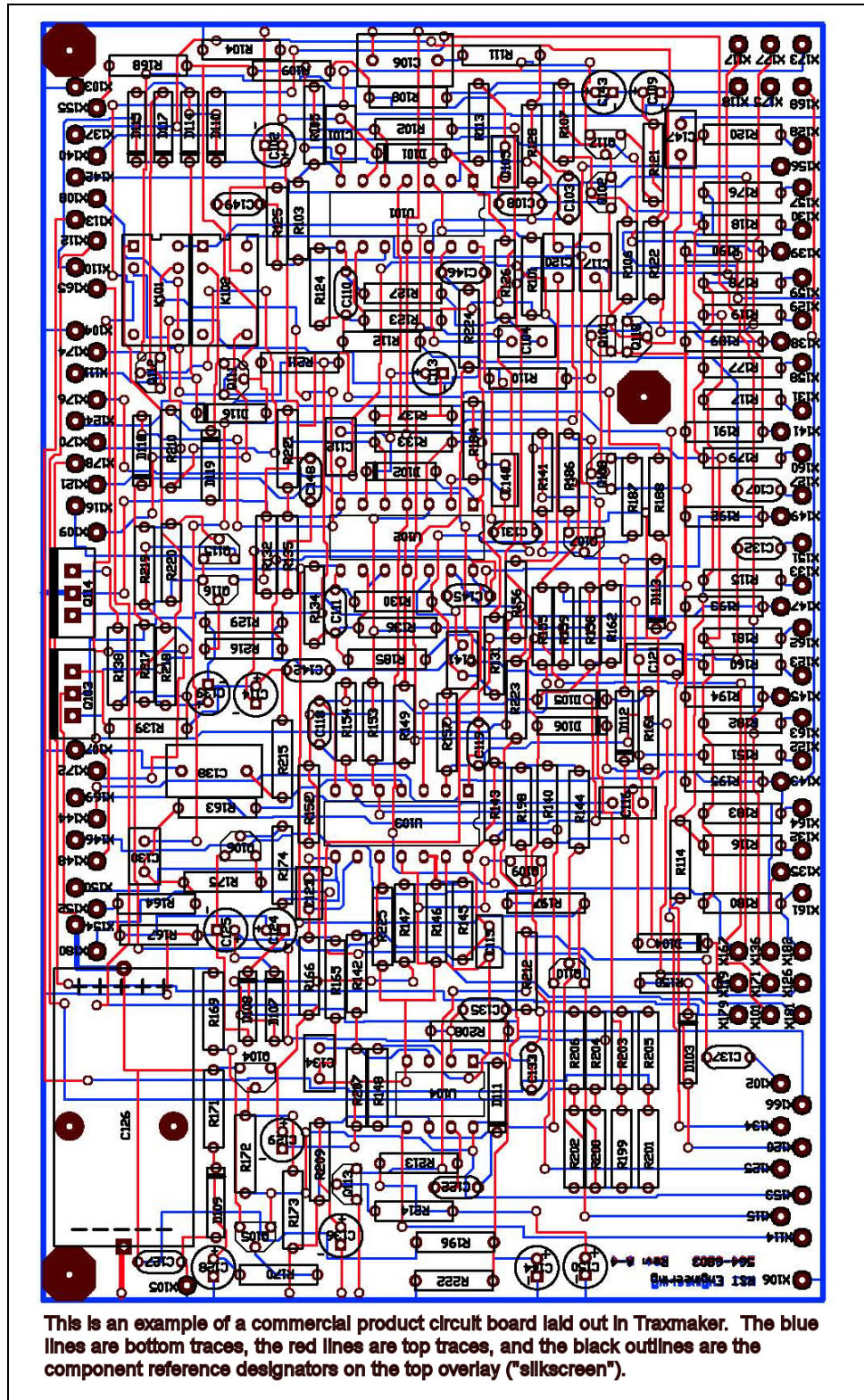
5. Similarly, the metal foil can be any metal, although copper is the universal metal of choice. It comes in many thicknesses but "1 ounce" copper 14 mils thick is the standard.

B. Layout

1. As we noted above, the original "layout" artwork was done using a silkscreen process derived from a paper ink drawing of the desired circuit. A little later on down the development trail we began using clear plastic instead of paper and black tape along with preprinted black tape patterns for the traces and component hole "pads". This latter process was called "tape and donuts" for the shape of the most common pad pattern -- a small black round shape with a clear hole in the middle for a drill bit guide.
2. Along about the same time that tape and donuts came along, also came along a method whereby a photographic acid-resistant emulsion could be laid down on the copper foil, the pattern generated by the artwork could be photographed onto a film negative, and the resultant pattern could be photographically transferred to the emulsion. The emulsion could be developed, the undesired exposed emulsion washed off, and the desired pattern then chemically hardened and tossed into the acid bath to etch off the unwanted copper.
3. When computers came along, we started getting rather sophisticated with how we make pc boards today. It is now common practice to use the same program to draw the circuit (called "schematic capture") and transfer that schematic design directly into a pcb drafting planning tool called a "pcb layout" routine.

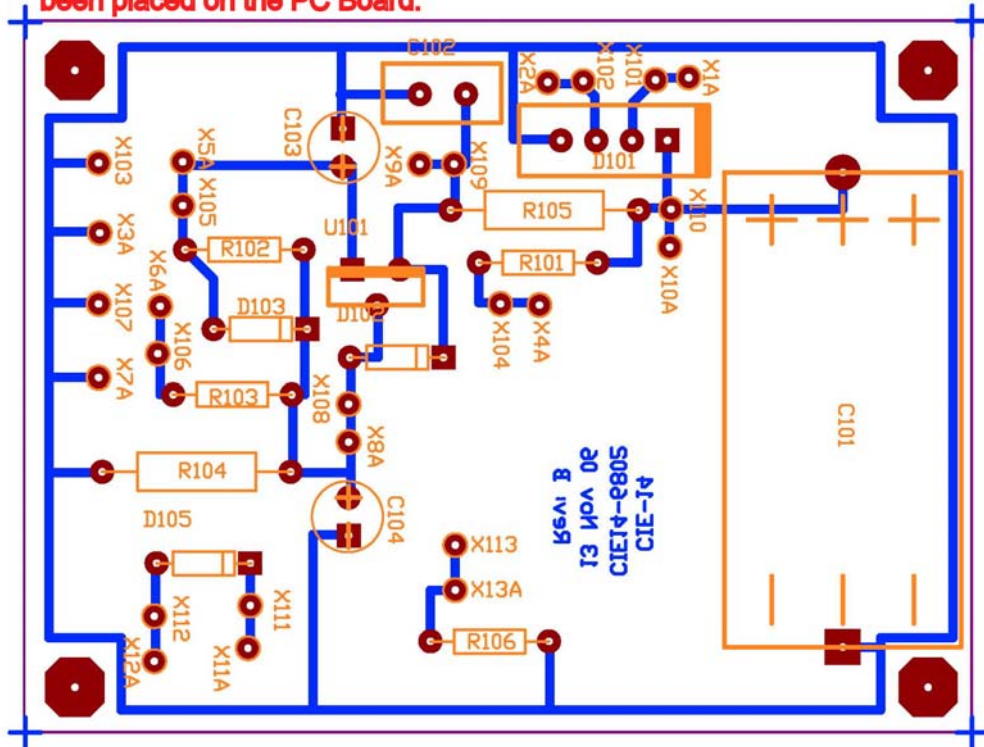


4. A further refinement of the pcb layout routine called "autoroute" will actually lay the components out on the board and automatically route the wire traces between the pads as shown on the schematic. Finally, files called "gerbers" (named for the company that invented them) and numerically controlled drill files (called NC files) are made from the pcb artwork. These files are electronically sent out for manufacture to a company with cameras and automatic drilling equipment that take those gerbers and NC drill files and automatically flash the layout onto the emulsion, drill the holes, etch the board, and it all comes out the other end of the machine never having been touched by human hands.



5. While this process is all well and good for production boards, the home hobbyist, the R&D engineer, or the student who only needs one or two boards for prototype use can still use the computer to do the schematic and pcb layout, but shortstop the process before it gets to the gerber stage and print out the circuit on a plain old laser printer for making boards themselves (see C.4. below).

This is the PC Board made by exporting the Circuitmaker schematic to the Traxmaker layout program. Note that every component on the schematic has been placed on the PC Board.



C. Resist

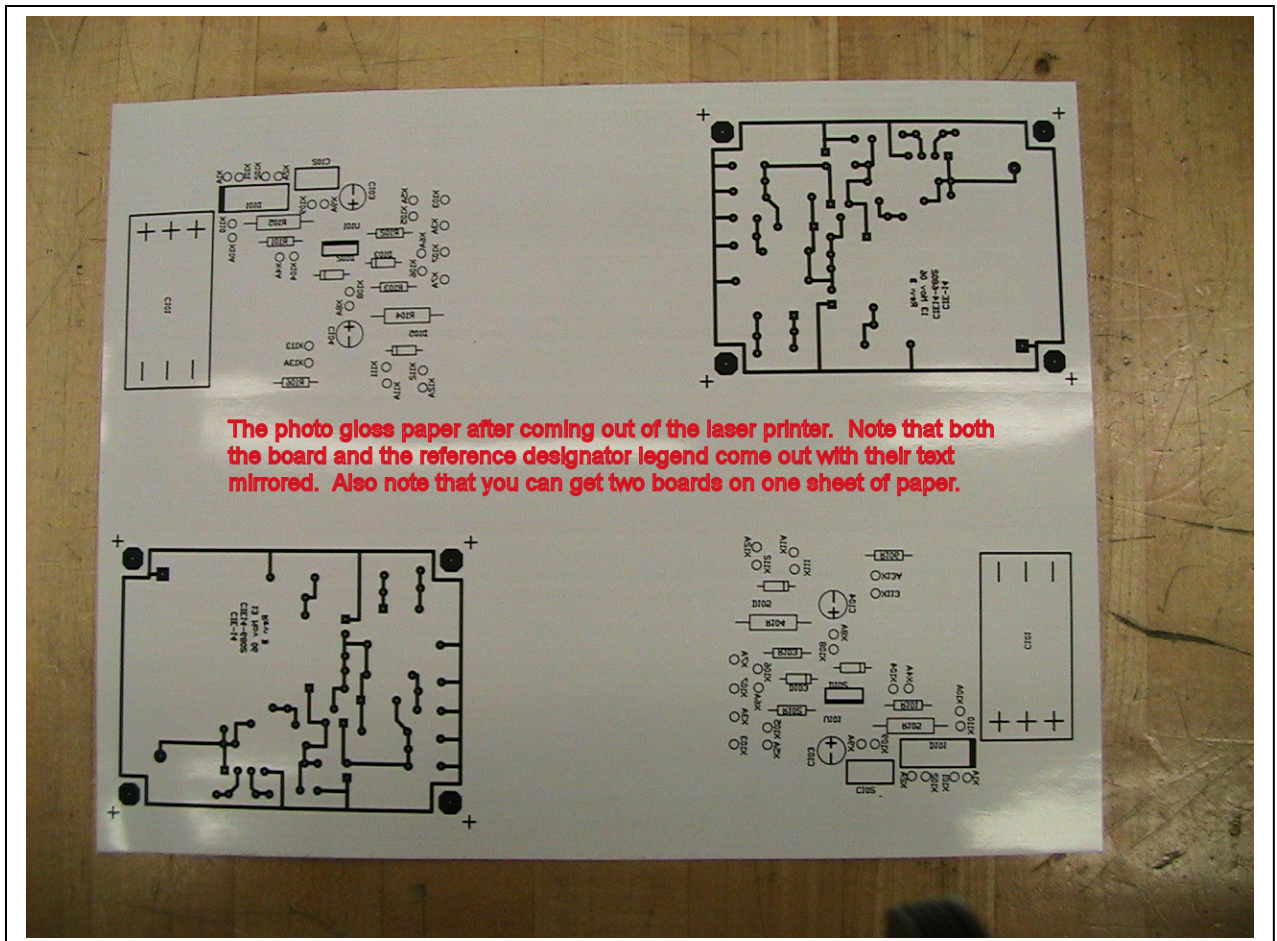
1. Most any organic substance will work quite well for an acid-resistant etch -- paint, photographic emulsion, ink from a "permanent marker", and even good old oil from your hands will work quite well.
2. One of the organic substances that has been used quite successfully in prototype manufacture of pc boards is laser "ink". This ink is actually just a plastic that has been loaded with black dye. When a laser printer "prints", it actually just lays a pattern of plastic down onto the paper and then melts the plastic into the pores of the paper. (Did you ever wonder why the paper comes out of the laser printer warm to the touch?)
3. The process for prototype resist is to print the desired pattern onto a slick paper that does not let the "ink" soak into the paper, then transfer this pattern onto the surface of the foil. Since the plastic ink is impervious to acid, the pc board can then be directly etched with an acid etch. This "paper" is actually a photo gloss paper intended for inkjet printers and with a little work, we can transfer this pattern directly from the paper to the copper surface by remelting the ink from the paper to the copper foil.
4. Here are the steps for making a "laser printer" pc board:
 - a. Clean and microscratch the pc board foil using an abrasive powder copper cleaner ("Copper-Brite®" available at the supermarket) and a heavy duty nylon pot scrubber "green pad". Wash the board thoroughly in warm water. Do not touch the surface of the board from this point until it goes through the acid bath.



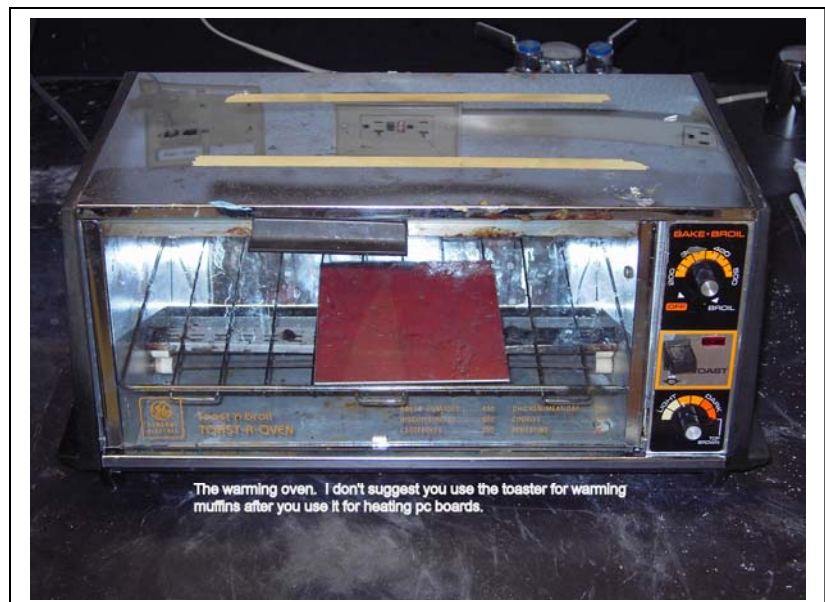
Green rough nylon potscrubber
with "copper brite" powder
underneath it

One of the secrets of getting laser
toner to stick on the copper are the
microscratches and copper cleaner
powder used to make the board
as clean as you can get it

- b. Laser print the desired pattern onto photo gloss inkjet paper. Be sure that the pattern has some mirrored words on it so that you can see instantly "Mirrored" means that the pattern is printed on the paper as though it was viewed in a mirror. When you turn the paper over to put the pattern on the board, this mirroring will be reversed and a non-mirrored pattern will be placed onto the board.



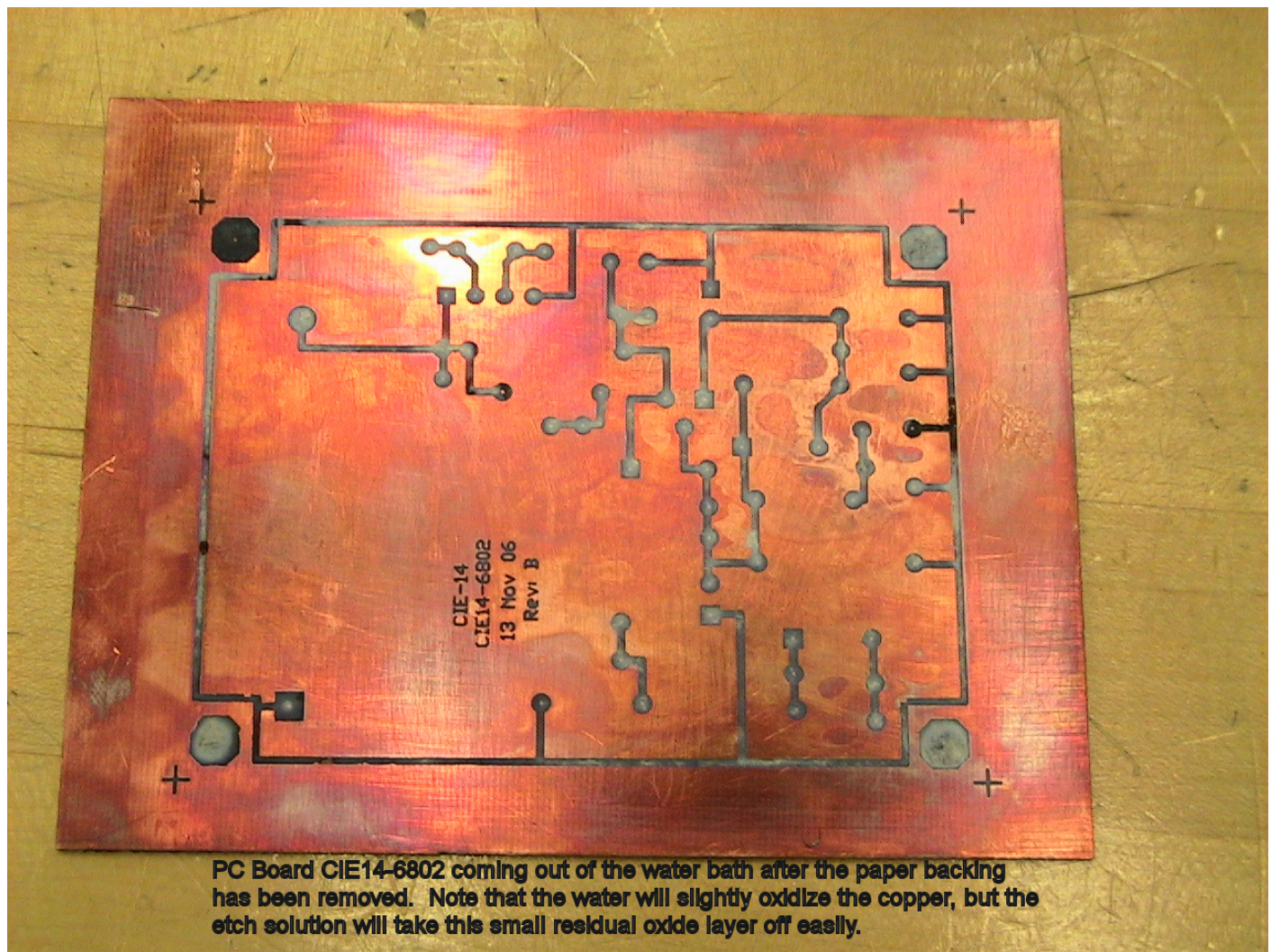
- c. Dry the board thoroughly and place it in a warm oven at about 220 degrees for no more than two minutes (a toaster-oven works well). The board must be perfectly dry for the toner to transfer.



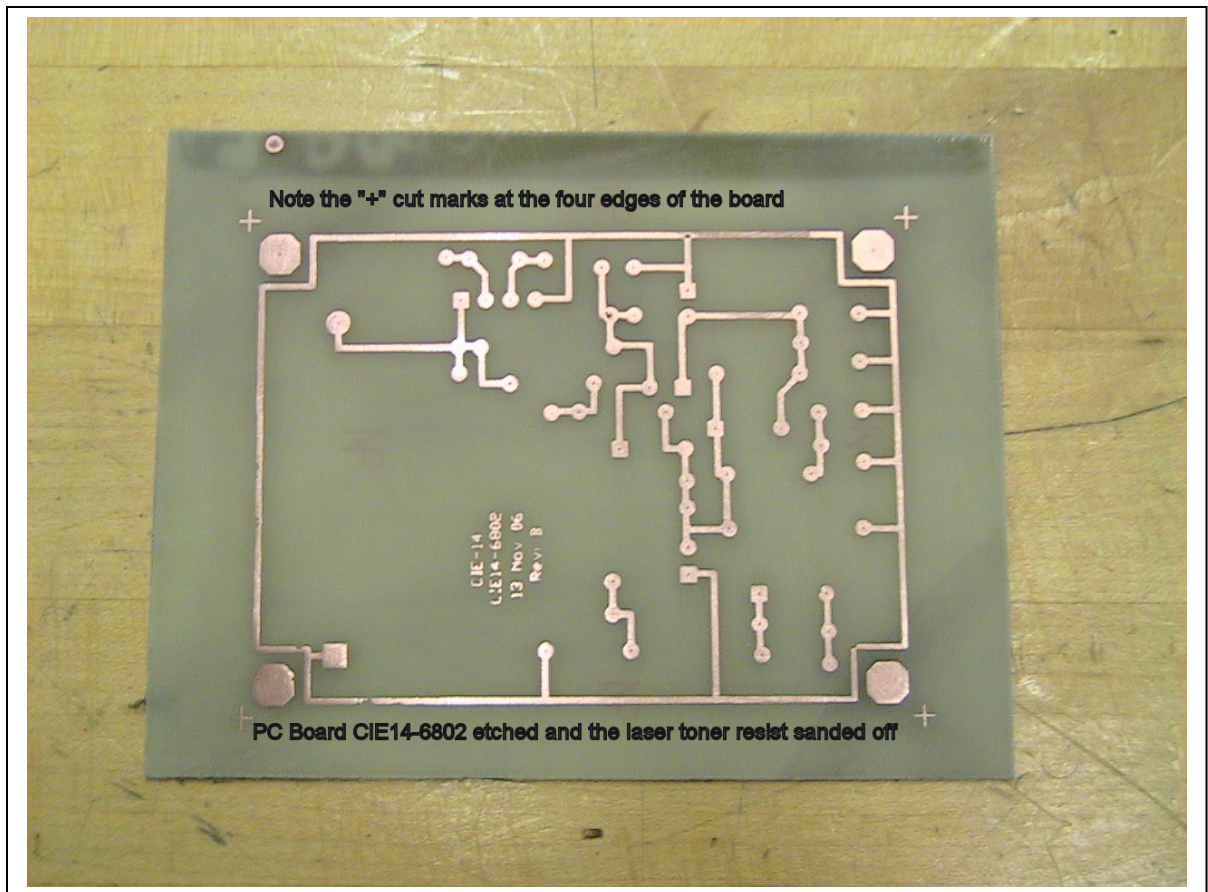
- d. Take the board from the oven and place the laser printed pattern directly onto the surface of the board. Do not allow the board to cool before placing the pattern onto the board.
- e. Immediately take a hot clothes iron set at the highest temperature and begin "ironing" the pattern onto the board using heavy pressure and move the iron like you are ironing a shirt -- back and forth, side to side. After two or three minutes (depending on your particular iron) you should start to see the resist pattern showing through the back side of the photo paper. When the pattern is clearly discernable (no more than 5 minutes) the pattern has been "ironed" onto the pc board.



- f. **CAUTION** the board is very hot. Transfer the board to a warm (NOT HOT) water bath. In about 15 to 30 minutes you will see the paper backing begin to lift from the surface of the board. When the bubbles of paper are almost all across the surface of the board, it is time to **GENTLY** begin to peel the paper backing away from the board (do the peeling under water). This is generally done in two or three stages. First the heavy paper will peel off of the back, then the lighter front coating will peel from the board, then the small bits that didn't peel with the other two steps can be peeled from the surface of the board. Don't worry about any of the paper still stuck to the laser ink on the foil surface. You now have a pc board ready for the etch bath.



- g. After the board is etched, the resist can be removed from the copper by means of 80-120 grit sandpaper (for hand sanding, wet-or-dry generally works best under a warm to hot water bath) . A small electric pad sander will make the job much easier.

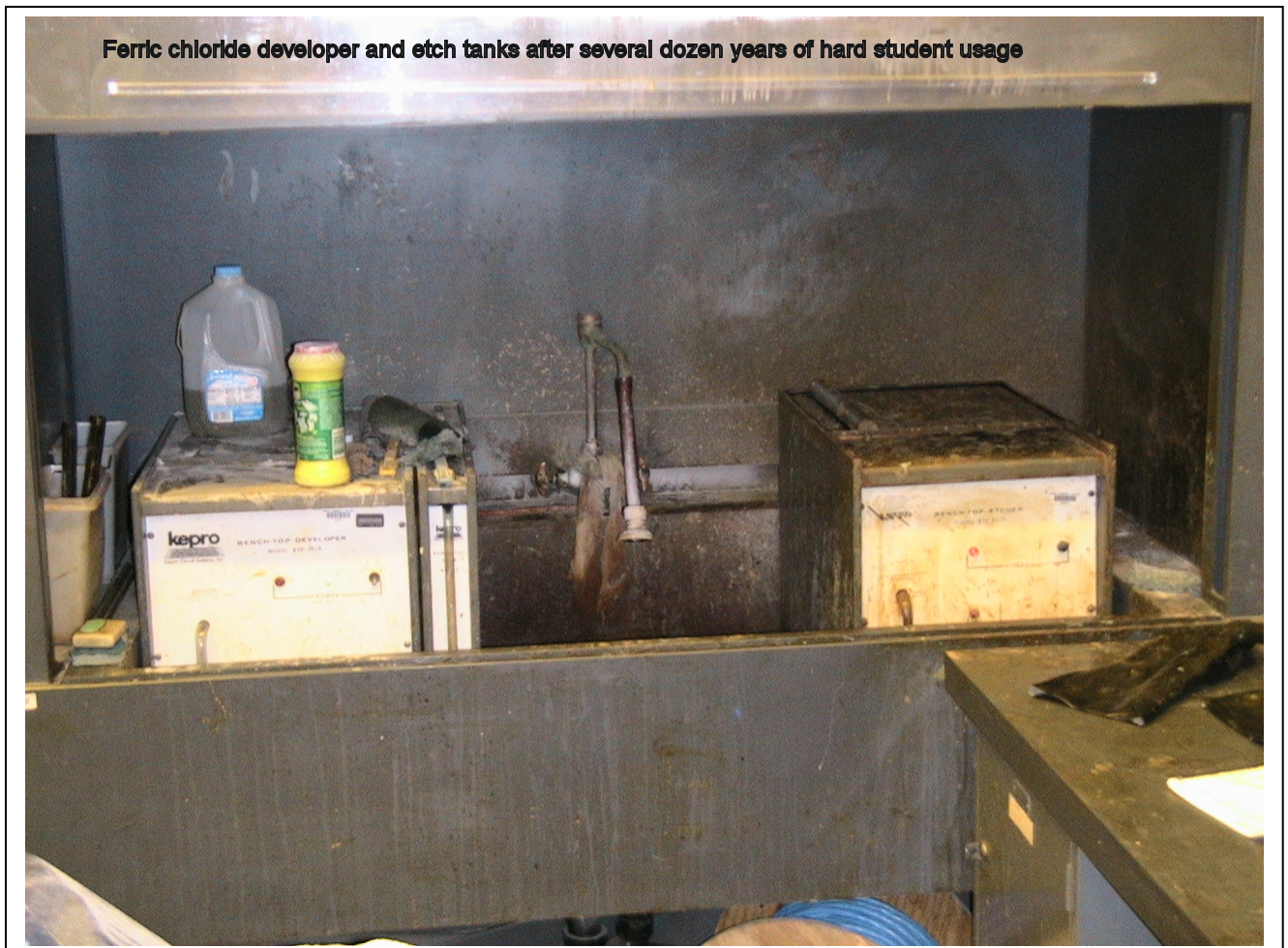


- h. It is possible to make double-sided PC boards this way.
- Before etching, drill two SMALL holes through the bottom resist, the bottom copper, and the top copper surfaces. These two holes should be at diagonal corners of the board to the extent possible in order to make registration easier (you may wish to place special "registration holes" at diagonal corners of the board strictly for this purpose). Make sure that there are overlapping holes on both the bottom and the top resist patterns and that they are exactly aligned.
 - Print out the top resist pattern on photo gloss paper MIRRORED. Make a small pinhole through both of the paper registration holes as close to the center of the hole as possible. Poke a small wire up through the bottom resist and place these pinholes down on the top of the board so that the paper makes good contact with the top copper surface of the board.
 - Thermal management is critical. The idea is to get the top copper/paper as hot as possible without melting the bottom traces. One technique that has proven itself useful in this management is to freeze a piece of aluminum larger than the PC board in the ice chamber of a refrigerator. Lay the bottom side of the pc board on this frozen aluminum while ironing & towel-pressing the pattern

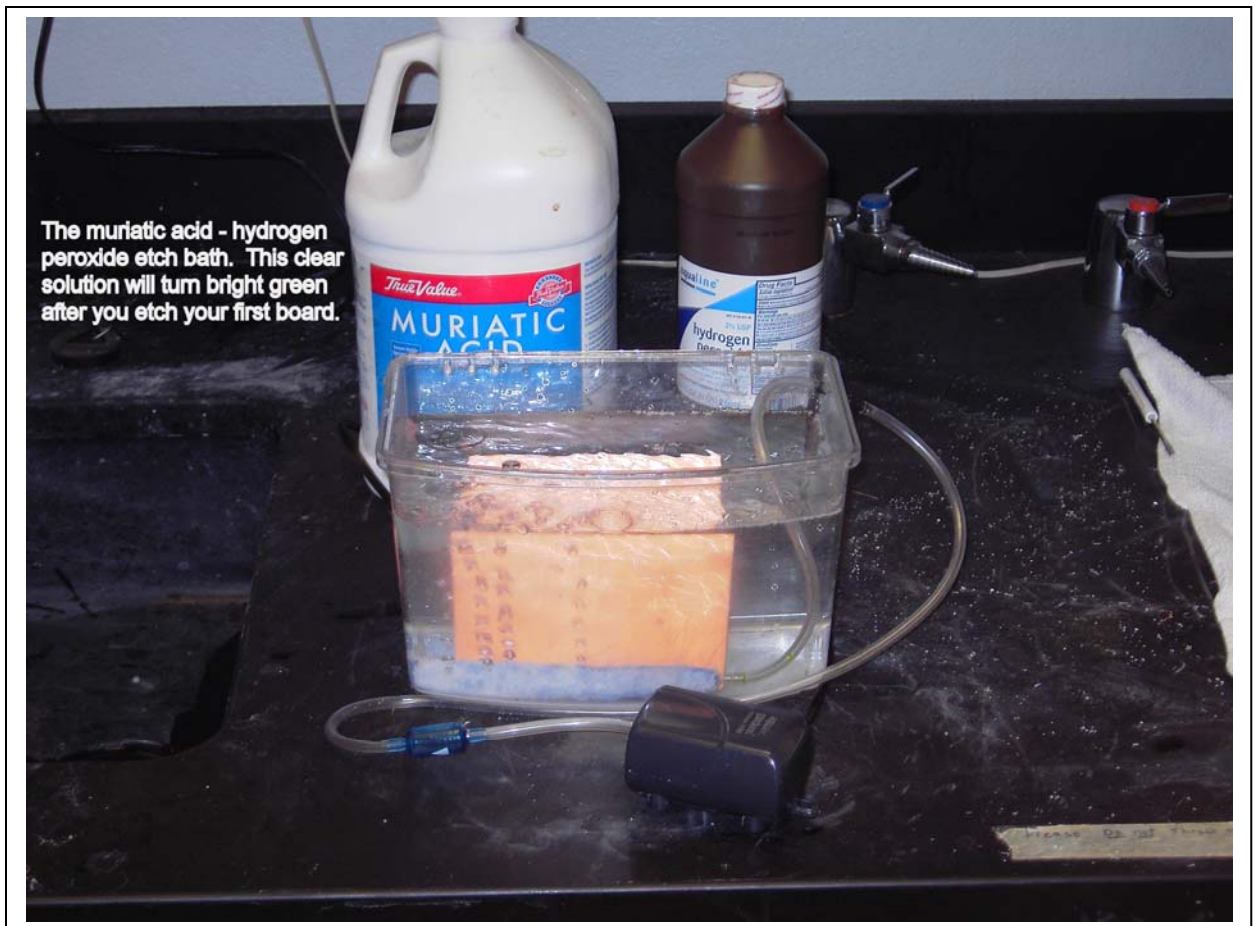
onto the top copper side. Use the minimum heat necessary to transfer the pattern to the top side.

D. Etching

1. Over the years, we've used many substances to remove copper from the pc board material. Three compounds have evolved that have shown themselves to be excellent at this job. They are, in the order most common in industry, ferric chloride (FeCl_3), ammonium persulfate ($(\text{NH}_4)_2\text{S}_2\text{O}_8$), and cupric chloride (CuCl_2).
2. Ferric Chloride is a brown acidic liquid that is obtainable from many of the large electronic supply houses. It readily stains anything organic that comes in contact with it (clothing, skin, etc.) a permanent rather ugly brown. It is considered a hazardous material and commands a premium shipping fee. The good thing about ferric chloride is that it is very fast acting as an etchant and made even faster by the addition of a small amount of heat (no more than 120°F). There is no known "home remedy" to remove the copper from the ferric chloride, so once it is "loaded" with copper it must be disposed of. This is not as simple as it sounds, as loaded ferric chloride is a very toxic hazmat. Some say that you can neutralize the acid with an alkali such as Drano or lye and then use the resulting solution to make a "concrete cage" with common cement. This merely postpones the problem for a few generations until the concrete begins to deteriorate and leaches the iron and copper back into the groundwater.



3. Ammonium persulfate is a white alkali crystal that is mixed with water to form an alkali etch bath. It used to be available from the large electronic supply houses, but has fallen from popularity for some reason. When mixed it does have a shelf life of a few weeks at most. Most of the comments regarding ferric chloride (with the exception of the brown stain) are equally valid for ammonium persulfate.
4. Copper chloride is a good candidate for prototype etching in that it can be obtained at the local home store and is fairly easy to use and dispose of when done. However, the process should be carried out either under a fume hood or outdoors.
 - a. The basic chemicals used in the process are muriatic acid (28% hydrochloric acid rock cleaner from the home improvement store) and hydrogen peroxide from the drug store. (The hydrogen peroxide is just used once to get the process going or to speed the regeneration process.)
 - b. Mix the muriatic acid and hydrogen peroxide together in a glass or plastic container like a fish aquarium. Use four parts of water to three parts of muriatic acid to one part hydrogen peroxide.
 - c. Use a plastic "fish bubbler" and aquarium pump to keep air moving through the etchant during the etching process.



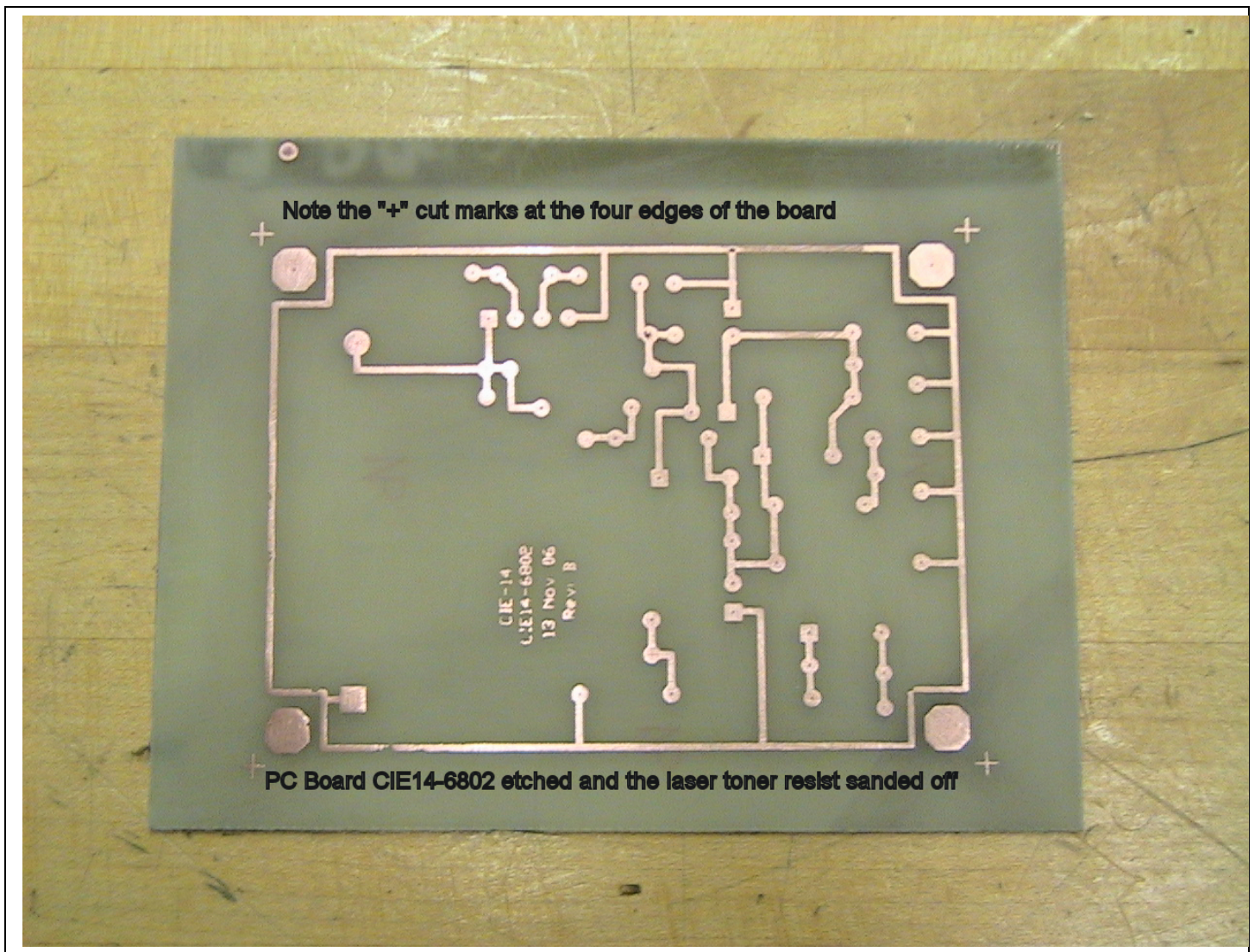
- d. Put the board to be etched into the etch bath. In a short time you should see the copper turning from a gold color to a dull red. In 20 minutes or thereabouts all of the copper not protected by the resist should be eaten away and the etch solution should be either a light emerald green or a bright lime green (or a dark green if you are using continuous aeration). Bring the solution back up to the same level as you started with by the addition of water and acid in a 1:1 proportion or removal of excess etchant (see g. below).
- e. The bright lime green is cuprous chloride which is formed by the action of the hydrochloric acid on the copper. However, continually bubbling air through this cuprous chloride will oxidize it to the emerald green cupric chloride, which can be reused to etch another board, form the cuprous chloride, be reoxidized back to cupric chloride and so on indefinitely. When etching a square 6" x 6" board, the cuprous chloride vigorously agitated by a fish bubbler will be oxidized back to cupric chloride in a few hours.
- f. If you are using a bubbler during the etch process, and especially if you are doing multiple boards in one session, the color will never get from the emerald green to the lime green stage. Instead, the solution will continue to darken until it is nearly opaque and the etching process will slow greatly. At this point, the addition of hydrogen peroxide as an oxidizer will return the solution to the bright emerald green state and the etching will continue at a normal rate.
- g. The problem is that after a few dozen etches, you will have more solution (due to the water and acid addition) than you started out with. However, the solution may be easily disposed of by simple neutralization and metal plating. While a discussion of neutralization is beyond the scope of this text, you will find that common drain cleaner ("Liquid Drano®") will work well to neutralize the chloride acid and common swimming pool testers ("Phenol Red") will indicate a neutral solution. Then a steel wool pad immersed for a day or so in the neutralized solution will cause the copper to plate out on the steel wool, which may then be disposed of as scrap metal. The now neutralized and deionized liquid solution (mostly common salt water) is relatively benign.
- h. Or, you can simply store the removed cupric chloride in plastic or glass jugs and donate it to somebody else who wants to start making their own pc boards. .

Within five or ten minutes, the hydrochloric acid and hydrogen peroxide will start to form the emerald green cupric chloride solution.



After etching three or four boards with continuous aeration, the solution will turn dark green and never get to the lime green state.



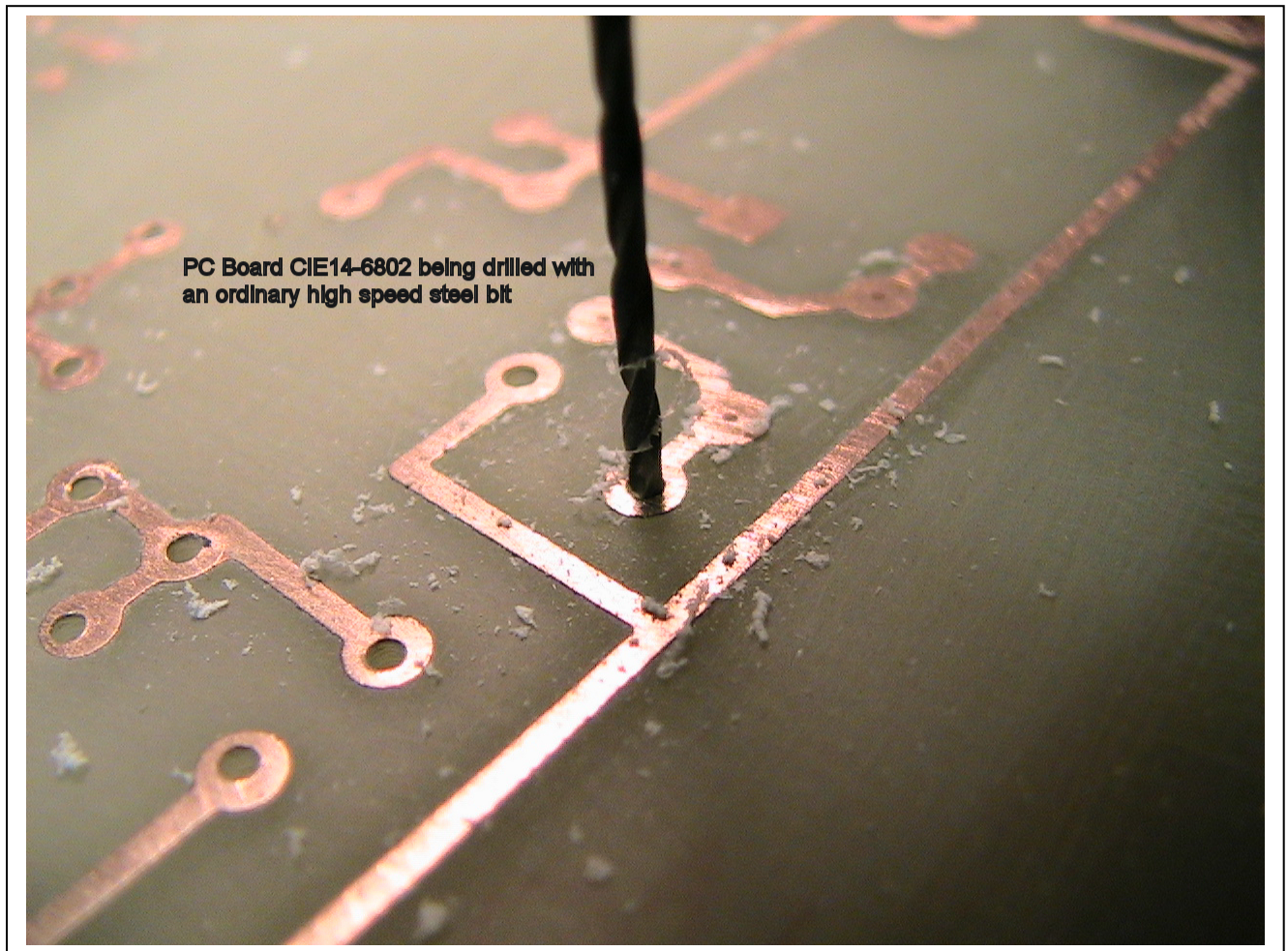


E. Drilling

1. In general, you can get by with three or four sizes of hole. Small parts (quarter-watt resistors, transistors, through-hole ICs, up to #24 wires, etc.) can be done with an 0.035-0.041" drill. Larger components and bigger wire can be done with an 0.045-0.051" drill, and mounting holes (#2 or #4) can be done with an 0.120" drill. Sacrifice a relatively loose ("sloppy") fit for a minimum number of bit sizes -- the solder will easily fill up a 10 mil annular gap.
2. You can use plain old high speed steel ("hss") bits if you wish, but the fiberglass will make short work of them. You might get a hundred holes with a single hss bit. A carbide bit is far more expensive, but will last a thousand holes or so.

One tradeoff you may wish to consider is that a relatively inexperienced person, especially with a manually operated drill press, will tend to snap the more brittle carbide bits more often than the softer yet more pliable hss bits. Experience has shown the cost to be roughly equal between a student using either hss or carbide bits. You can tell when a hss bit is nearing end of life by examining the copper around the hole -- a dull bit will tend to give a burr much like

a hole in aluminum will give a burr with a dull bit. (The technique for removing pcb burrs is identical with aluminum -- a larger bit will remove the burr quite nicely.)



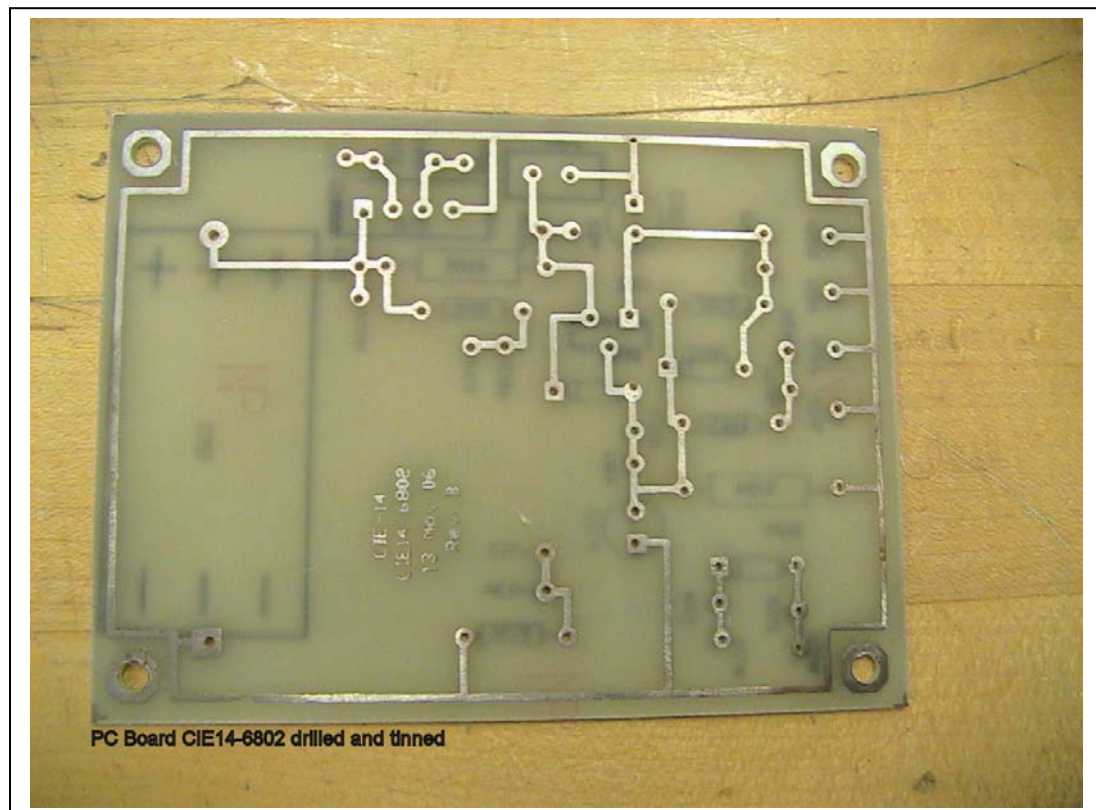
3. You can use a normal "metal drilling" drill press on its highest speed, but for better hole drilling in pc board material (less burrs, better centering, etc.) a high speed drill press meant specifically for pcb drilling is recommended.

F. Plating

1. PC board copper, left to itself, will oxidize and sulphate when left exposed to air. While this oxidation and sulphation does not materially affect the conduction of electricity of the underlying copper, it does make soldering a challenge. It is also true that bare copper needs to be "tinned" by the action of the solder and flux when making a joint directly onto copper pads.
2. Several companies make tin plating solutions that plate by simple immersion at room temperature (Google "Tinnit"). The tin plating thus formed is very thin (0.1 mil or so) but it will keep the copper from oxidizing plus give a very good surface for soldering.



3. One problem with simple immersion plating is that the tin, too, will form an oxide after a while. While flux will strip tin oxide, it is best that the board does not sit around unsoldered for a couple of weeks. One good rule of thumb is to tin plate the day you are going to

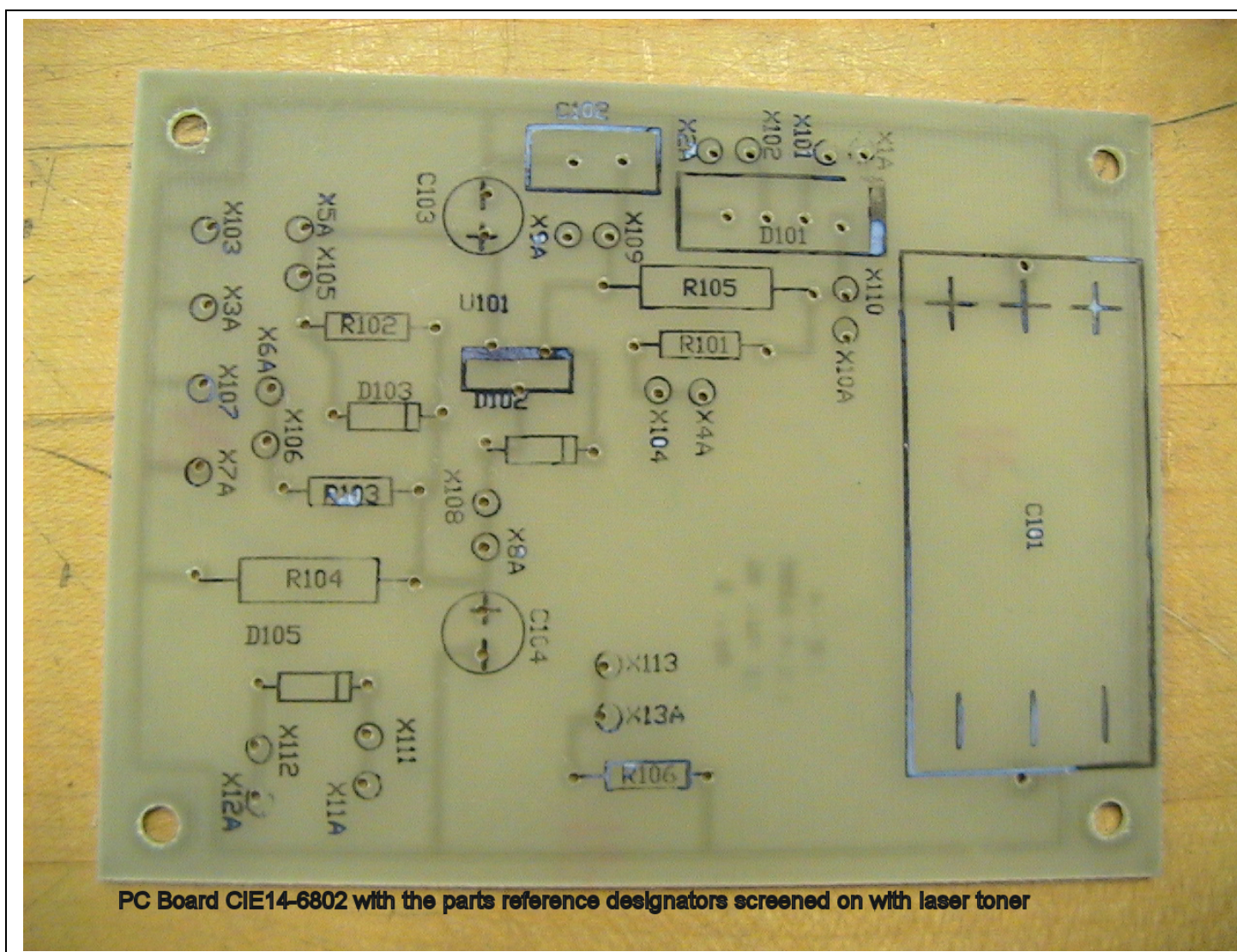


PC Board CIE14-6802 drilled and tinned

assemble the board. If you do happen to forget and leave a tinned board around for a couple of weeks before assembling it, you can strip the tin oxide with a green potscrubber and abrasive kitchen cleanser powder.

G. Labeling

1. Labeling the component side of the PC board will put a "legend" or physical representation of the components in their proper place on the board. This is achieved by doing a 100% (1:1) printout of the "Top Overlay" on the same type of photo gloss paper that we used for the resist step above and then transferring this laser toner pattern onto the top (component) surface in exactly the same manner that we ironed the resist pattern onto the copper side of the board.

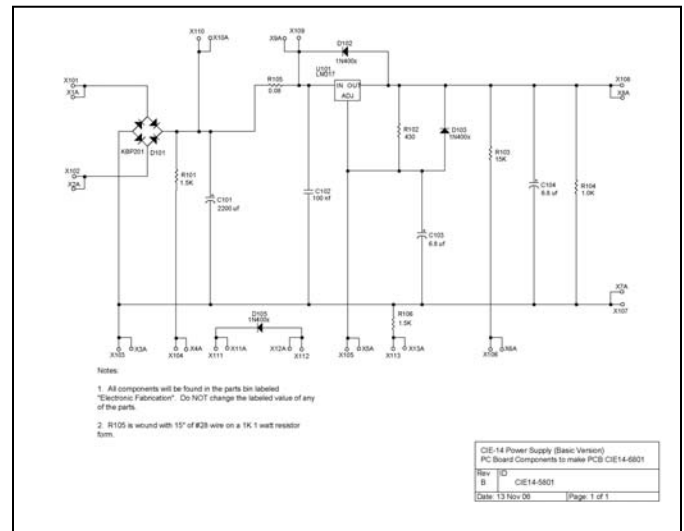
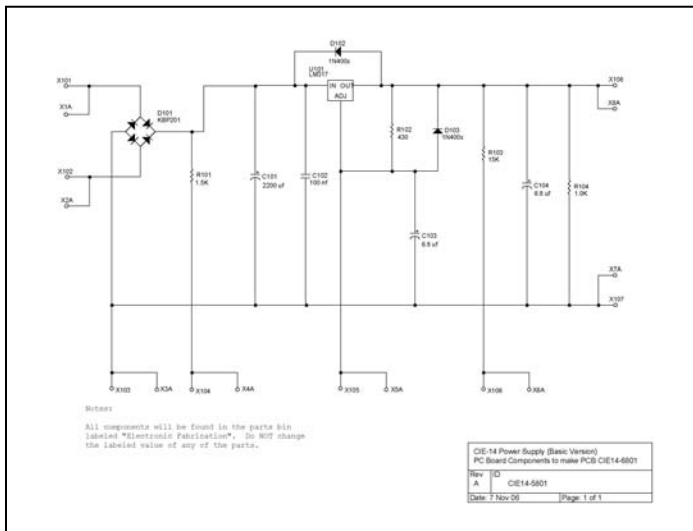


2. Set all components for Label-Value OFF and all Designations ON. Print the Top Overlay MIRRORED 1:1 onto photo gloss paper.

3. After the board has been etched, drilled, and plated, see the instructions for "double sided board" in section C.4.h above and iron-towel the legend onto the top side of the board.
4. CAVEAT -- to the extent of the measuring equipment and the types of laser toners that we have used here, we have determined that laser toner is nonconductive. If your double-sided high-impedance or high-frequency circuits are not acting "normally", etch a board without top labeling and see if perhaps your brand of laser toner is not acting like our tests have shown.

H. Project PC Board.

1. Use either schematic CIE14-5801.sch (basic model) or CIE14-5802.sch (advanced model). Using the supplied schematic as a guide and the components in the library "Electronic Fabrication", draw your own schematic. Use the title block to give it your own particular ID (xxxx-5801.sch or xxxx-5802.sch, where xxxx is the 4-digit number you have chosen as your own).

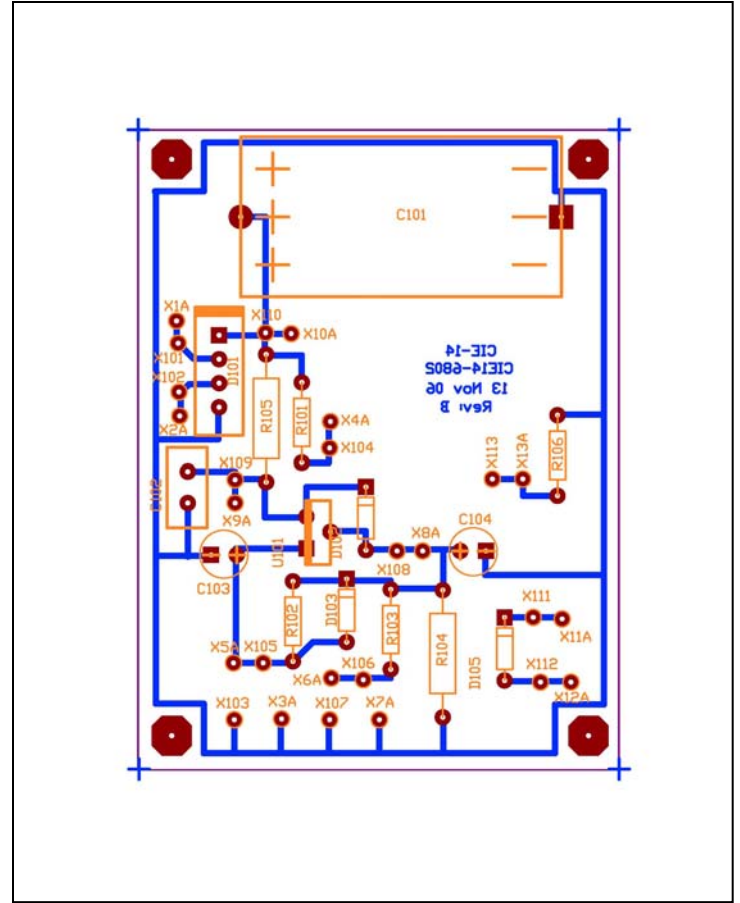
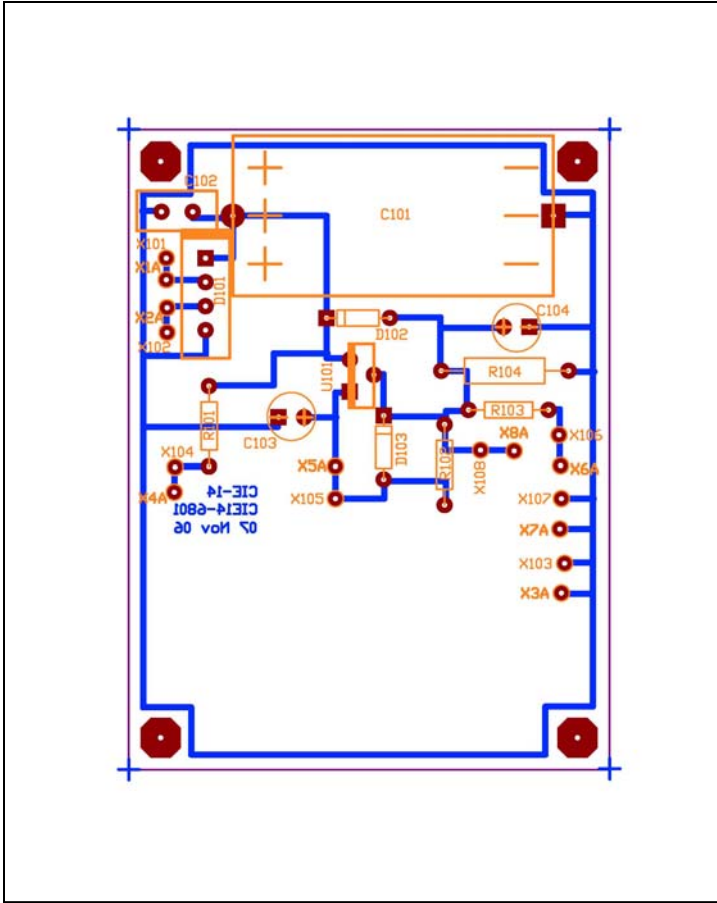


All of the parts for the schematic will be found in the Circuitmaker parts bin "Electronic Fabrication". Do NOT use parts from any other bin or change the values of the parts you take from the Fabrication bin .

Have the instructor check your work before proceeding.

2. Export (File, Export) the netlist. When the netlist screen appears let the screen load Traxmaker with a board size X=3000 Y=4000. Let Traxmaker automatically place the parts.
3. Move the board and parts by dragging a selection box around the entire board and moving the entire board and parts up and to the right about two or three inches.
4. Using the CIE14-6801 (or 6802).pcb file as a guide, make provision for mounting nuts by placing an OCT 250 pad two tenths of an inch away from each edge in the corners (4 places). Use the MEASURE tool to measure center to center distance between the nuts. Cross check this measurement with the mounting holes that are already drilled on your chassis.

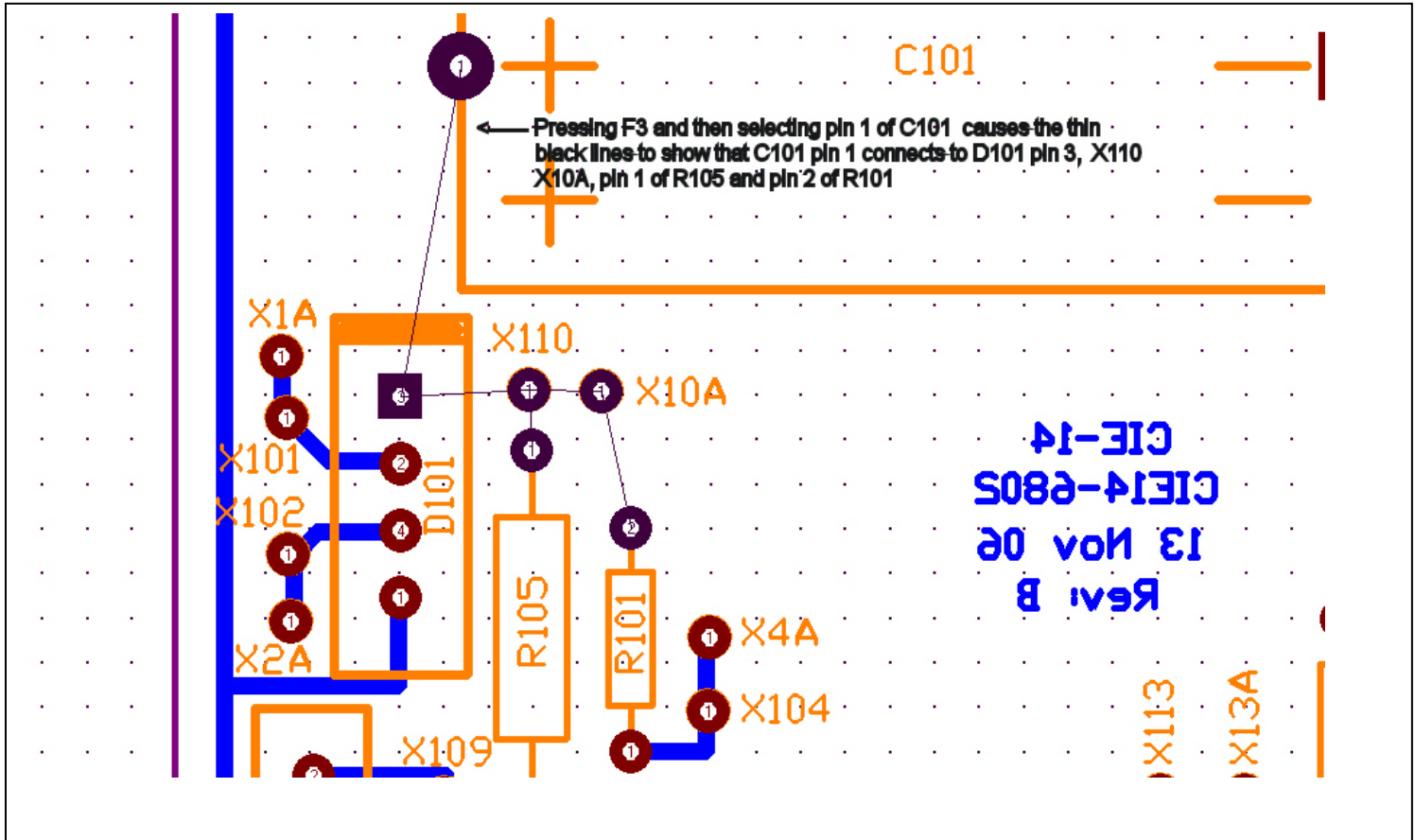
5. Run a blue (bottom) trace around the outside of the board as shown.



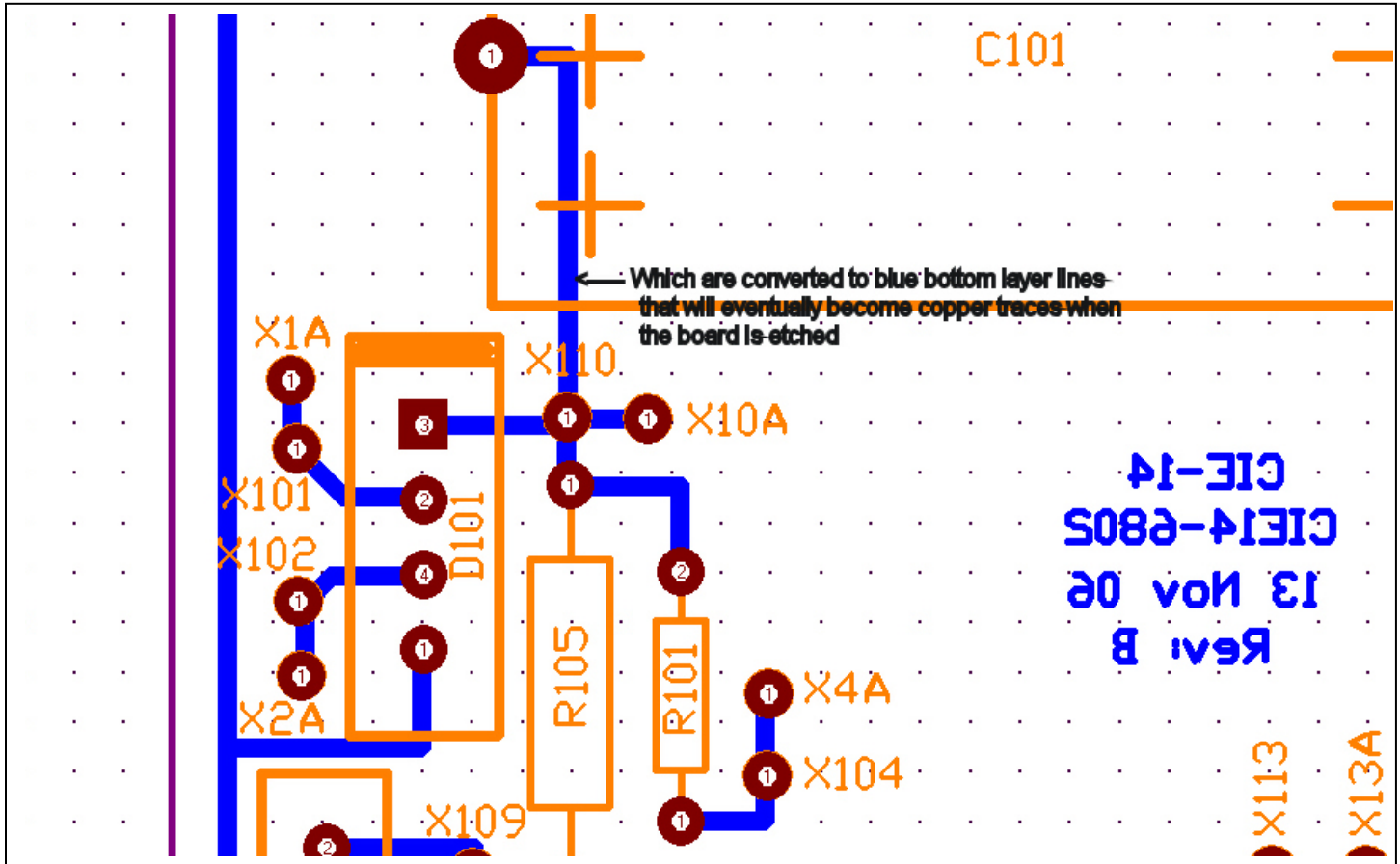
6. Position the parts inside the keep out layer and inside of the blue trace you laid down in step 4. Note that in the sample .pcb file that the parts are all "jammed" up into the upper left hand corner. You may make it much easier on yourself later when assembling the board if you spread the parts out fairly consistently across those areas of the board that have no parts on the sample.

The only part that is somewhat critical for placement is bridge rectifier D101. It should be somewhere along the left edge of the board to mate up with the power transformer leads coming from the chassis (see chassis schematic CIE14-2901 or 2902.sch for the chassis diagram).

- Using the F3 function key, select a pad and notice that it tells you what other parts or pads are connected to that pad. Move the parts and pads around until you can easily interconnect the pads with traces. (Note that if you rotate a component, you will have to separately rotate its reference designator so that you can read all the reference designators with either the bottom or the right edge facing down.)



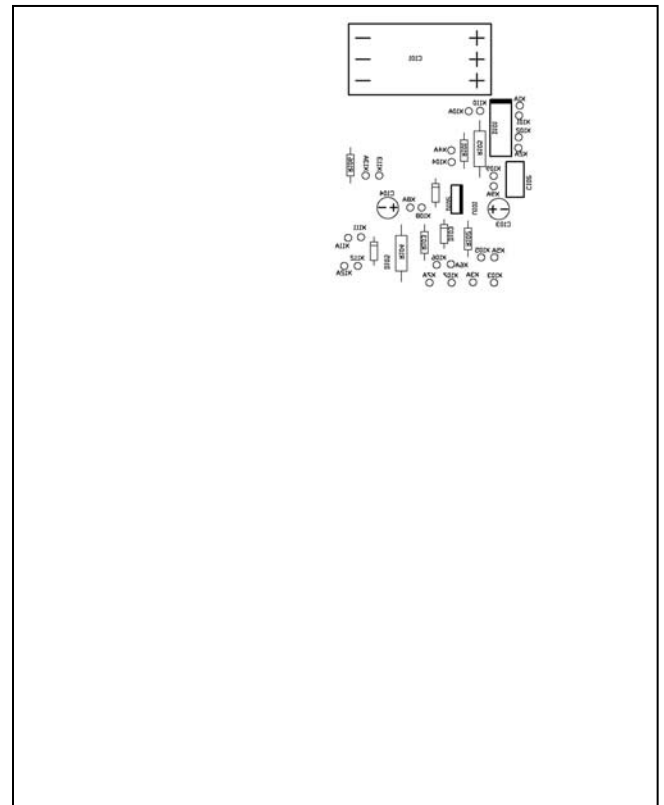
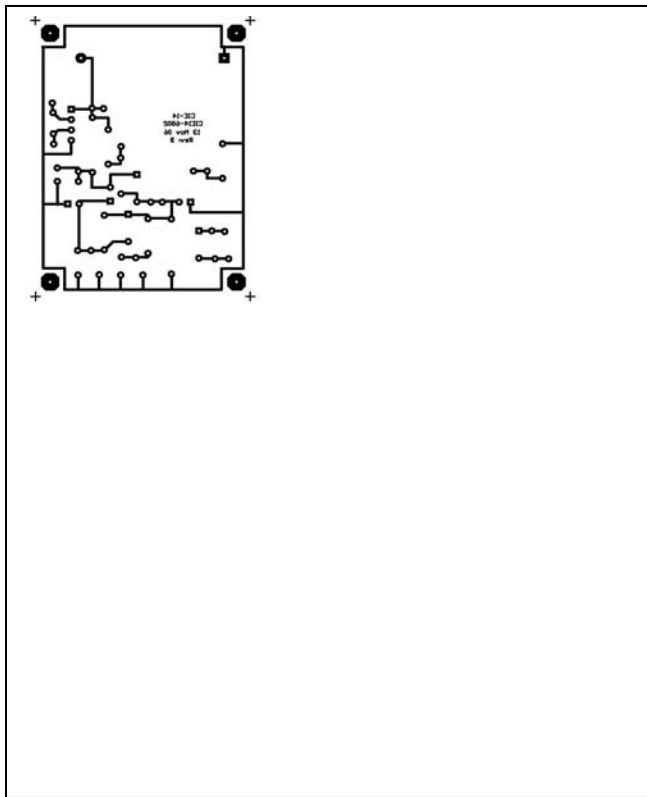
8. Run the traces as the F3 "show nets on selected pads" key tells you to. When you are completed running traces, have the lettering completed, and have the corner shear marks placed, do a "Design Rules Check" with the Netlist, Design Rules Check. Investigate any errors that the DRC routine tells you about.



2. Resist

1. We are going to use the "photo paper - laser printer" method of resist application. For this application, print the bottom traces as follows:
 - a. Go to "File", then "Print". Set the "X Scale" to 1.000 and the "Y Scale" to 1.000 . Set both offsets to zero.
 - b. "Position" is Normal, "Quality" is Final.
 - c. Open "Options" and set as follows -- all unchecked except Check Print, Bottom Layer. Include Pads and Single Layer Pad Holes. Other Settings Pad Hole Guide Size 20 mils and the other 3 don't matter. Color mode Black & White. Then OK

2. Make a TEST PRINT on regular paper. It should come out something like this (on the left):



3. Reopen "File", "Print". Leave everything else the same except "Options", Check Print **MIRRORED**, uncheck Bottom Layer and check Top Overlay. Then OK.
4. Choose an X-Offset of 4.500 inches and reprint **ON THE SAME SHEET**. What should have been added to the sheet is the top overlay components on the right..
5. When you have these two items printed correctly on the same sheet of copy paper, then it is time to show the sheet of paper to the instructor. Once the instructor agrees that this is the correct format, then repeat steps 1-4 above on Photo Gloss (fairly expensive) paper. Wait for another class member to be done with their circuits, turn the photo paper upside down and use the other half of the paper for the other class member.
6. Cut the individual quarter pages with scissors, leaving the most paper margin that you can outside the boundaries of the circuits.
7. Cut a piece of pc board material approximately 4 x 5 inches. Thoroughly clean both sides of the board with a green pot scrubber and copper cleaner. Wash thoroughly with water and dry using paper towels. From this point on through etching, your hands should never touch the top or bottom sides of the pc board.
8. Warm the board **SLIGHTLY** in the 120° F heating oven for no more than two minutes. Place the bottom trace image face down on the pc board and iron the image onto the pc board using the iron on the highest setting. After one minute of ironing, take a doubled-over cloth towel and press the paper onto the surface for half a minute using the heel of your hand for

pressure. Repeat the ironing-pressing routine two more times (one minute of ironing and half a minute of pressing).

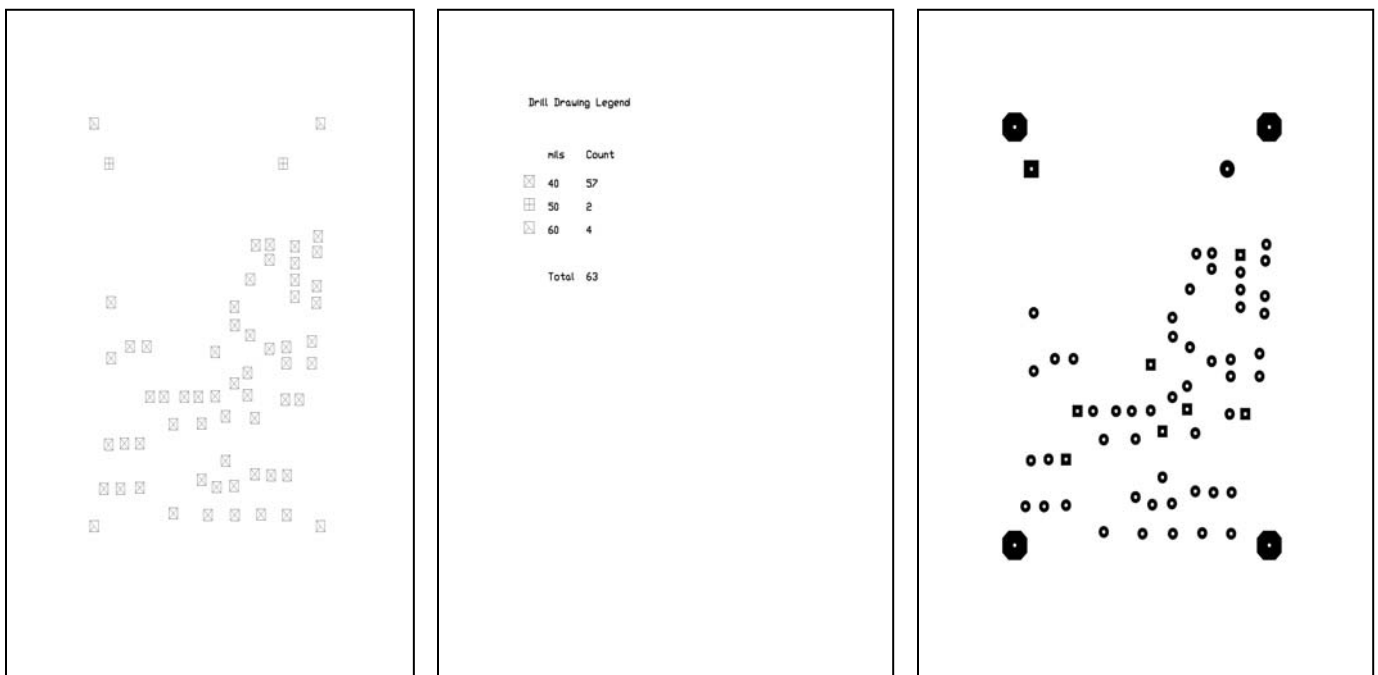
9. Soak the pc board in warm water until the paper backing starts to peel off of the traces using gentle hand rubbing. It is generally a three step process -- one layer of paper, then another layer of paper, and finally the bottom layer of paper. A 5-minute soak between layers should be adequate in warm water.
10. Remove the board from the water and dry it thoroughly. Inspect the board to see that all the traces transferred from the paper to the board. If there are any traces missing, you will have to repeat this entire section with another piece of pc board material and another set of images. If there are only a couple of minor trace breaks, you can repair the breaks using a fine tip permanent marking pen.

3. Etching

1. Etch the board in either ferric chloride or copper chloride. If you are using the "homemade" copper chloride method, drill the four corner holes 0.12" in diameter and use nylon standoffs in each corner before etching. (Center punch the holes before drilling.)
2. After etching and thoroughly washing the board, remove the resist using 300-400 grit carborundum paper.

4. Drilling

1. In Traxmaker, File, Print, set the X Scale and Y Scale to 2.000 and the offsets back to zero, Position Centered. In Options, Batch Print, nothing check boxed except Drill Drawing (mirrored), Drill Guide (mirrored), Pads, Vias, Single Layer Pad Holes. Drill Draw Symbol Size 100 mils.
2. You will get three sheets out that will look like this:



3. The sheet on the left is called a "drill drawing". It shows the location of all the holes that need to be drilled. The sheet in the middle is the "drill guide". It shows the size of all the holes that need to be drilled (see note). The sheet on the right is called a "pad master". It shows the shape and location of all the pads on the board.
4. You will note that there are 57 holes that need to be drilled with a 40 mil bit, 2 that need to be drilled with a 50 mil bit, and 4 (in error) that it says to drill with a 60 mil bit -- NOTE, these 60 mil holes actually need to be 120 mils.
5. When you are through drilling, wipe the board thoroughly with running water and dry it with a soft towel.

5. Plating

1. Plating in Tinnit® puts a thin coating of tin over the copper. However, the board needs to be thoroughly cleaned and the copper bright and shiny for the tin to plate out. Clean the board with a green potscrubber and copper cleaner and then tin.
2. The tinning should be done NOT MORE THAN 24 HOURS before you begin to assemble the board. The tin forms the oxide with air slowly but surely and tin oxide does not solder without being well soaked in hot flux from the soldering iron. Do not tin one day and attempt to assemble the board next week. It will not solder well at all.

6. Labeling

1. On the COMPONENT (non-copper trace) side of the PC board, take some fairly coarse sandpaper (grit 80-120) and rough up the surface of the board. Wipe any sanding dust from the board with a damp cloth and dry the board thoroughly.
2. Put the board into the warm oven for two minutes.
3. Pick two X-holes (i.e. X105 & X101) that are furthest apart diagonally on the board. Take a scrap piece of wire and place the wire into these holes so that the wire comes out the top (components) side of the board.
4. On the photo paper with the reference designator labels, poke a tiny hole into the holes that you selected in step (3) above. Place the photo paper ink side down onto the board.
5. Iron the legend onto the top of the board in the same manner that you ironed the traces onto the bottom side of the board. Remove the paper backing in warm water in the same way you removed the paper from the trace side of the board.

VIII Project Fabrication

1. PCB Assembly

Final Assembly Directions

These directions should be followed once the board is etched and the chassis metalwork completed.

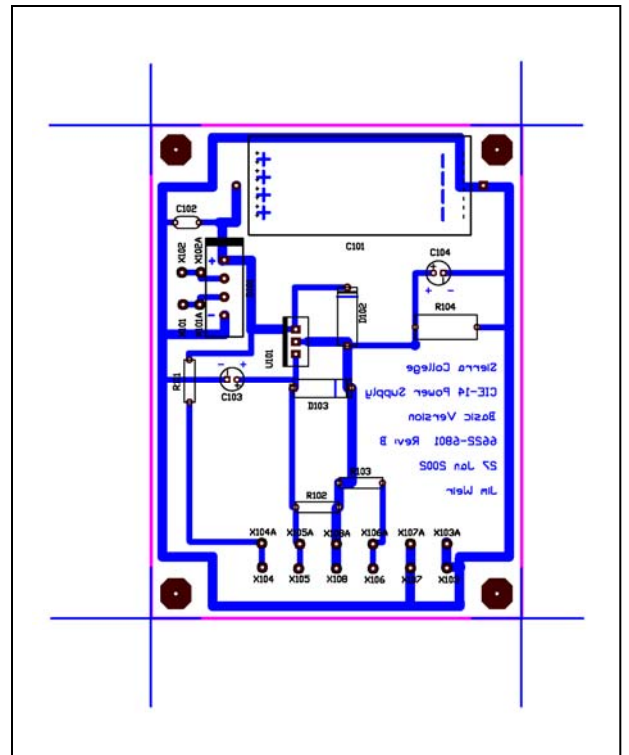
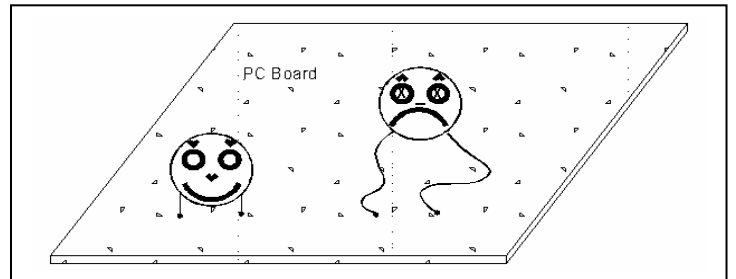
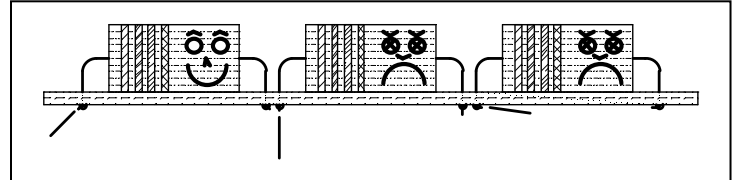
☑ each step off as you go.

You will need the following drawings to complete the fabrication of this project:

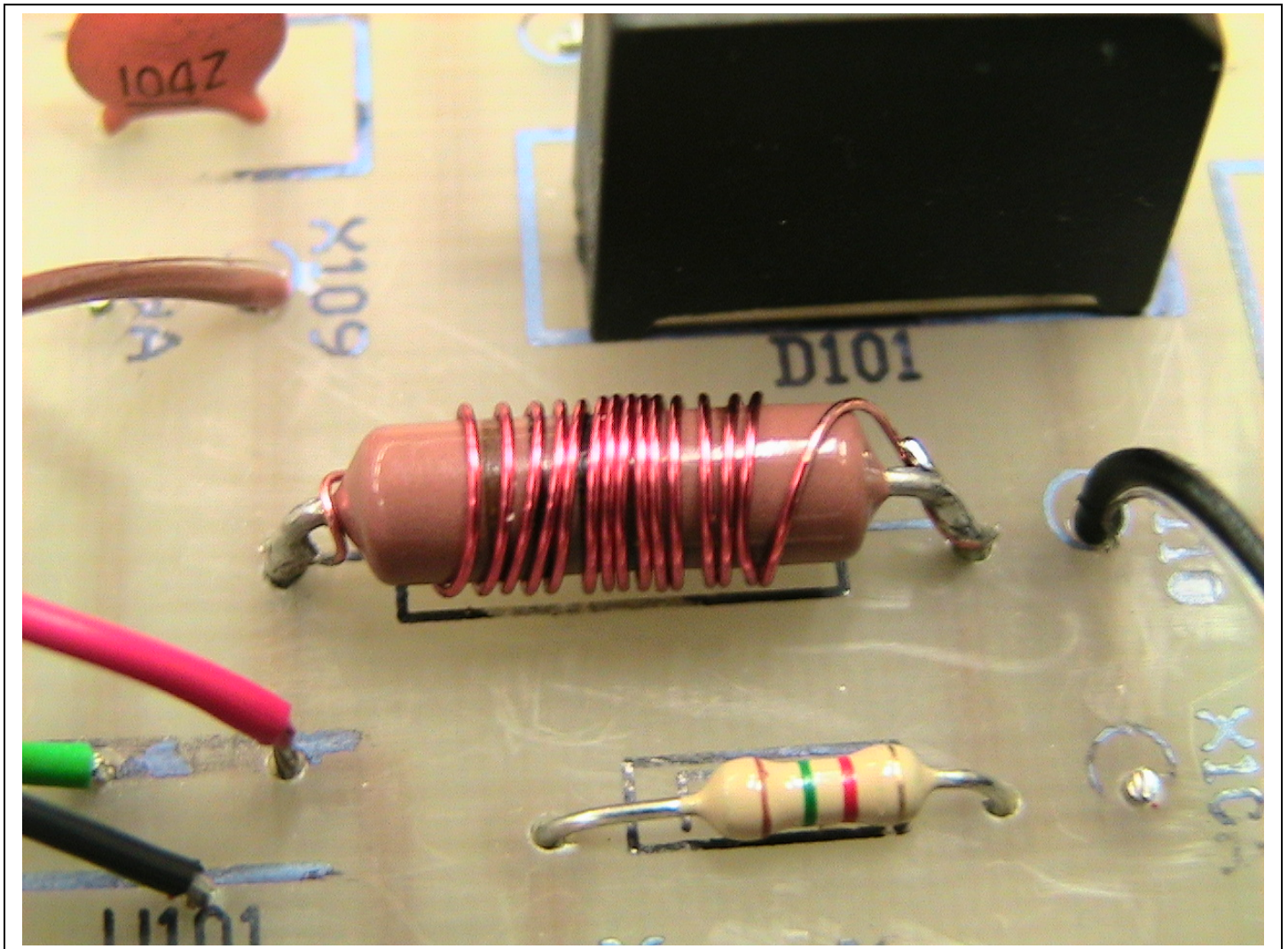
*6622-4200 sheets 1 through 5 from "Metal Fab".
(See the last 5 sheets of this section.)*

*CIE14-2801, 2901 or 2902, 5801 or 5802, and either
a 6801 or a 6802 drill drawing made from the PCB
artwork file.*

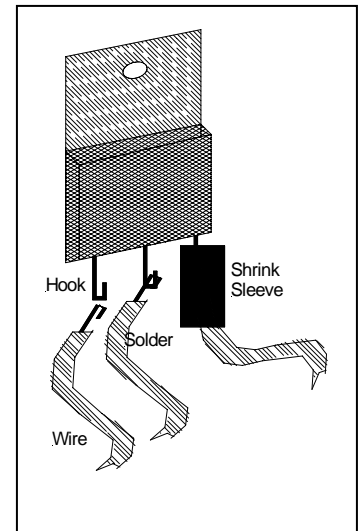
- ☐ Drill all the holes on the pc board 0.041" (#59) diameter. After you have drilled all the holes, drill out the two holes for the electrolytic capacitor 0.052" (#55). Drill out the four large corner mounting holes 0.120" (#31) diameter.
- ☐ Place the bridge rectifier D101 and large electrolytic capacitor C101 into their respective mounting holes on the board. Please note that the mitered end of D101 goes to the (top) dark bar on one end of the legend and that the electrolytic capacitor has a positive and negative end. Press both parts down until they are flat on the board and then splay the leads out at a 45° angle. (If you are unsure of how to identify these parts, consult the instructor or lab tech). Solder the parts to the board and clip off the excess wire length. Save these wire clippings for the next step.
- ☐ Take one of the wire clip-offs from the last step. Insert it into hole X1A so that it extends out the top of the board about an inch. Solder it and clip off the excess wire. (X1A on this layout is just to the left of the bridge rectifier.) Similarly, solder a wire clip-off into hole X2A.



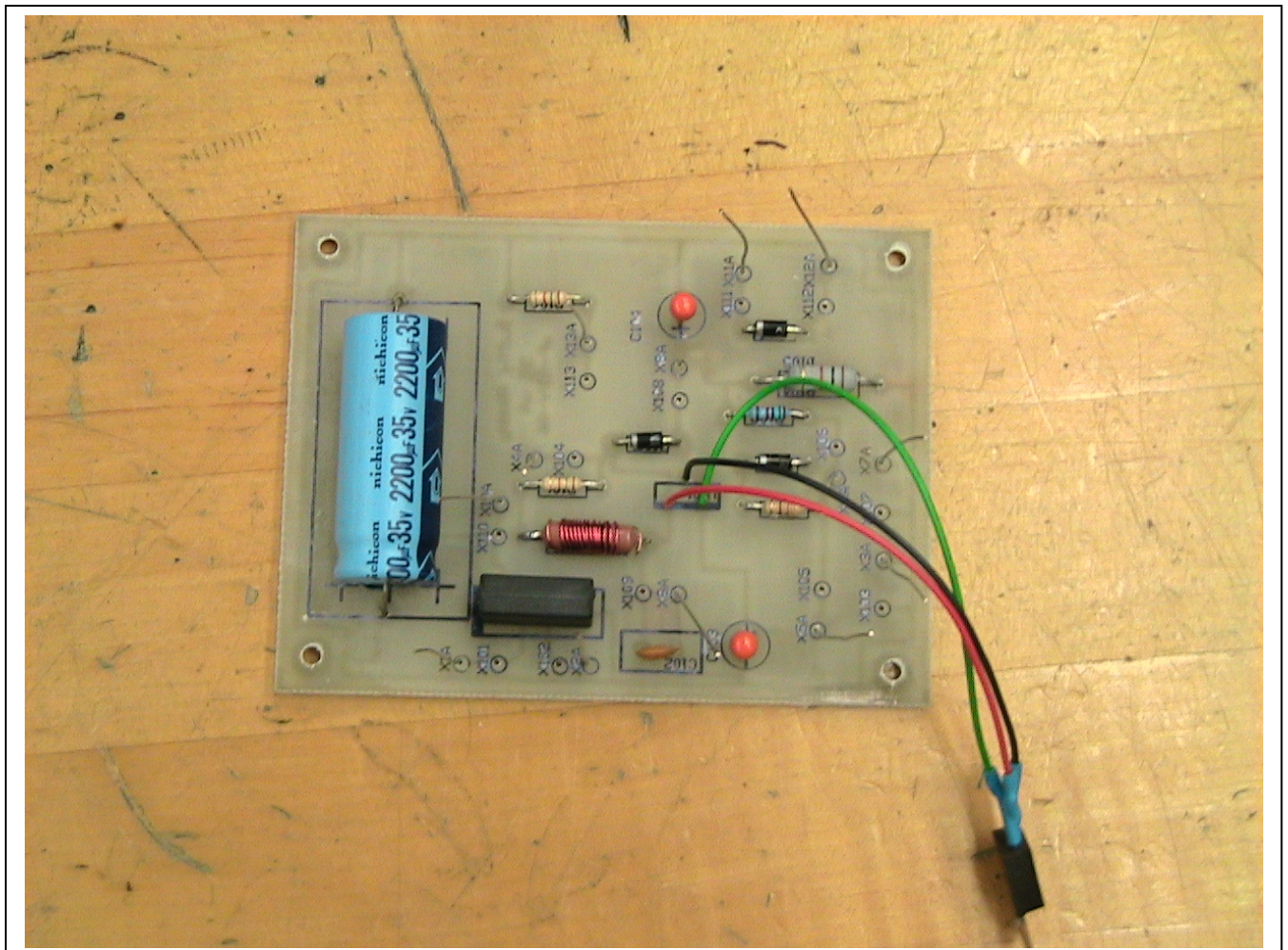
4. ☐ Take the partially assembled board to the test fixture and place it over the mounting screws so that it is solidly in place and cannot short out to the chassis bottom. Have the instructor or lab tech show you how to hook up the transformer red leads to X1A and X2A. Have the instructor plug the test fixture into the wall and turn the power switch on. Take a digital voltmeter and hook it to the (+) and (-) leads on the electrolytic capacitor. It should read plus 17 volts or thereabouts. If it does not, have the instructor show you how to troubleshoot your soldering.
5. ☐ Place the rest of the parts on the board **with the exception of the integrated circuit voltage regulator U101 and the 1 watt 1.0K resistor R105**. Solder each part as you put it onto the board. Save the clipped off leads as above. Remember, the 15k Ω resistor is on the back leads of the Radio Shack meter.
6. (Advanced version only). Take the remaining 1.0K 1 watt resistor and bend the leads down so that it will fit into the holes for R105. With an exacto knife, scrape the enamel coating for $\frac{1}{2}$ " on both ends of the 10" #28 wire so that they are shiny copper. Hook one end of the #28 wire to one end of the 1.0K resistor and crimp it tight with needlenose pliers. Solder this end of the wire as close to the body of the resistor as you can. Wind the remaining wire firmly onto the body of the resistor and solder the remaining end of the wire to the other lead of the resistor.



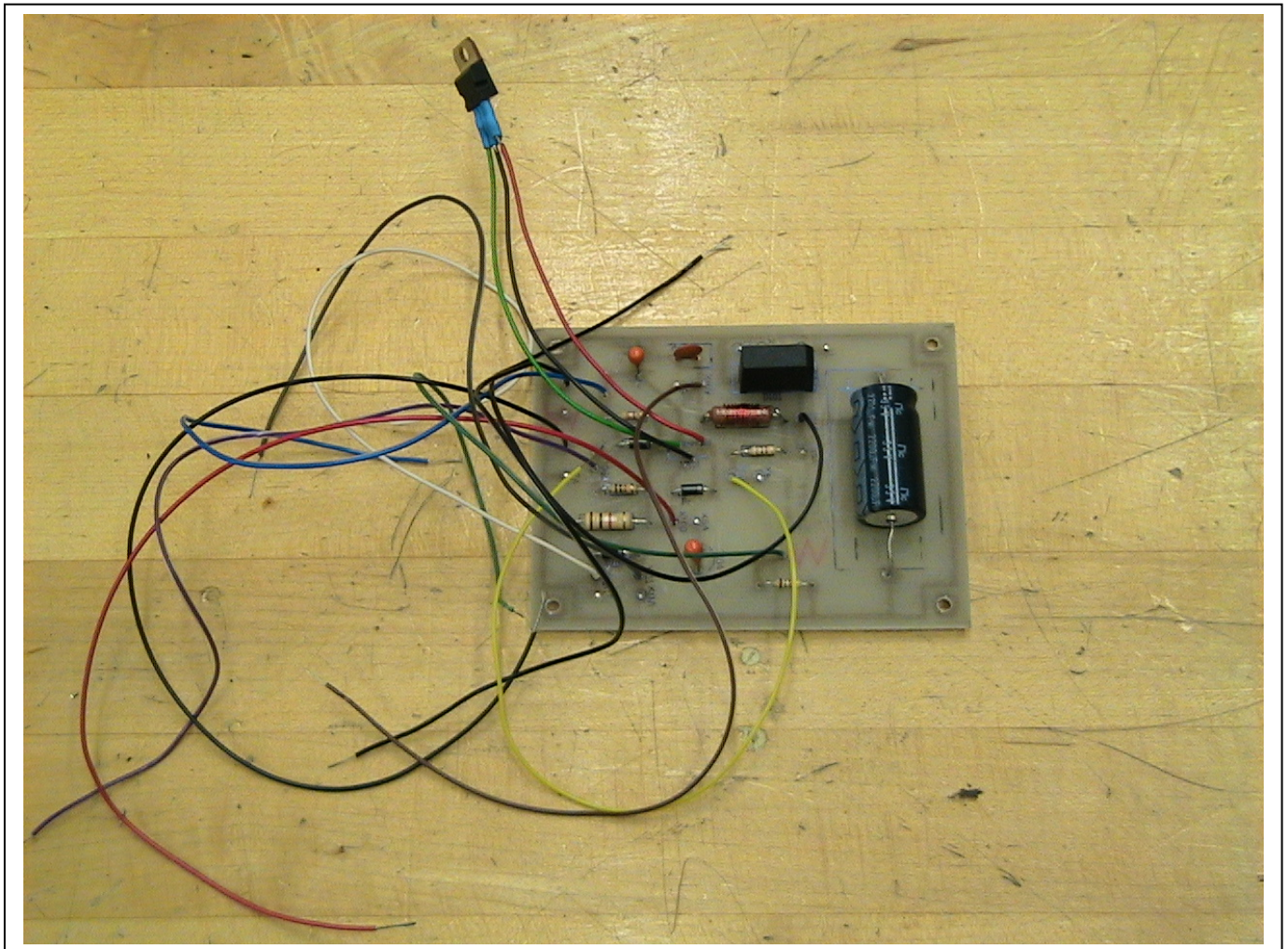
7. ☐ Clip the 3 leads coming out of the LM-317 voltage regulator IC so that about $\frac{1}{4}$ " of lead remains on the IC. Bend each lead into a U-hook with the bend going towards the FRONT of the IC. (The back of the IC is completely flat; the front has a bulge containing the IC.) Obtain a 12" long red wire stripped from the multiconductor ribbon cable, a 12" long orange wire, and a 12" long green wire. Strip $\frac{1}{2}$ " of the insulation on one end of each wire and tin (i.e. coat with a thin coating of solder). Bend each of the tinned wire ends into a hook. Looking at the front of the IC (letters and numbers printed side), hook the red wire into the **rightmost** pin-hook of the IC, squeeze both hooks together, then solder the IC lead to the wire. When you are done, the wire should be coming straight out of the bottom of the IC. Similarly, connect the orange wire to the center lead of the IC and the small green wire to the left lead of the IC. Slip a small piece of shrink sleeving over each one of these connections and shrink-insulate the wires and leads of the IC.



8. ☐ Place the PC board into your chassis at the approximate point where it will eventually reside. Hold the IC to the inside back of the chassis over the small 0.120" diameter hole drilled for it. Take all three leads together and run them onto the board where they are going to connect into U101. Give yourself at least an inch but no more than two inches of excess wire and clip the wires down where they are going to attach to the board. Strip the free end of each wire about a quarter of an inch. Place the wires one by one into their corresponding hole (note the heavy bar on the legend indicating the back or metal tab side of the IC), red to top or right, orange to center and small colored wire to bottom or left. Solder each wire as you install it and inspect **carefully** to make sure no little wire whiskers are shorting out between the wire holes.



9. ☐ As you did with X1A and X2A, solder a 1" test wire (clipped component wire) into all remaining test (A-suffix) holes. Return your board to the instructor and have the instructor show you how to test the completed board in the test fixture.
10. ☐ When the board tests "good" on all functions, wire up the board as follows (make all wires 12" long; use the "colors" that you have stripped from the multicolored ribbon cable):
- a. X103, small black wire
 - b. X104, small yellow wire
 - c. X105, small blue wire
 - d. X106, small violet wire
 - e. X107, #24 black wire
 - f. X108, #24 red wire
11. ☐ The following 7 wires are for the "advanced" version of this project
- a. X109, small brown wire
 - b. X110, small black wire
 - c. X111, small gray wire
 - d. X112, small white wire
 - e. X113, small green wire
 - f. D1 cathode, cut both leads to $\frac{1}{4}$ ", small red wire (hook to hook, solder, shrink sleeve)
 - g. D2 anode, cut both leads to $\frac{1}{4}$ ", small red wire (hook to hook, solder, shrink sleeve)

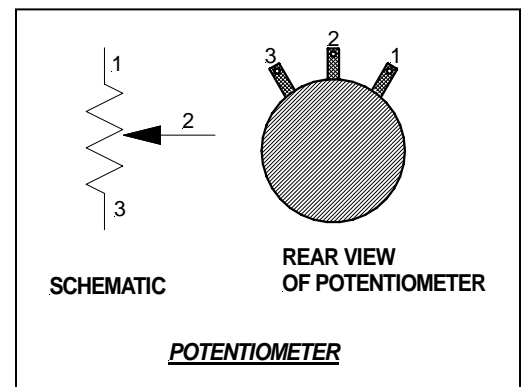


2. Chassis & Final Wiring

1. Place the PC board to one side of your workspace. Using #6 hardware (screw, lockwasher, nut) bolt the transformer to its spot on the chassis (see drawings 6622-4200 sheets 1 through 4). Install a ground lug under the lockwasher and nut on the transformer's rear mounting hardware (point the ground lug out the side of the chassis closest to the transformer. Install the transformer so that the RED leads exit the transformer nearest where the PC board is going to go.
2. ☐ Reading left to right looking at the faceplate, install the on-off switch S1, the potentiometer R1 (lugs facing up), the three banana jacks (J2 - green on top, J3 - black to the right, and J1 - red to the left) and the meter M1 on the front of the chassis. Take an ohmmeter and press on-off switch S1 until the ohmmeter reads "infinite" ohms.
3. ☐ Advanced version only, install a DPDT no-center-off switch in the center of the three ¼" holes under the meter (S2).
4. ☐ Install the fuse holder in the top hole on the rear of the chassis. Install a ¼ amp slo-blo fuse into the fuseholder.
5. ☐ Strip the line cord back 3" so that the black, white, and green wires are exposed. Cut off any string or packing material. (Note -- if the line cord is not color-coded black, white, and green, have the lab tech identify the line hot, neutral, and ground for you.) Using the special Heyco pliers, install the Heyco bushing and line cord onto the chassis remaining large hole. Be sure that the bushing grips the outer black plastic insulation of the line cord when it is seated in the mounting hole.
6. ☐ Strip the black line cord wire ½". Twist the small stranded wires of the black wire together tightly and tin. Slip a ¾" long piece of shrink sleeving over the black wire. Insert the black wire into the bottom lug on the fuse holder, bend the wire into a hook, and solder the wire to the fuse holder lug. After the joint cools, slip the shrink sleeving over the joint so that no metal of any sort shows on this joint and shrink the sleeving using a heat gun. (Note: from this point forward "connect" means to twist the strands of the wire tight, tin the wire, insert the wire into the appropriate lug, bend into a hook, crimp shut, and solder.)
7. ☐ Cut a piece of relatively heavy (#20-22) black stranded wire to fit from the fuse holder, down the chassis, up the chassis, and to the switch. Strip each end of the black wire ½", twist, and tin. Slip two ¾" pieces of shrink sleeving over the wire. Connect one end of the wire to the remaining lug on the fuse holder. Connect the other end of the wire to either of the lugs on the switch. In the same manner as you used in step 14, shrink sleeve both the fuse connection and the switch connection so that no metal shows from the connections.
8. ☐ Cut one of the black transformer leads so that it goes along the chassis bottom, up the chassis front, and to the remaining lug on the switch. Strip ½" and slip a ¾" shrink sleeving onto the lead. Connect the black transformer lead to the remaining switch lug. Shrink sleeve the connection.

9. ☐ Cut the remaining black transformer lead and the white line cord lead so that they will "hook" together (similar to how you connected wires to the integrated circuit) and not apply any tension to the wires when they are joined. Strip both leads $\frac{1}{2}$ ", twist, and tin. Slip **two** $\frac{3}{4}$ " pieces of shrink sleeving over one of the wires. Bend both wires into a hook, join together, and crimp tight. Solder. Slip one of the pieces of shrink sleeving over the connection and shrink the sleeving. Slip the other piece of shrink sleeving over the same connection and shrink again.
10. ☐ Cut a piece of small (#22-24) green wire 12" long. Cut both this green wire and the green wire coming from the line cord so that they will connect to the ground lug on the rear of the transformer. Connect one end of the green wire and the green wire coming from the line cord to the ground lug. Connect the free end of the green wire to green banana jack J2.
11. ☐ Place the PC board in its approximate final location on the chassis. Cut the two transformer red leads so that they will easily go into holes X101 and X102. Connect the red leads (it does not matter which one is which) one lead to each of these two holes.
12. ☐ Using 4-40 x $\frac{3}{4}$ " screws, spacers, lockwashers, and nuts, mount the PC board to its spot on the chassis. Note that two rubber "feet" will be used on the chassis edge side of the hardware. If you find the #4 hardware pulling through the foot, use a flatwasher between the screw head and the foot.
13. ☐ (Basic version only)

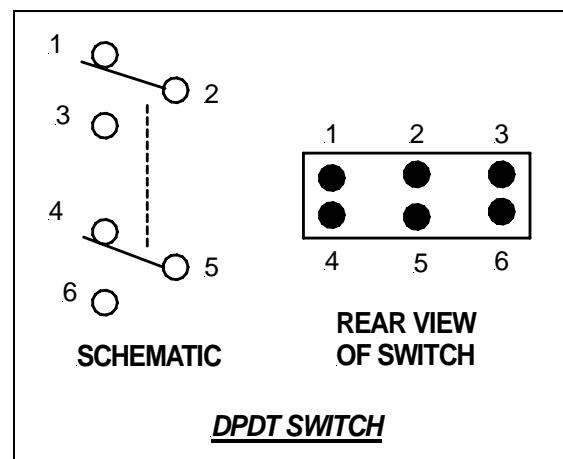
- h. Connect the black wire coming from X103 to pins 1 & 2 of R1 (you may use a clipped component lead to make the connection from pin 1 to pin 2 of R1 if you wish). Before soldering R1, run a small black wire to the (-) minus lug of the meter and solder at the meter lug. Do not solder R1 until the next step.



- i. Cut the leads of light emitting diode D1 to $\frac{1}{2}$ ". Connect a small black wire from R1 pins 1 or 2 to the cathode of D1. (Note, the cathode is the side with the "flat" on the case.) Cut the wire to length and use shrink sleeving on the connection. Use the "hook and hook" method of connection. Similarly, connect the small yellow wire from X104 to the anode of D1.
- j. Connect the violet wire from X106 to the (+) plus lug on the meter.

14. ☐ (Advanced version only):

- k. S2 pin 1 to violet wire coming from X106
- l. S2 pin 2 to white wire coming from X112, and a second white wire to the meter (+) plus lug
- m. S2 pin 3 to orange wire coming from X110
- n. S2 pin 4 to black wire coming from X103 and a second black wire to R1 pins 1 & 2 (you may use a clipped component lead to make the



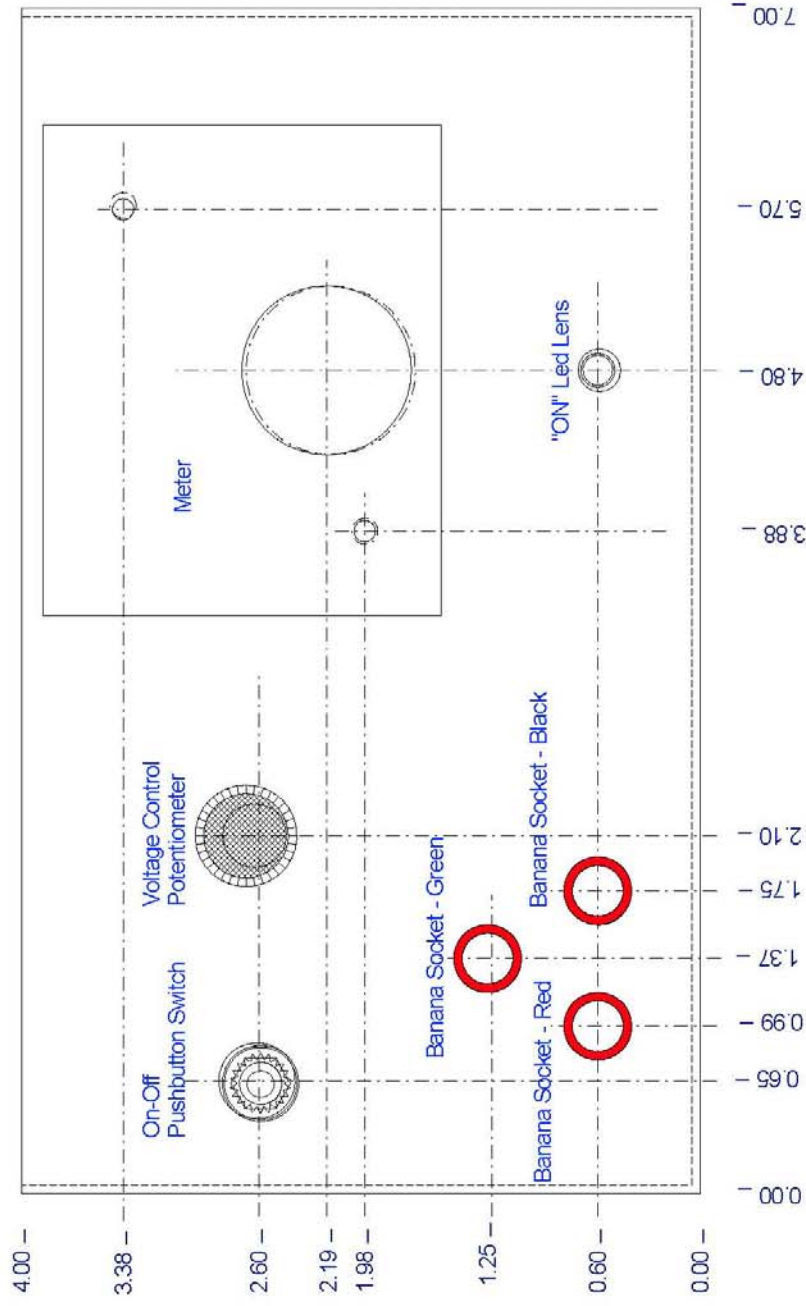
- connection between pins 1 & 2 if you wish)
- o. S2 pin 5 to the gray wire coming from X111. Before soldering pin 5, run another gray wire to the meter (-) lug along with both red wires coming from D1 cathode and D2 anode. If you wish and to make soldering easier, you may twist these last 3 wires together along with a single gray wire, solder them all together, shrink sleeve them, and then run the free end of the single gray wire to the (-) lug of the meter.
 - p. S2 pin 6 to the brown wire coming from X109.
 - q. Green wire from X113 to D2 cathode.
15. ☐ Connect the heavy black wire from X107 to black banana jack J3. Similarly, connect the heavy red wire from X108 to red banana jack J1.
16. ☐ Drawing 6622-4200 sheet 3 details how the integrated circuit bolts to the rear of the chassis. There needs to be thermal compound ("goose grease") between the case of the integrated circuit and the chassis. Use a TINY bit of grease, and don't get it on your hands or your clothes. It isn't particularly poisonous, but it is very difficult to remove. If your integrated circuit has a metal tab for fastening, then you will have to insulate it from the chassis with a mica washer and a shoulder washer. These insulation kits are available from the lab techs. After you have installed the IC, take an ohmmeter and test to see whether there is continuity from the metal tab to the chassis. There should be an infinite reading on the ohmmeter if you have installed the hardware properly.
17. ☐ Install the remaining two rubber feet on the side of the chassis closest to the transformer.
18. ☐ Assemble the red test lead with a red banana plug on one end of the red wire and an alligator clip and red insulator on the other end of the red wire. Similarly, construct a black test lead.

This completes the assembly of the power supply.

Testing

- 1. ☐ Plug the AC line cord into an AC outlet.
- 2. ☐ Rotate the voltage control R1 maximum counterclockwise.
- 3. ☐ Press the on-off switch S1 once. The red LED D1 should light and the meter should show a little over a volt and a half. Rotate R1 and the voltage should smoothly go from the bottom of the meter scale to nearly 15 volts.
- 4. ☐ Rotate R1 so that the meter reads 10 volts. Connect a 10 ohm 10 watt resistor from the black to the red banana jacks J3 & J1. The meter should drop less than half a volt. (Advanced version, the current function on the meter should read "10" on the meter, indicating 1.0 amperes of current.)

This completes the testing of the power supply. Semester nearly over. Congratulations.



Assembly Drawing Basic Version

Power Supply Main Chassis

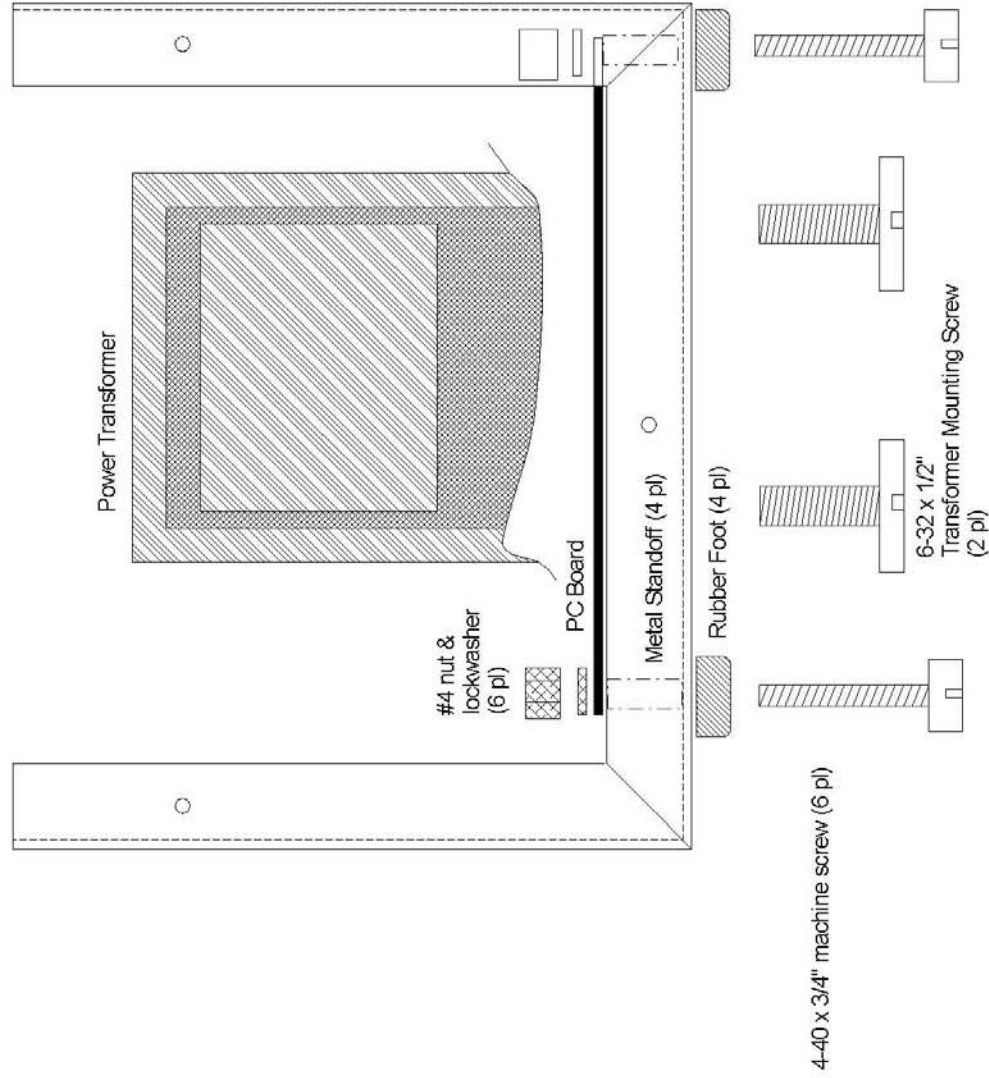
Scale: 1:1
Date: 02 Oct 04

Drawn by: OWJ
Rev: C

Approved by:
Jim Weir

RST Engineering
13249 Grass Valley Ave Grass Valley, CA 95945
530.272.2203
tech@rst-engr.com

Drawing #
6622-4200
Sheet 1 of 5



Assembly Drawing Right Side View

Power Supply Main Chassis

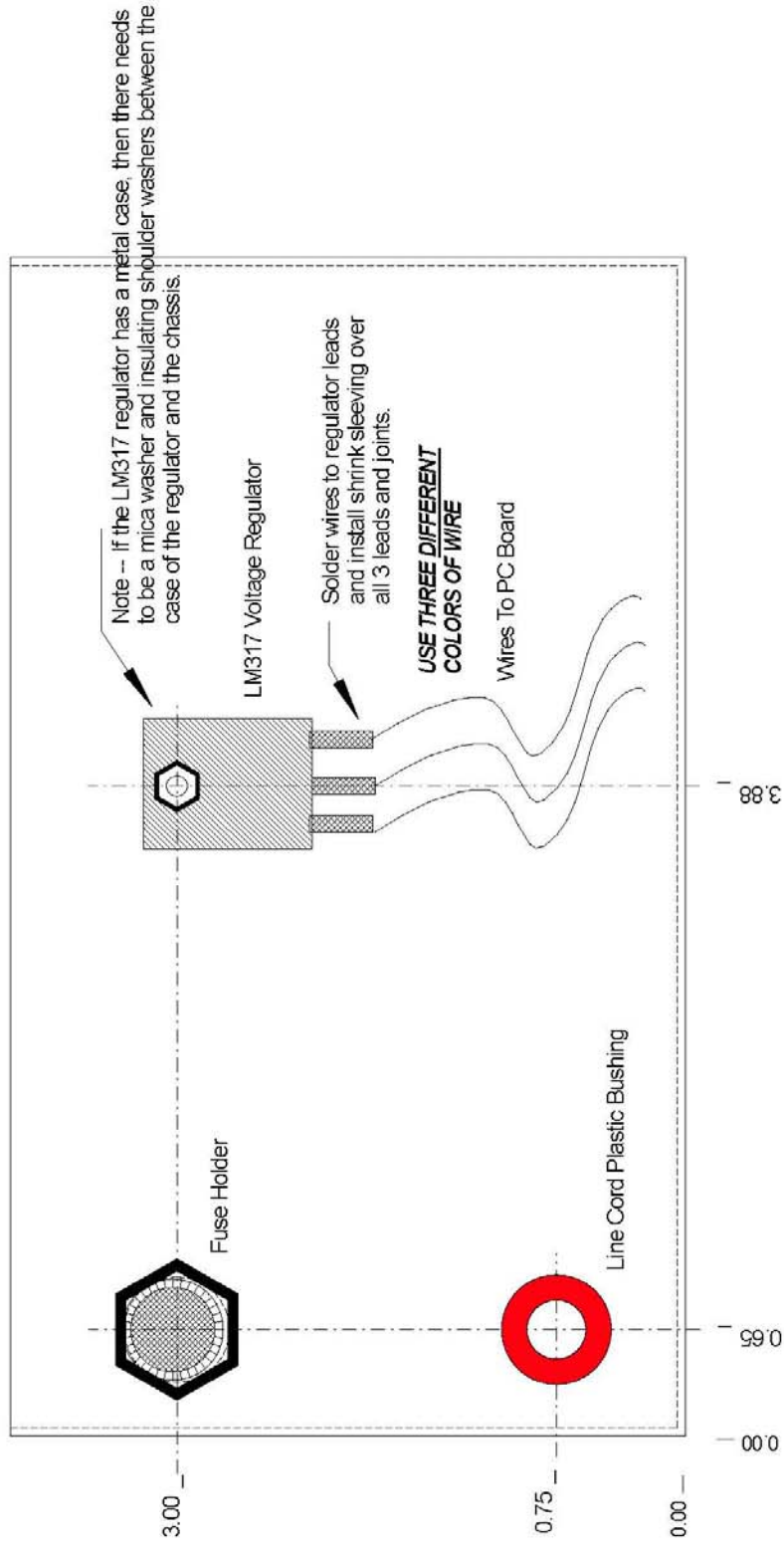
Scale: 1:1
Date: 20 Nov 01

Drawn by: O/WJ
Rev: B

Approved by: Jim Weir

Drawing # 6622-4200
Sheet 2 of 5

RST Engineering
13249 Grass Valley Ave Grass Valley, CA 95945
916.272.2203 tech@rst-engr.com



Assembly Drawing

Back Seen Through Front

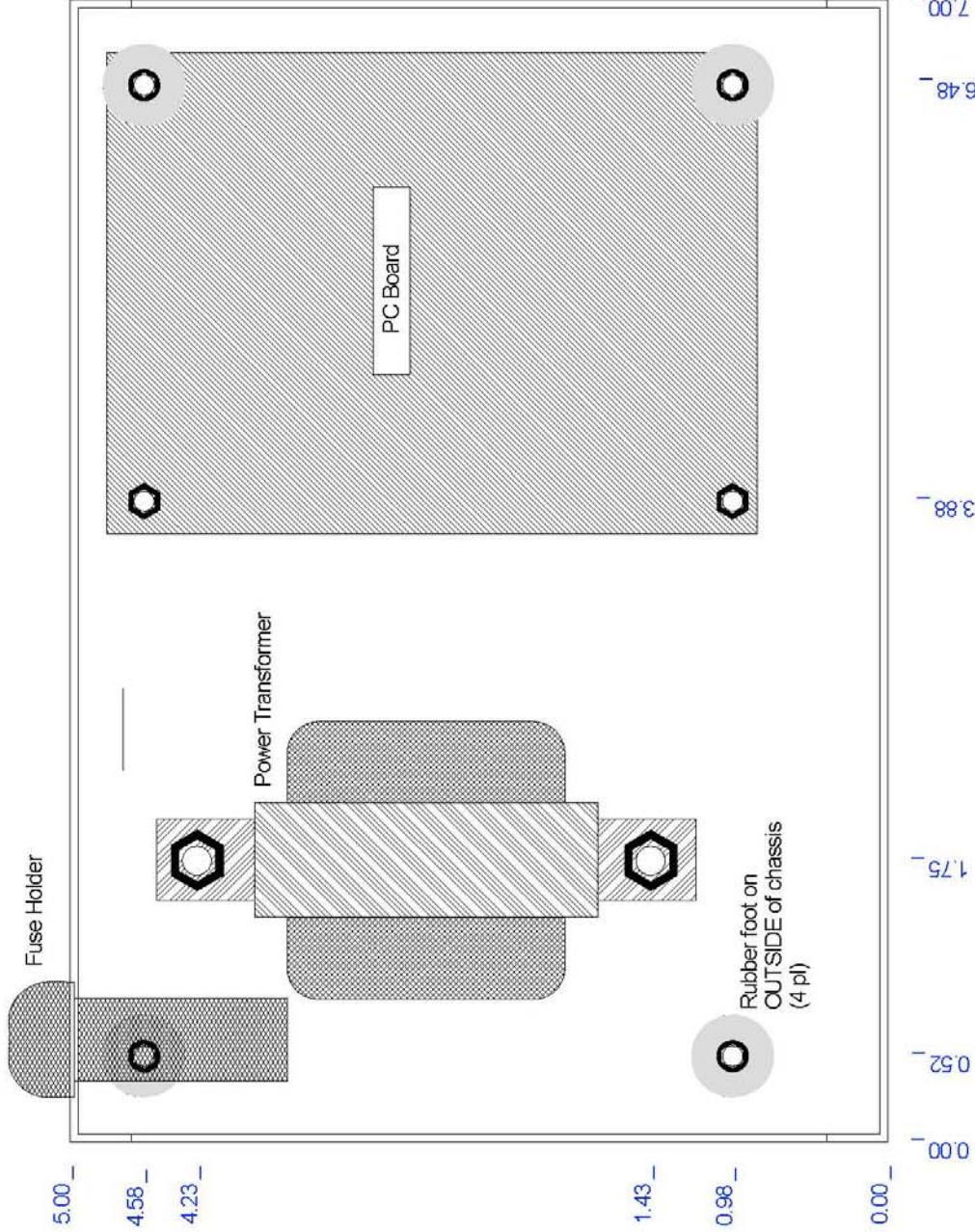
Power Supply Main Chassis

Scale: 1:1
Date: 30 Aug 04

Drawn by: OWJ
Rev: D

Approved by: **RST Engineering**
13249 Grass Valley Ave Grass Valley, CA 95945
530.272.2203 tech@rst-engr.com

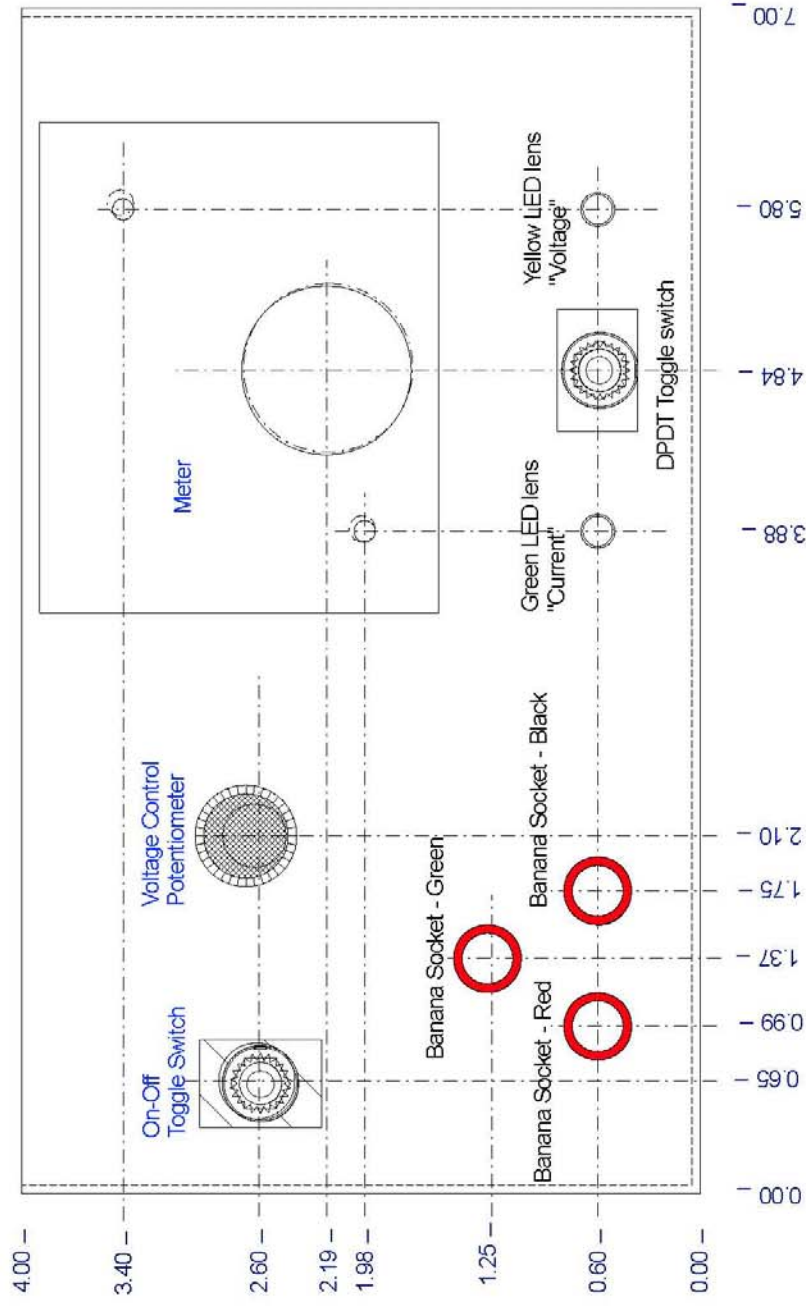
Drawing #
6622-4200
Sheet 3 of 5



Note: Use shrink sleeving on both white and black power cord wires to the MAXIMUM extent possible. Do not expose ANY metal objects to which white or black wires are connected.

Assembly Drawing Bottom View

Power Supply Main Chassis			RST Engineering 13249 Grass Valley Ave Grass Valley CA 95945 530.272.2203 tech@rst-engr.com		Drawing # 6622-4200 Sheet 4 of 5
Scale: 1:1	Drawn by: OWJ		Approved by:		Jim Weir
	Date: 24 Oct 06	Rev: C			



Assembly Drawing Advanced Version

Power Supply Main Chassis

Scale: 1:1
Date: 22 Nov 01

Drawn by: OWJ
Rev: B

Approved by:
Jim Weir
13249 Grass Valley Ave Grass Valley, CA 95945
530.272.2203 tech@rst-engr.com

Drawing #
6622-4200
Sheet 5 of 5