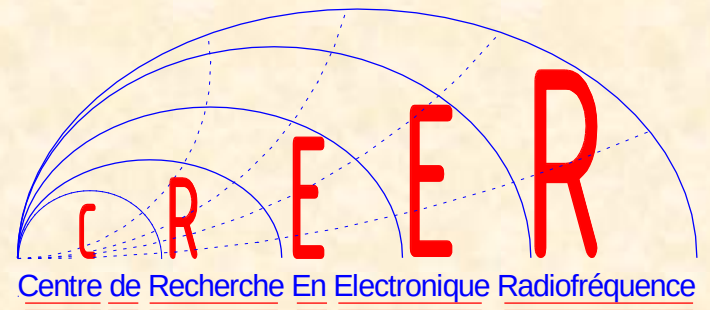
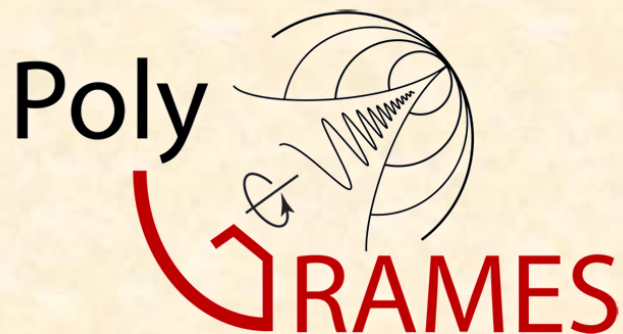


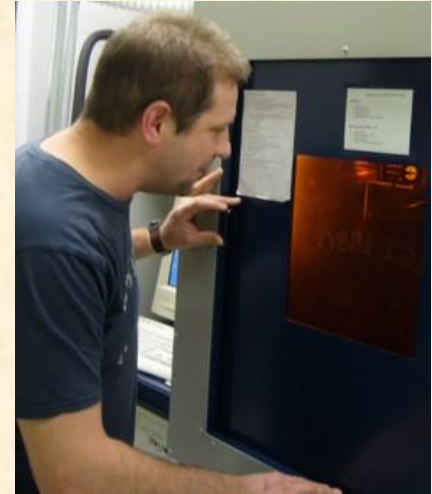
Fabrication Process



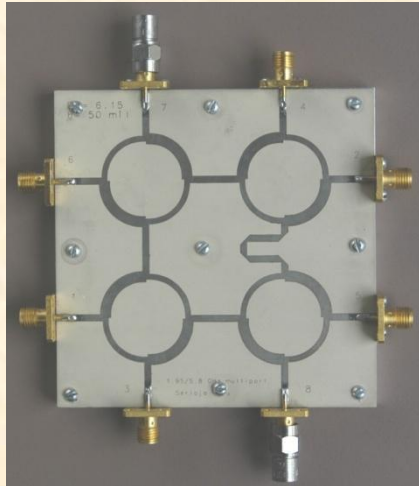
Fabrication process

- Single layer (ref. page [4](#))
 - No vias (ref. page [4](#))
 - With riveted vias (ref. pages [4,5,6](#))
 - With plated vias (ref. pages [4,5,7,8,9,10,11](#))
 - Component assembly (ref. pages [12,13,14,15](#))
 - SIW (ref. pages [4,5,7,8,9,10,11,15,16,17](#))
- Multilayer (ref. pages [18-24](#),)
- MHMIC (ref. pages [25-35](#))
 - Without vias (ref. pages [27-30](#))
 - With vias (ref. pages [28-30](#))
 - Resistor (ref. pages [31-35](#))
 - Circuit installing and connection (ref. pages [36-40](#))
- Waveguide (ref. pages [41,42](#))
- Circuit test (ref. pages [43-45](#))
- Appendix (ref. pages [46-54](#))

Team



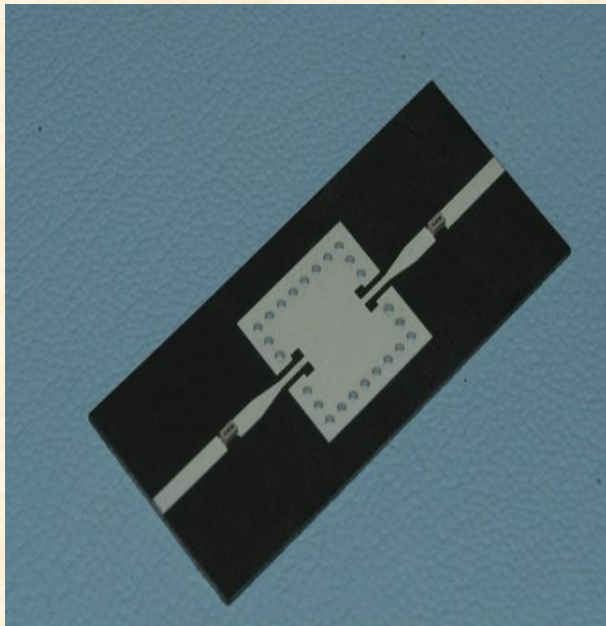
Single layer



Specifications for circuits

- Max size 8"X 8"
- Min. line width 6 mils
- Min. gap between lines 6 mils
- Connector spacing 0.5"
- Drawing file: Gerber RS-274X , precision 2.4
- Units are **inches**
- Each layer has to contain text with a minimum height 80 mils and track width 8 mils
- Cutting marks: 200x200x20 mils
- Origin at the lowest left corner of the layout

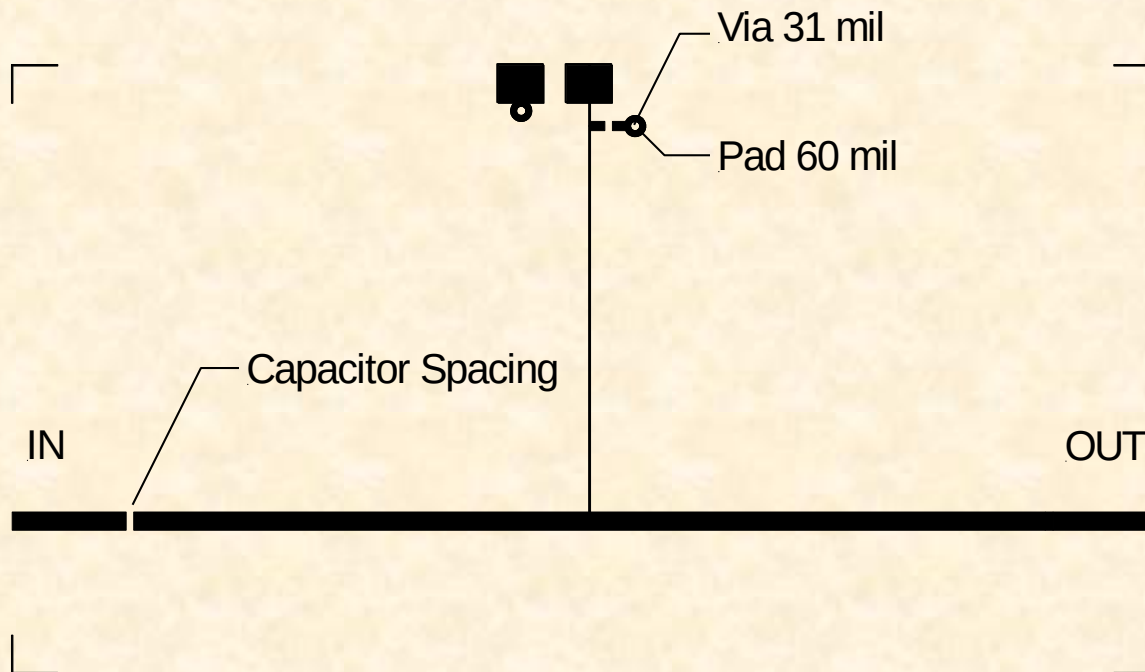
Single layer with vias



- Same as single layer
- Riveted vias
 - Used only if circuit contains less than 15 holes
 - Hole 31 mils diameter
 - Pad 60 mils diameter
 - Center of pad uncovered
- Plated vias
 - Used for large quantity of holes on circuits or if size of hole is critical
 - Min. aspect ratio of 2:1
 - Center of pad covered in the layout
 - .dxf file for holes (precision 2.4),
 - Max. circuit size 6"x6"

Circuits with riveted vias

- Layout files: Gerber RS-274X, precision 2.4
- No drill file required
- Only one hole size: 31 mil



File name, Student/Professor
Substrate/thickness

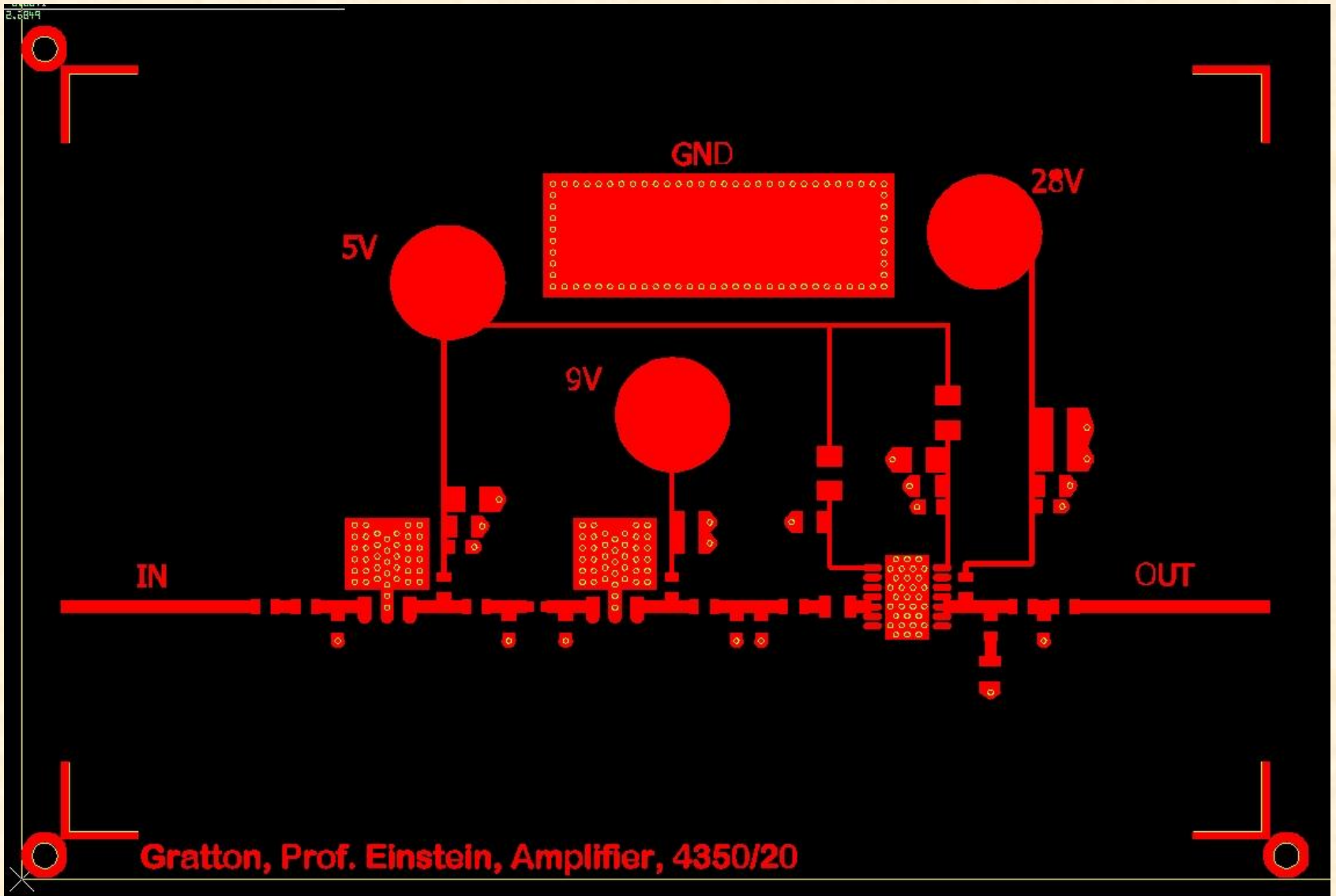
Plated Vias

- Layout file: Gerber RS-274x, 2.4
- Hole file dxf, 2.4, single layer
- Holes are filled on the layout file
- Three alignment holes of 66/120 mils dia with donuts outside of layout
- Aspect ratio for plated hole max 2:1
- Clearance of 10 mils
- Max PCB size 6" X 6"
- Laser cutting area is limited to 3.75 X 3.75

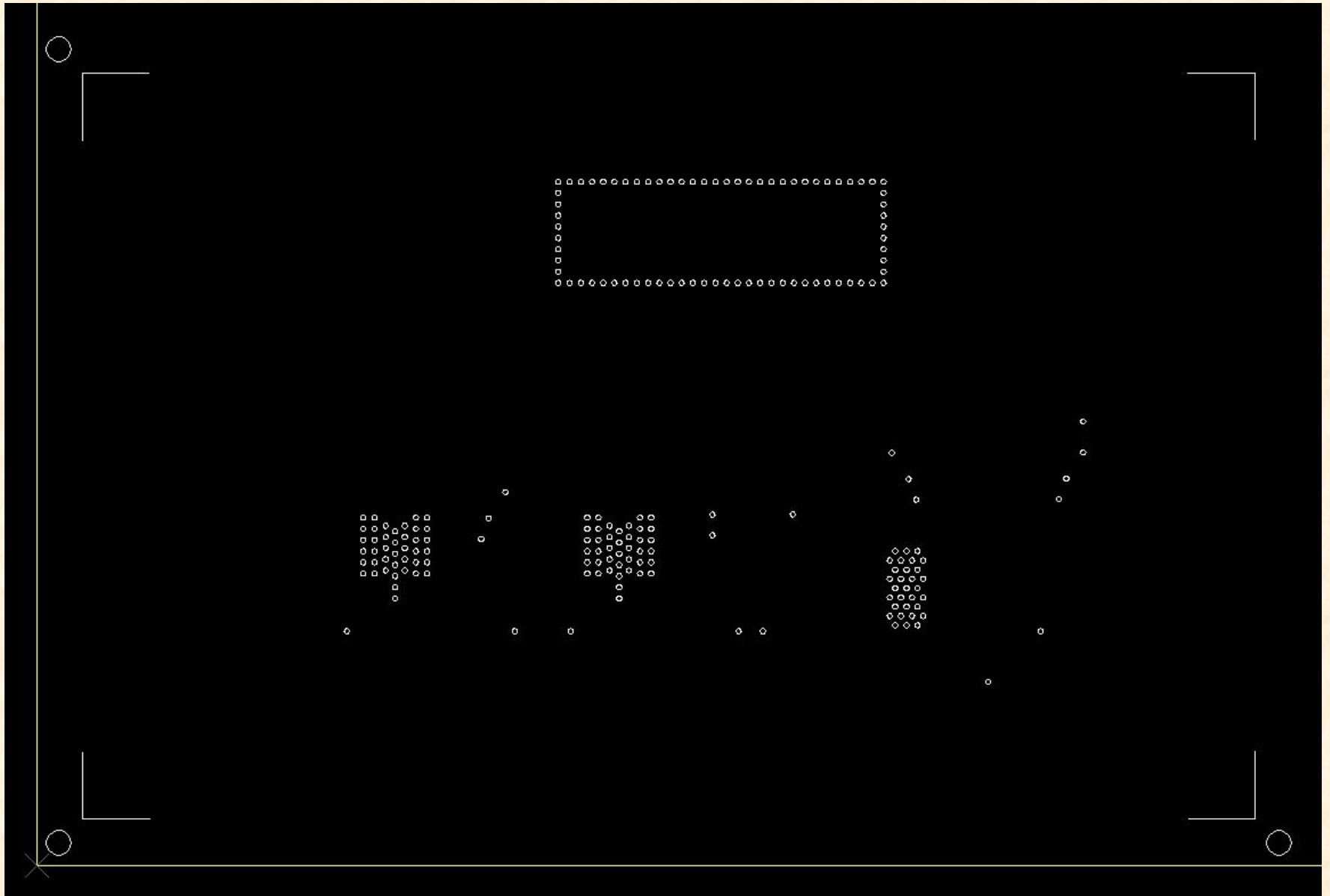
Laser drilling

- For laser drilling use poly-line
- Holes need to be circles, not line segments
- Minimum space 10 mil between 2 holes
- For soft substrates maximum thickness: 60 mils
- For ceramic: 20 mils
- Minimum hole diameter 5 mils
- Maximum circuit size 3.75" X 3.75"
- Oversize of cutting 2 mils

Plated Vias – Gerber and dxf files

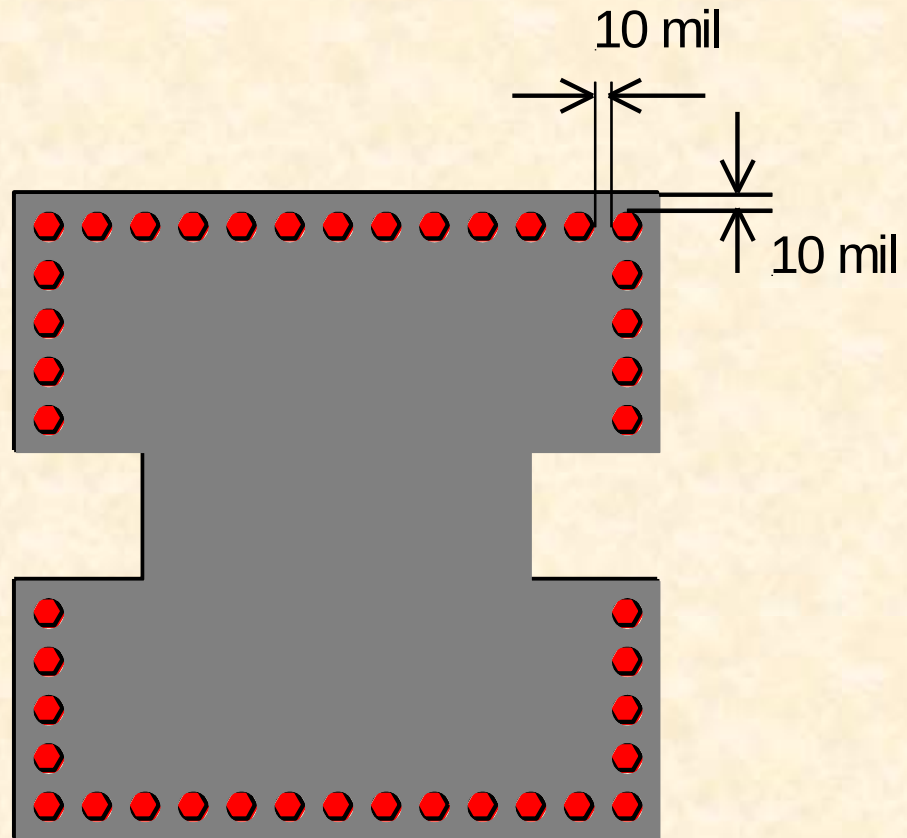


Plated Vias –dxf files



Clearance

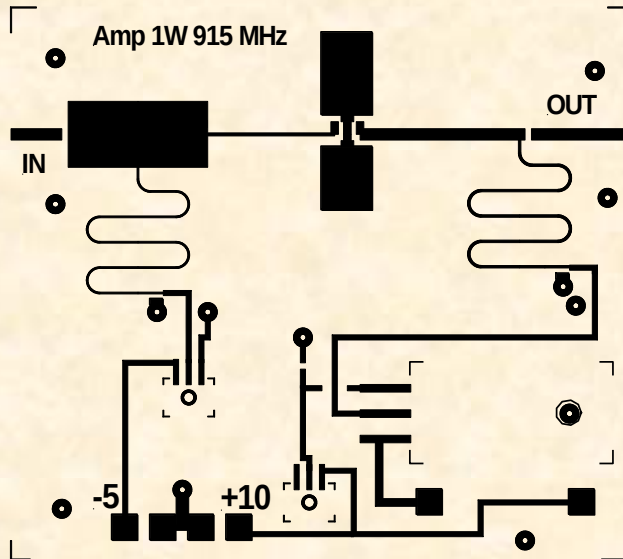
- Spacing of 10 mils min. between hole edges
- Spacing of 10 mils min. between hole edge and copper



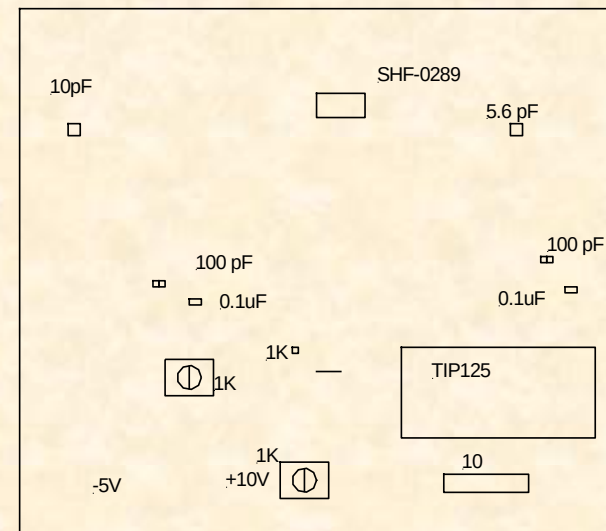
Component assembly

You have to provide the following drawings:

Layout



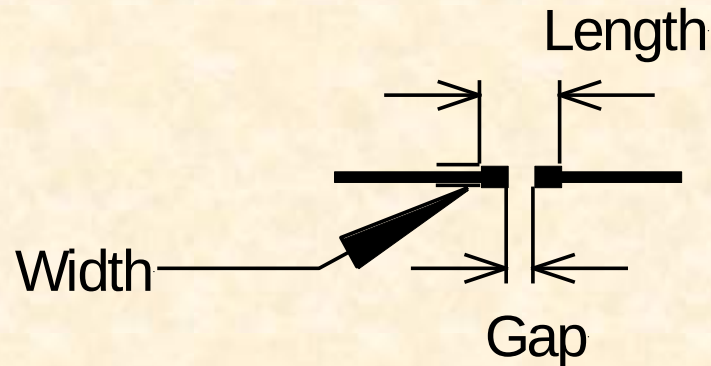
Component location



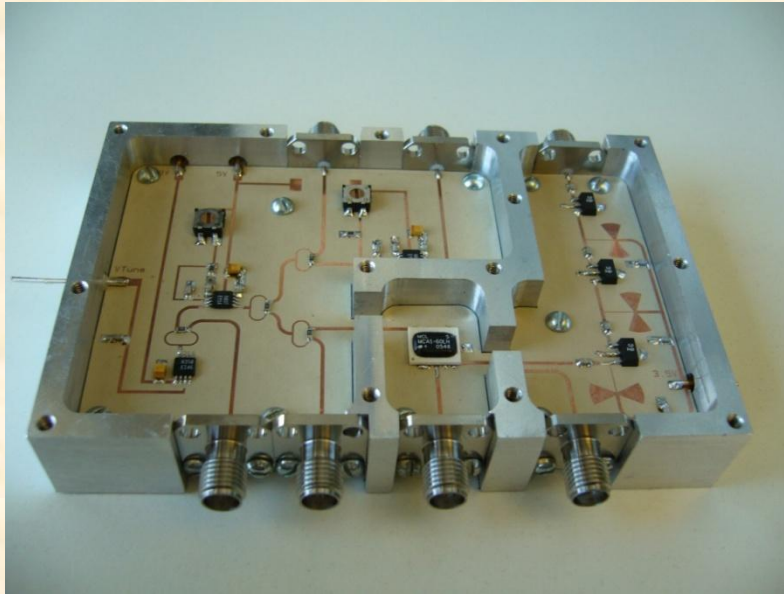
A component list is required (type of component and quantity)

Component spacing

- 0402 Gap 20 mil Pad width 25 mils length 55 mils
- 0603 Gap 30 mil Pad width 35 mils length 75 mils
- 0805 Gap 40 mil Pad width 55 mils length 95 mils
- 1206 Gap 40 mil Pad width 65 mils length 140 mils
- For integrated circuits refer to the specification of manufacturer
- Consider that you will have to solder the components, leave enough space

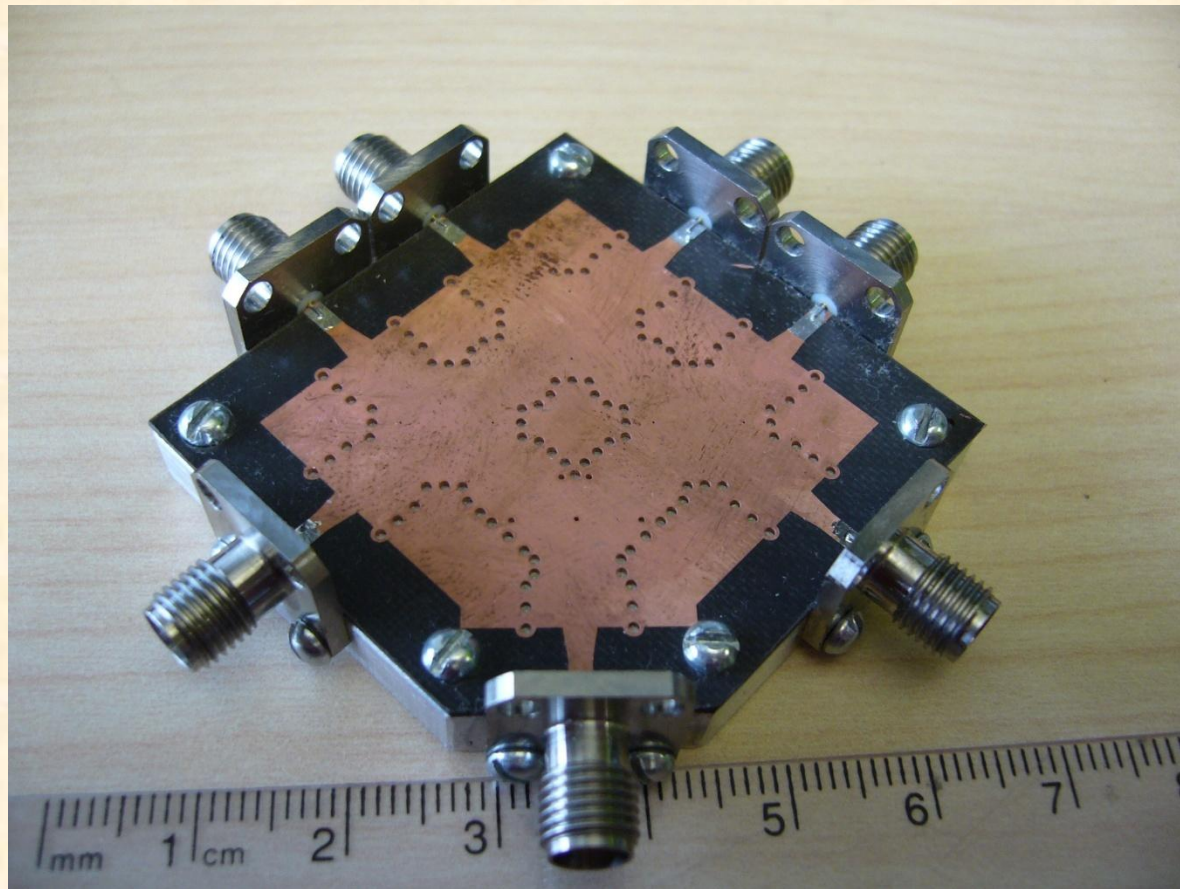


Assembly box

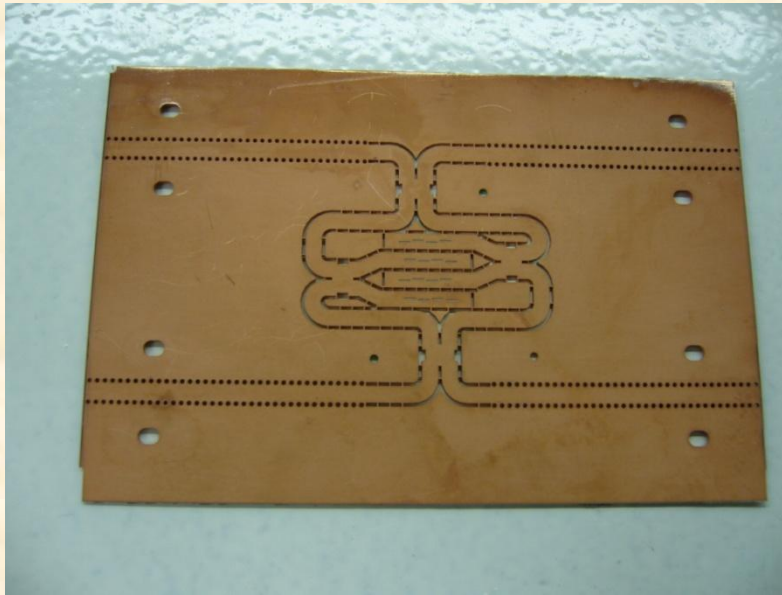


- Drawing of a 2D box
- dxf file
- Position of hole on drawing should be accurate and at scale
- Unit is inch
- Origin in the lowest-left corner
- Provide all the information needed (thickness, material, etc)

Assembly Base

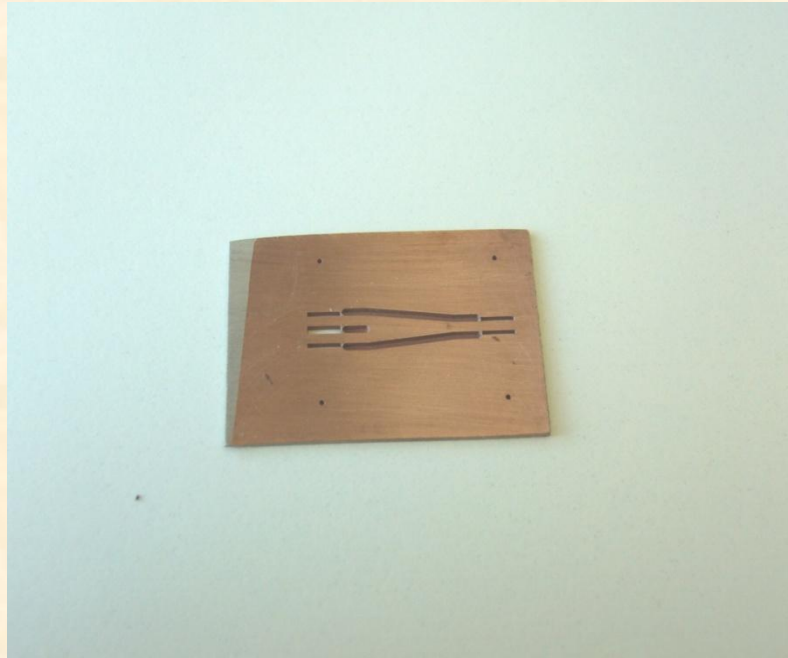


SIW



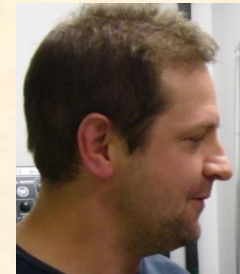
- Holes or slots are cut by laser
- Consider an oversize of 2 mils for laser cutting
- Laser cuts only through-hole
- Aspect ratio of minimum 2:1 should be maintained for plating
- dxf file precision 2.4, single layer

Partly machined substrate



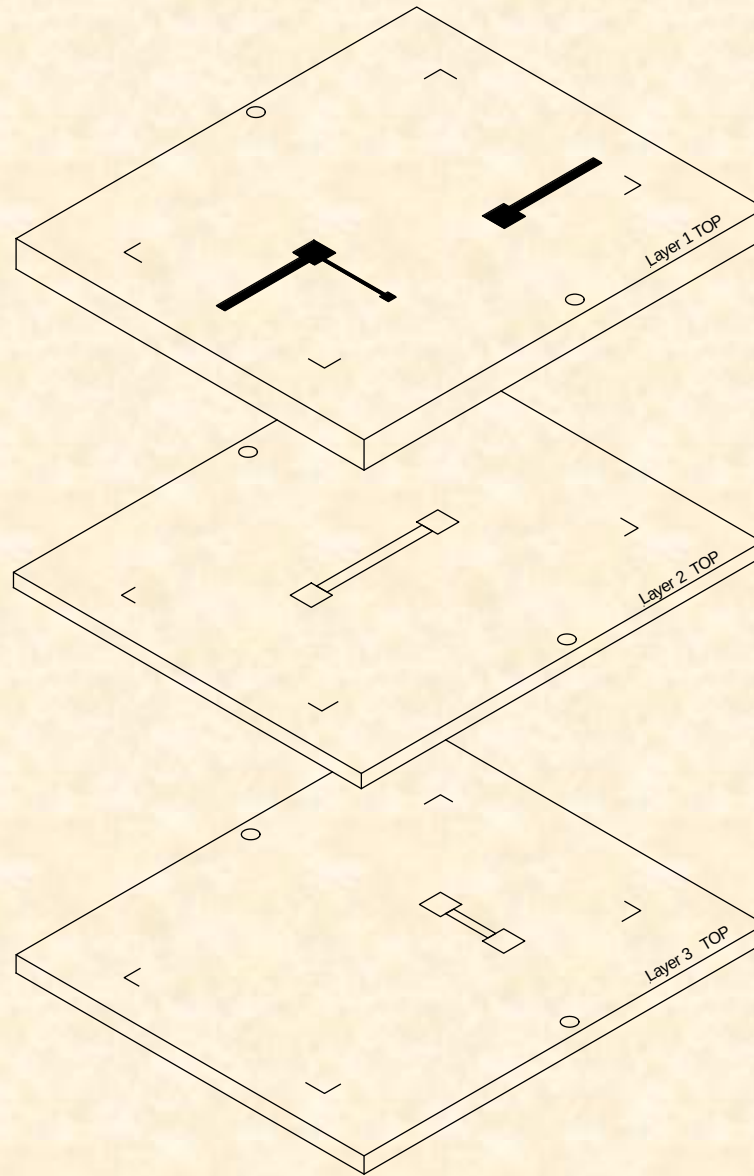
- The channel is cut by milling, only standard sizes are available.

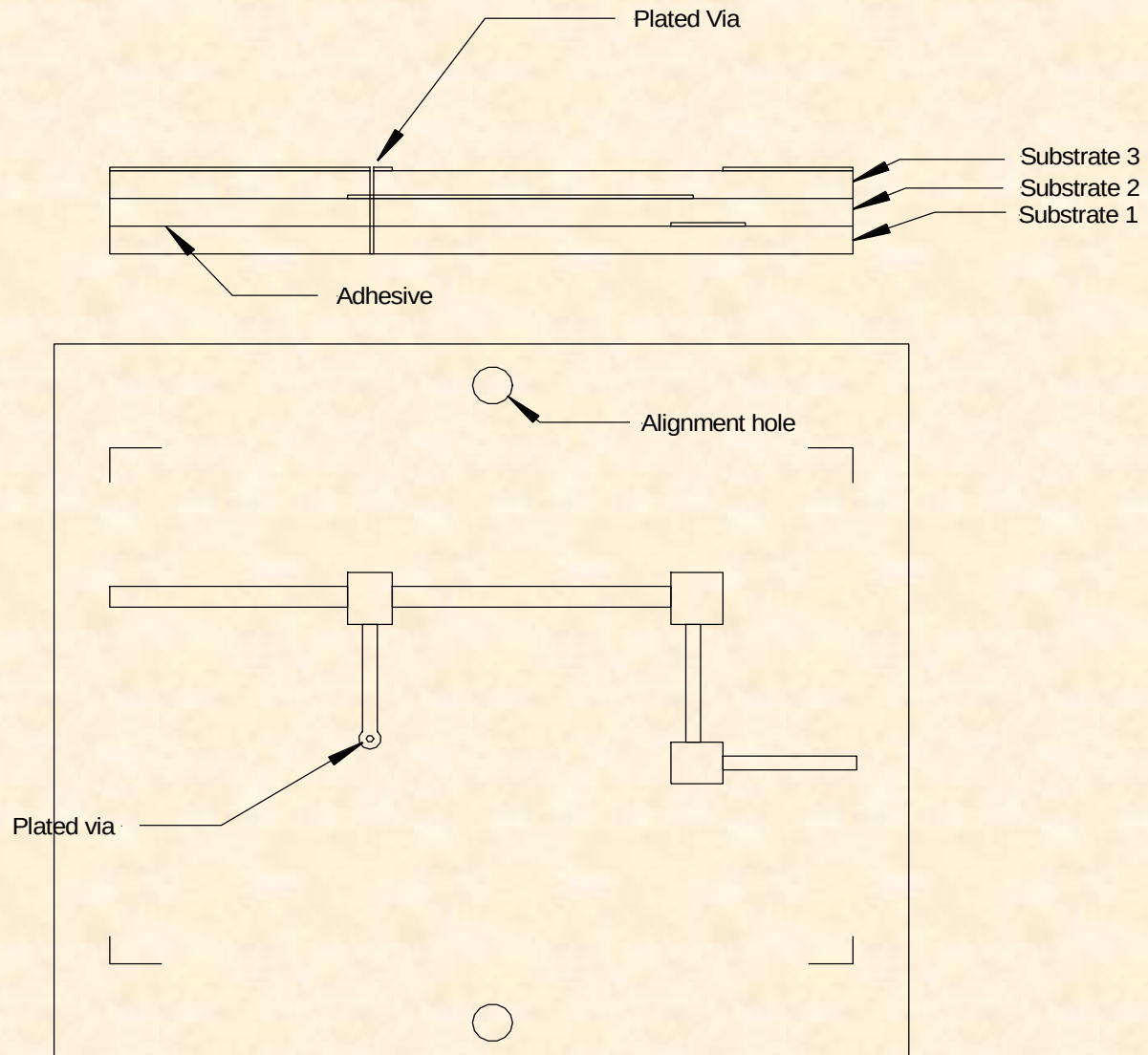
Contact: Steve



- Patterning required: Gerber file.
- Minimum remaining thickness of substrate 10 mils

Multilayer





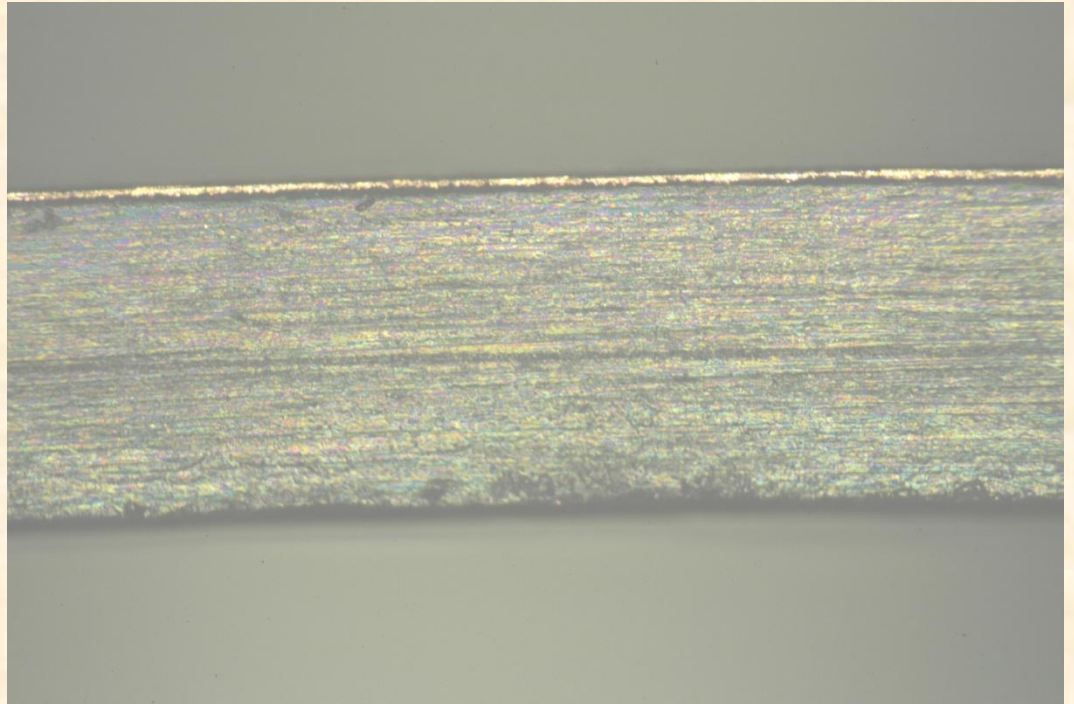
Side and top view of multilayer circuit

Required files

- Need a 3 D drawing to represent the entire circuit
- Plated hole is possible and inner connection is possible too.
- If connection is required between two substrates, a plated via is required
- Max size is 2" X 5"
- First substrate is on the bottom
- Text needs to be printed on each layer
- Indicate on the drawing the top or bottom layer
- Ex: S1B (substrate 1 bottom)
- Gerber file scale 1 for each layer
- Drill file is in .dxf
 - Limited to 3.75" X 3.75"
- The alignment between substrates is made using 0.125" diameter pins at 2.5" center to center distance

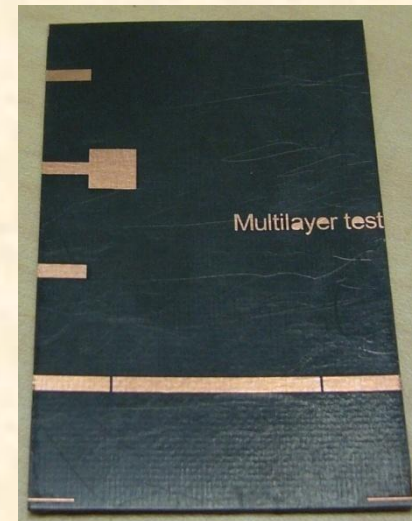
Adhesive specification

- Adhesive is epoxy
- $ER = 3.5$
- $T_d = 0.03$
- Thickness $5\ \mu\text{m}$
- Solvent resistant
- Hole can be plated through adhesive

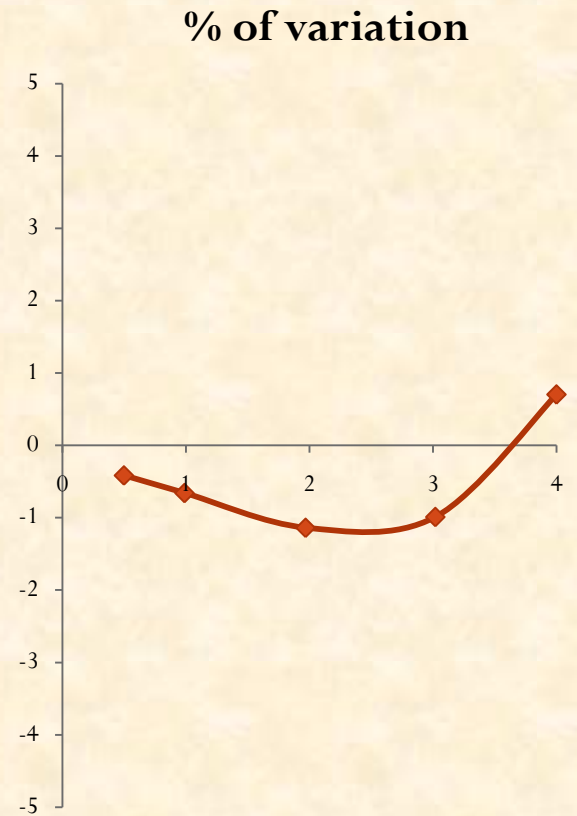
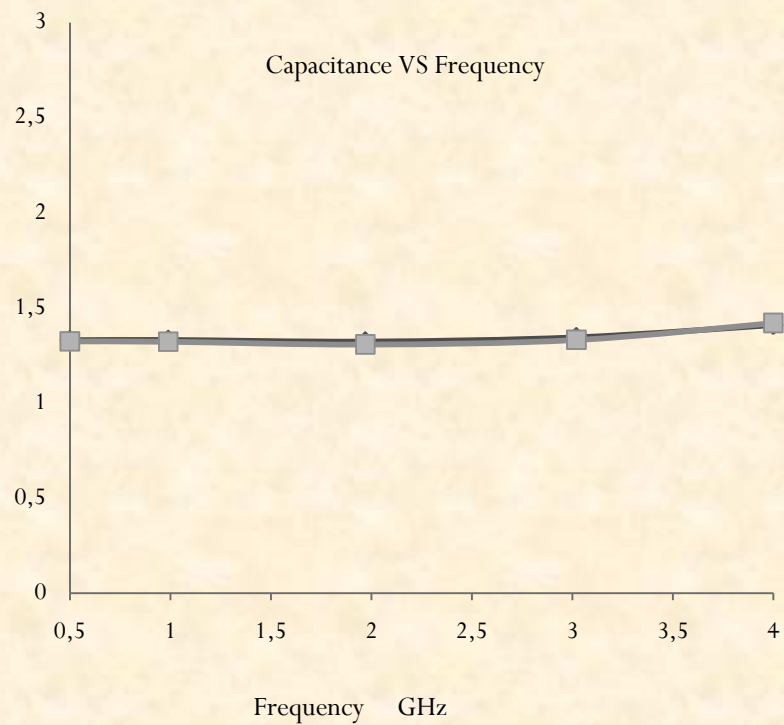


Capacitor test

- Without epoxy
 - $C = 1.33 \text{ pF}$
- With epoxy
 - $C = 1.325 \text{ pF}$
- Res. Freq
- Without epoxy 5.194935420 GHz
- With epoxy 5.193403890 GHz
 - Variation of .2%
- Q without epoxy 182
- Q with epoxy 167
 - Variation of 8%



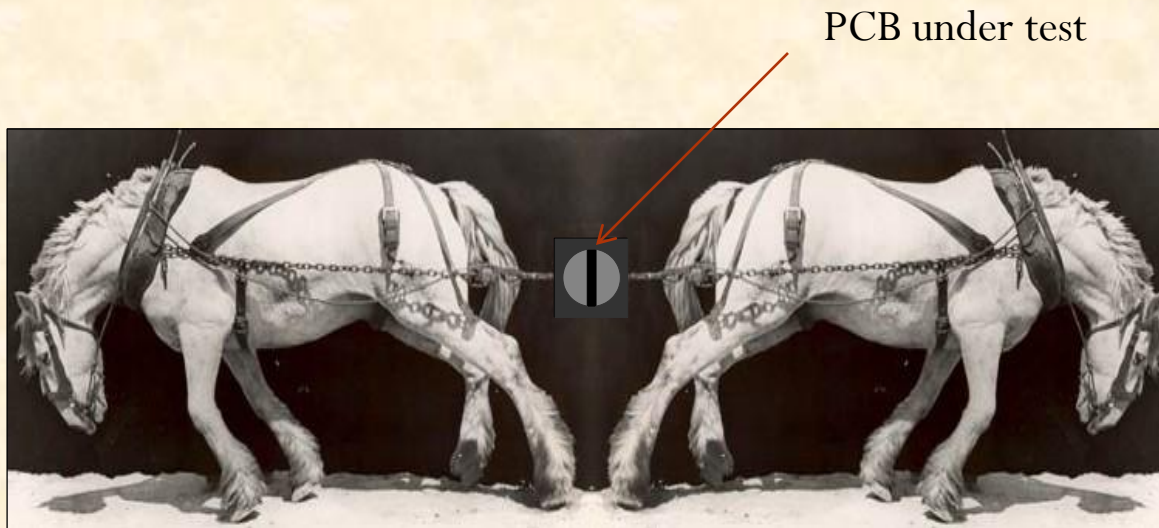
Capacitor test



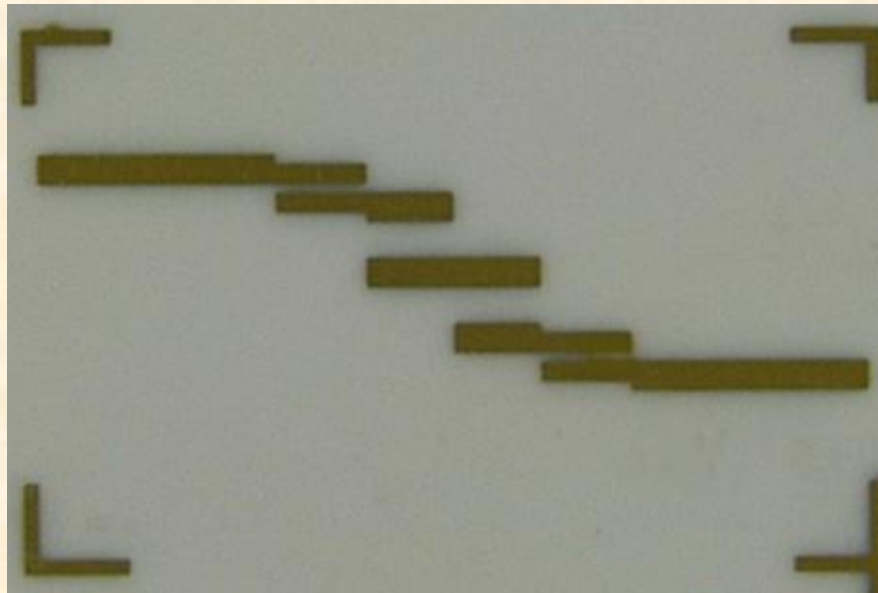
Adhesive pull-test

Copper to copper surface pull force is 300 PSI

Dielectric to dielectric surface pull force is 400 PSI

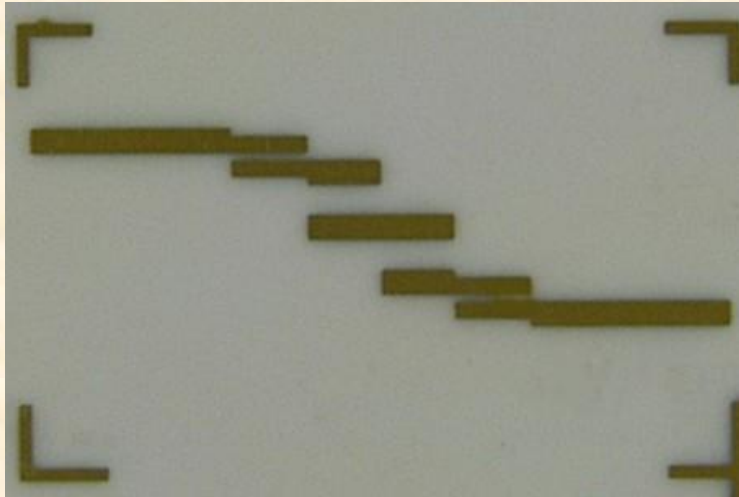


MHMIC Fabrication process

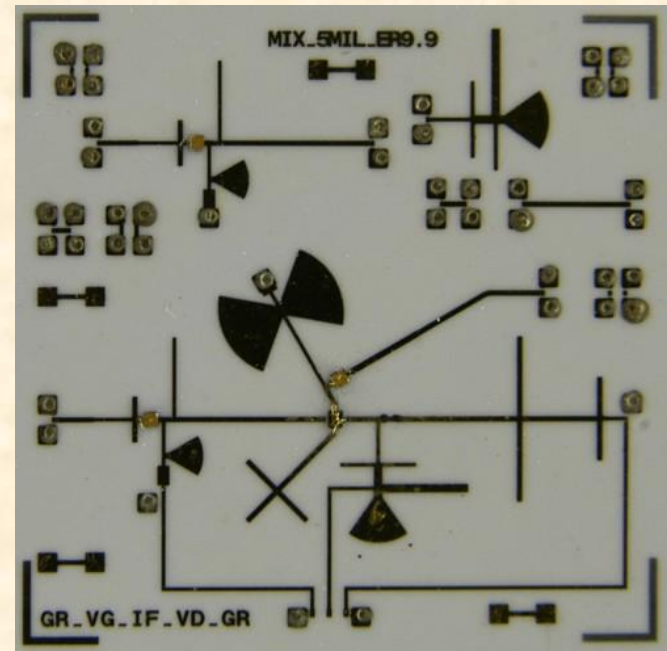


MHMIC

- Without Via

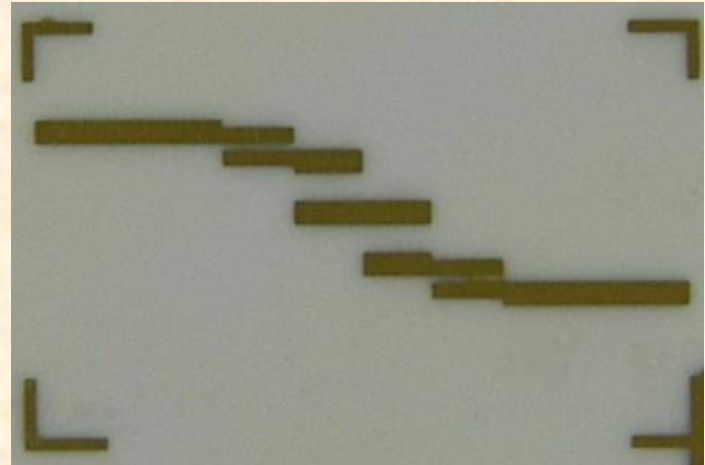


- With Via



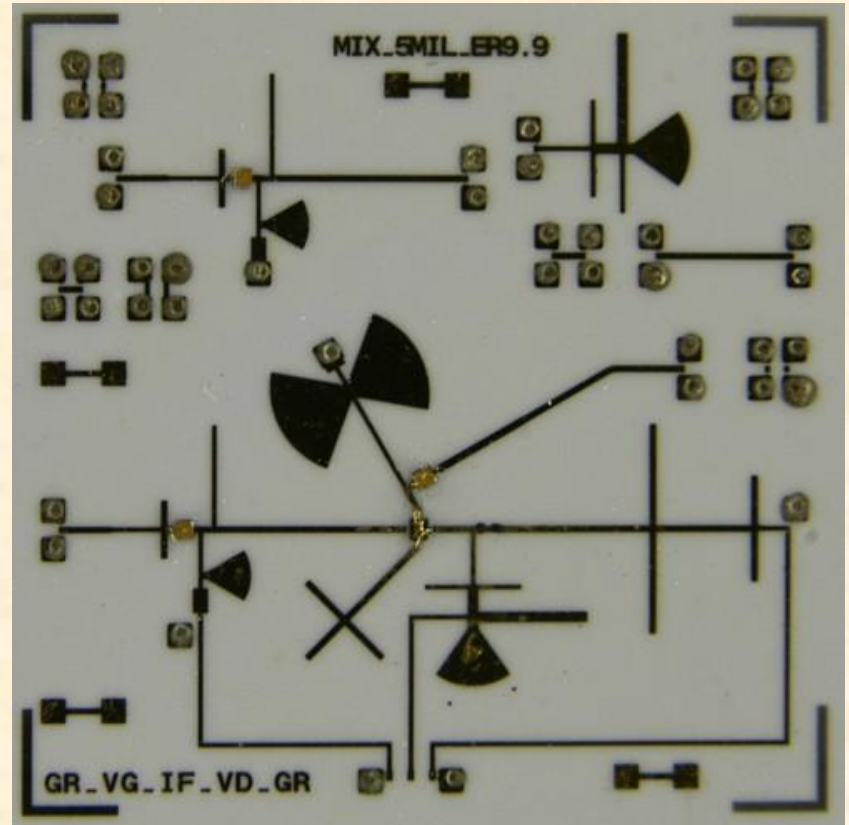
MHMIC without Via

- Circuit is made with photoresist material, resolution is better
- Max size 0.950" x 0.950"
- Ceramic Al_2O_3 10 mil thick
- Min line width 1 mil
- Min line to line gap 1mil
- Resistor 100 Ω/S
- Conductor is Au 1 μm

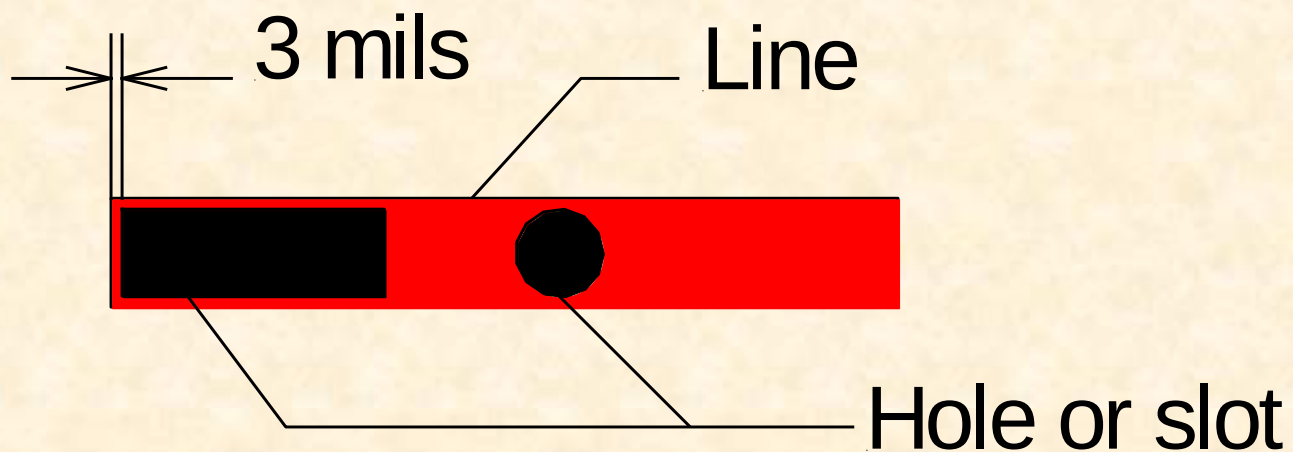


MHMIC with via

- Max size 0.950" x 0.950"
- Ceramic Al_2O_3 10 mils thick
- Min line width 1.5mil
- Min line to line gap 1.5 mil
- Min hole dia. 5 mils
- Overdevelopment of 0.2 mil
- Minimum hole covering is 3 mils



Hole covering



File

Without Via

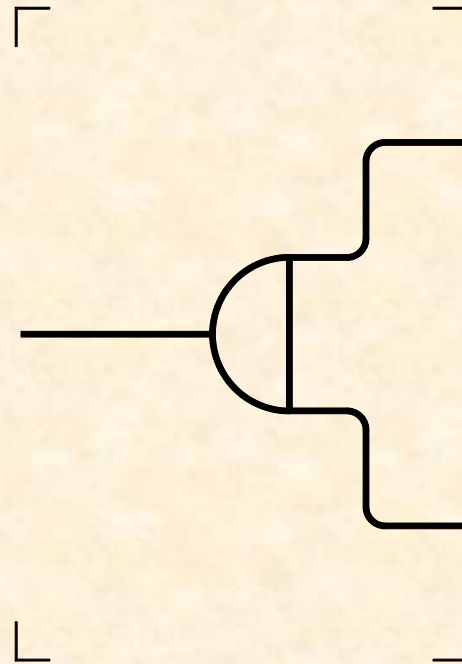
- Mask for conductor: Gerber file scale 5X
- Mask for resistor: Gerber file scale 5X
- Conductor mask has to cover the resistor mask

With Via

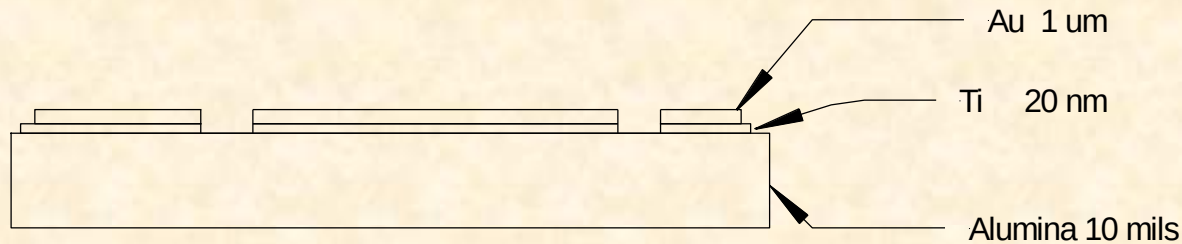
- Mask for conductor: Gerber file scale 5X
- Mask for resistor: Gerber file scale 5X
- Via DXF file 1X
- Slot or rectangle can be cut in the ceramic with laser
- Mask is covering vias

Resistor

- Resistive coating is a layer of 20 nm of Ti under the gold
- First step: conductor is etched including Ti

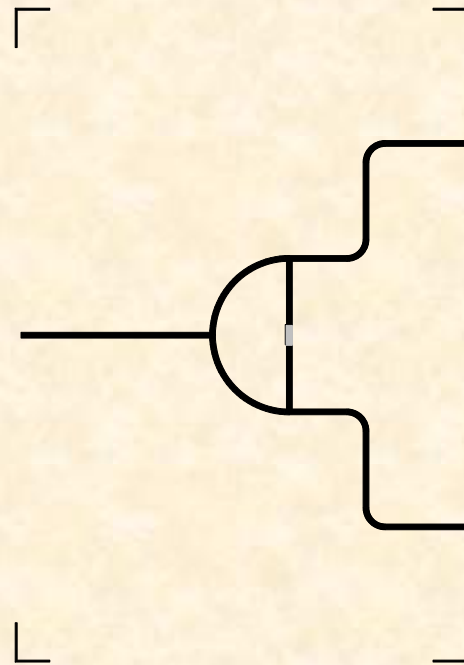


Student's name and Professor
Alumina 10 mil

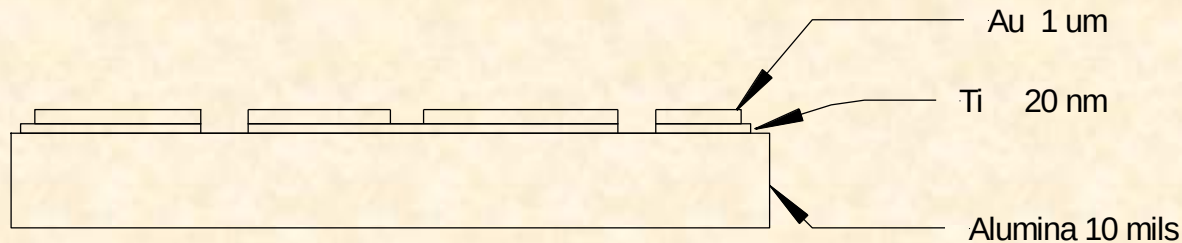


Resistor

- Resistive coating is a layer of 20 nm of Ti under the gold
- First step: conductor is etched including Ti
- Second step: resistor is open trough the conductor

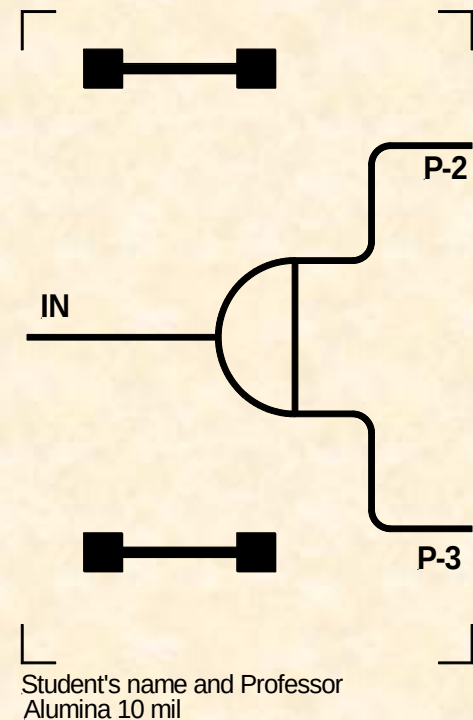


Student's name and Professor
Alumina 10 mil



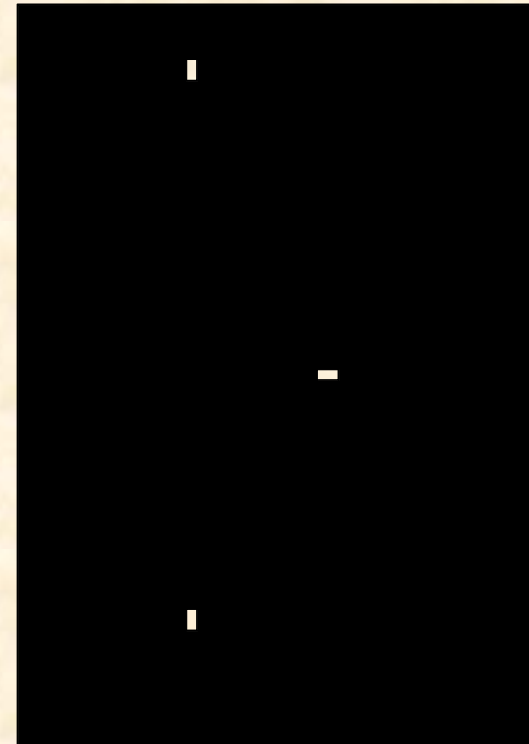
Conductor Gerber file

- The conductor layer is positive mask
- Indicate your name and professor on circuit
- Always print text on circuit
- Indicate cutting mark
- Layout on Gerber scale 5X
- Need at least 2 test resistors

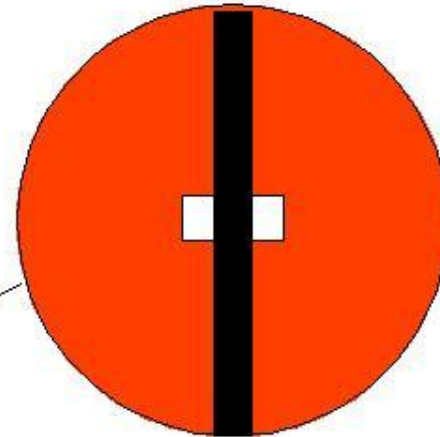
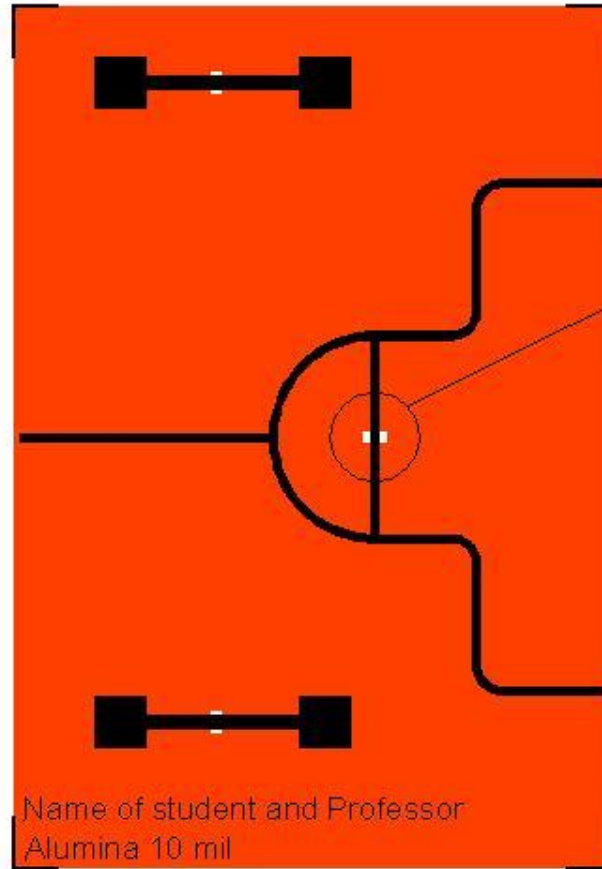


Resistor mask

- Negative mask
- Open only the resistor area
- Mask is Gerber scale 5X
- The opening is larger than the line by 5 mils
- Alignment is corner to corner of circuit in the cutting mark



Resistor mask

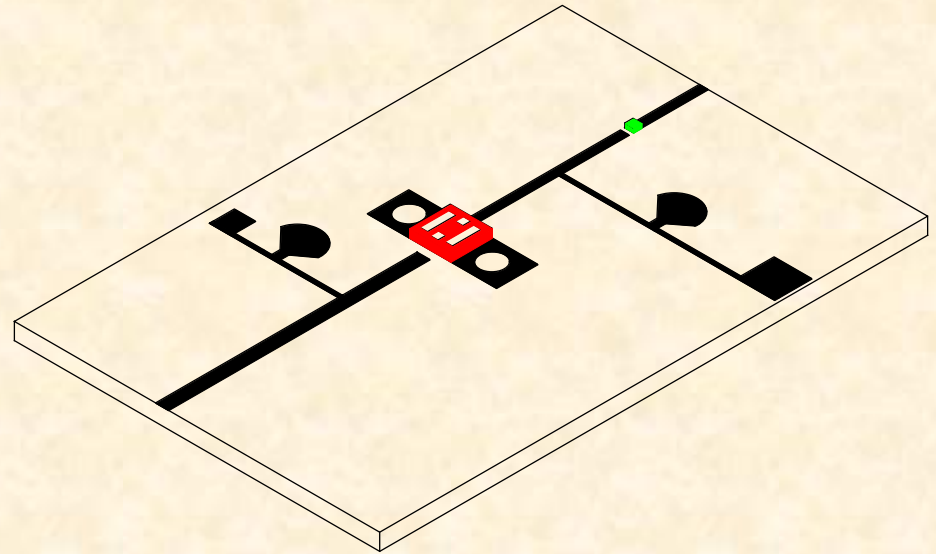


Opening is larger
By .005"

Align inside the
cutting mark

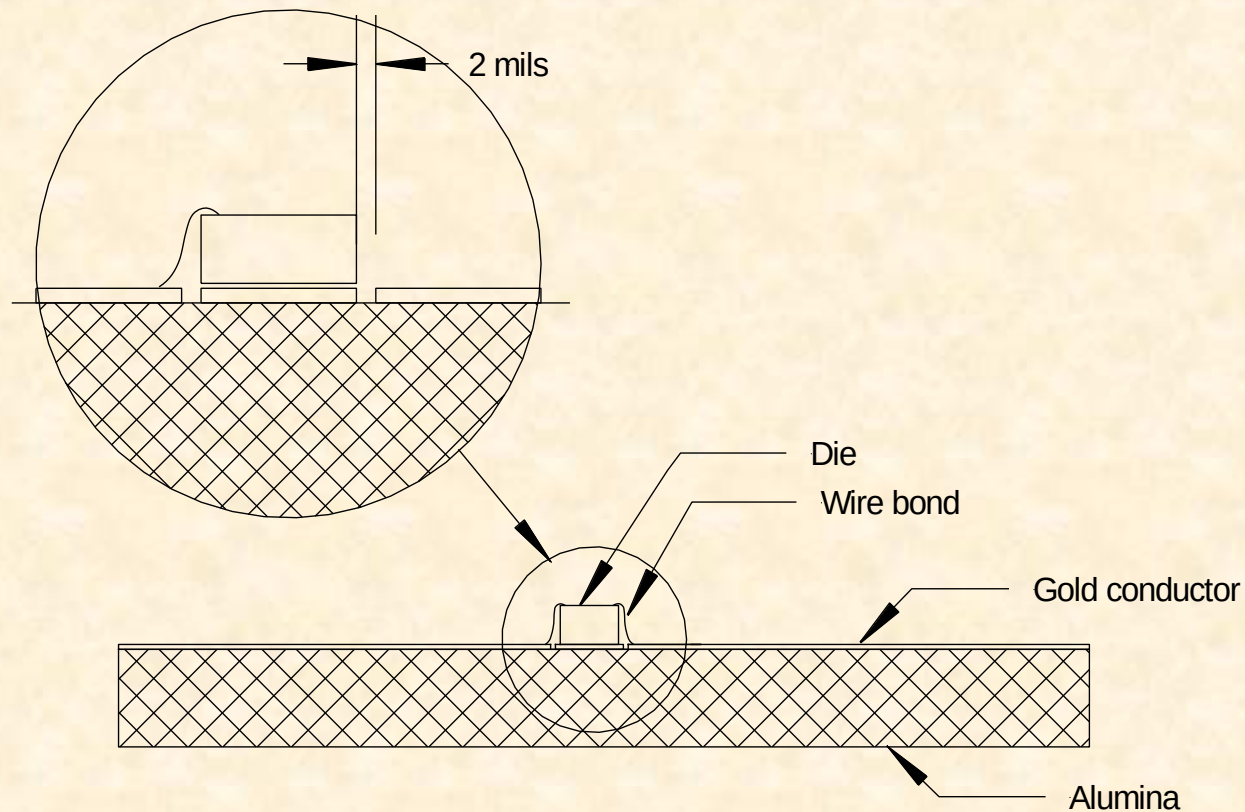
Die assembly

- Mounted on substrate
 - No power dissipation
- Mounted on base
 - Better grounding
 - Power dissipation
 - For higher frequency
- If you use the S parameter of device manufacturer use the same mounting parameters



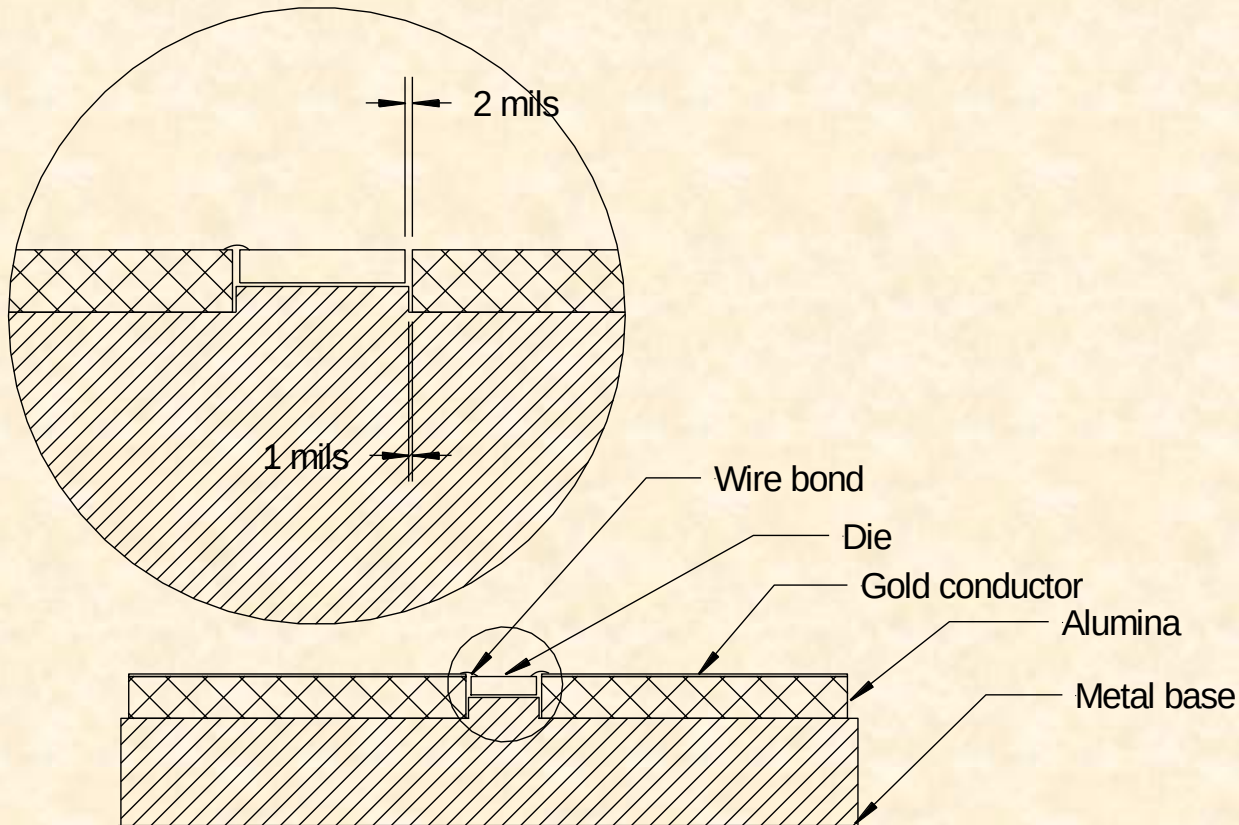
Die mounted on substrate

- Die is bonded onto alumina with conductive epoxy
- A gap of 2 mils is required between die and line



Die mounted on base

- The step is 1 mil larger than Die
- The cutting hole on ceramic is 2 mil larger than Die
- DXF file for mechanical work

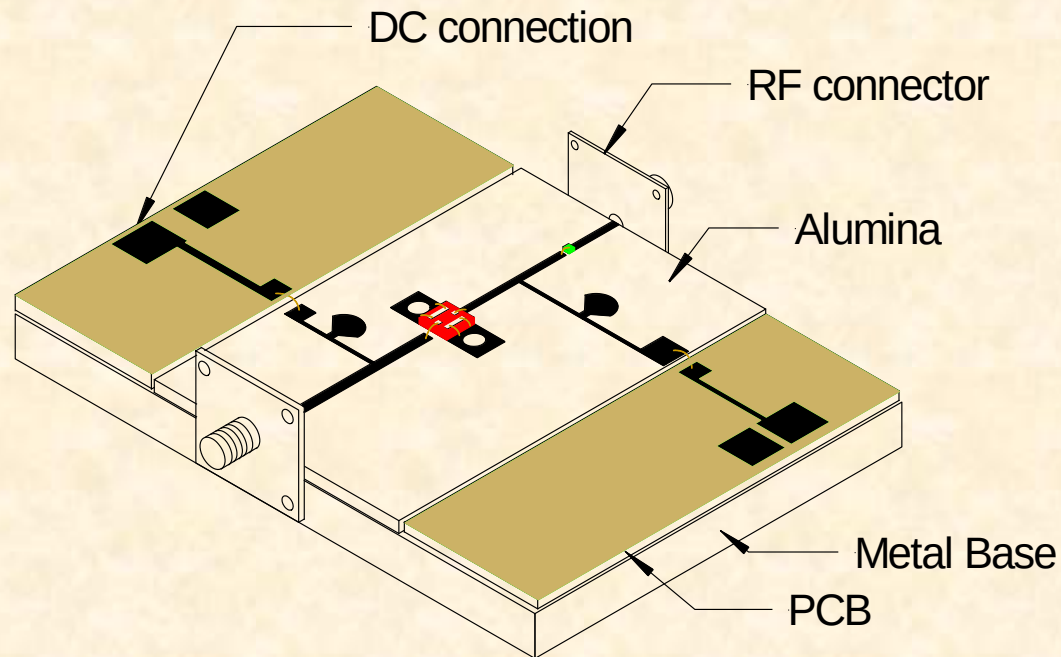


Wire Bonding parameters

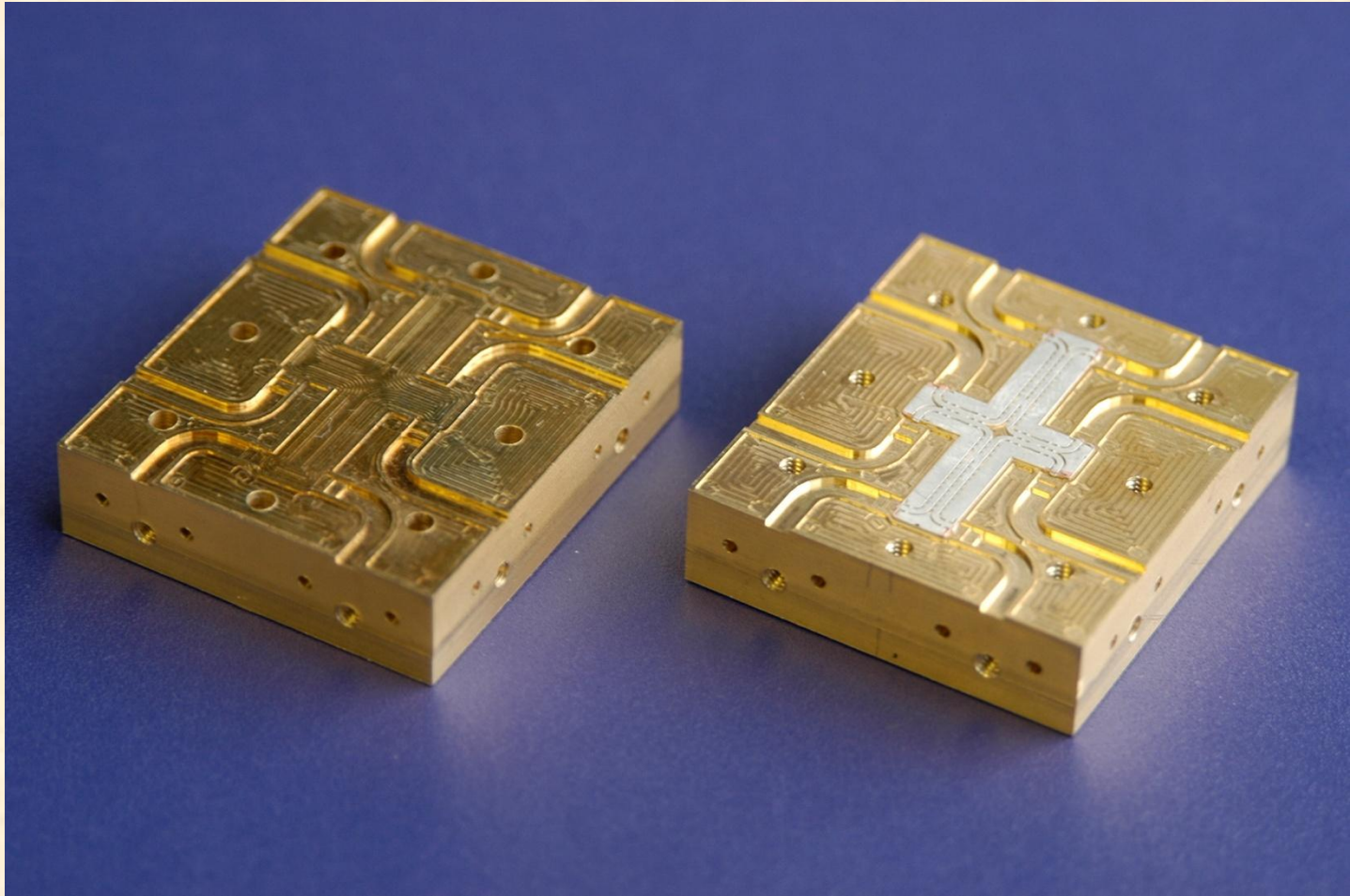
- Wire diameter: 0.7 mil or 18 μm
- Ribbon: 3 mils X 0.5 mil
- For die mounted on substrate bond length is 2 times the height of die
- For die mounted on base the bond length is the spacing between pad and line
- Always provide a wire bonding layout

DC connection

- It is impossible to solder wire to thin film gold
- Small PCB line is added to make interconnection

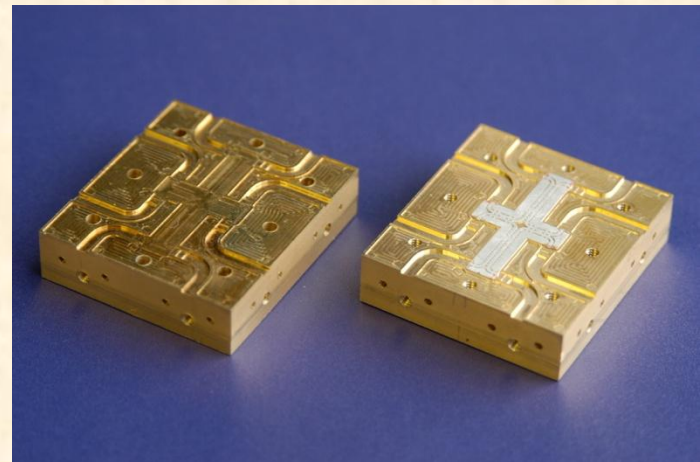


Machined waveguide



Machined waveguide

- For 2D
 - Drawing needs to be a DXF file
 - Drawing is scaled 1:1
 - Unit is inch
 - Indicate material Al or Brass
 - Side view of piece with indication of height
- For 3D
 - Drawing needs to be .SAT or VDA, not DWG
 - Drawing is at scale 1:1
 - Unit is inch
 - Indicate material Al or Brass



Circuit test

- Never forget, you need to test your circuit
- Anritsu Test fixture
 - Microstrip
 - Coplanar
 - 40 GHz 4"
 - 65 GHz 2"
- Connector SMA, K, V
 - Circuit needs to be mounted on a base
 - To be provided by your professor
 - Take care of spacing between connectors 0.5"
- SIW requires waveguide



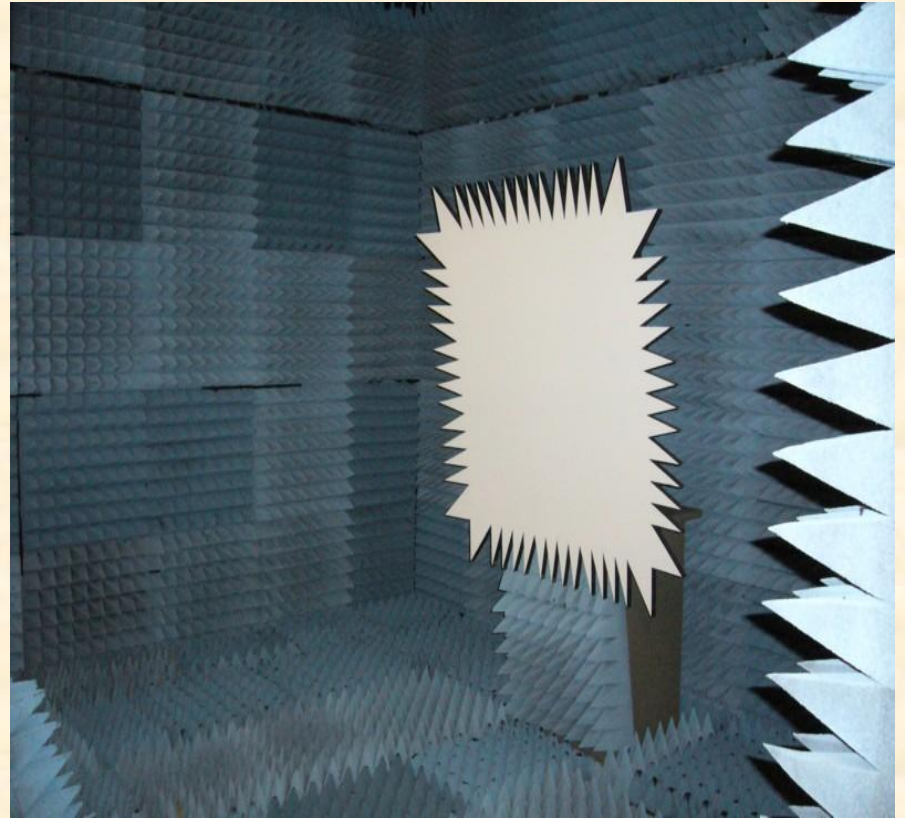
Probing station

- Probe size 150 μm and 250 μm
- Frequency range up to 110 GHz
- Coplanar or microstrip



Antenna

- Compact Range
- Frequency band
 - 4 to 40 GHz
 - 75 to 110 GHz
- Quiet zone
 - 75 cm x 75 cm x 75 cm



Useful softwares

- Gerber viewer
 - ViewMate Pentalogix (on the 97 and 100 servers)
- Dxf editor and viewer
 - Autosketch (on the 097 and 100 servers)
- Unit converter
 - Convert.exe (<K:\convert\convert.exe>)
- AutoCAD to Gerber translator
 - LinkCAD 4 (Free demo at <http://www.linkcad.com/>)

Please take note that the layout will look exactly as in Viewmate, so, before sending any Gerber file, double-check it, including the size (D-key in the software)!

How to name the file

- To simplify the comprehension please include in file name the following information
- **Name-circuit-layer-substrate-thickness.ext**
- Example:

Gratton-converter-S1T-5880-20.gbr

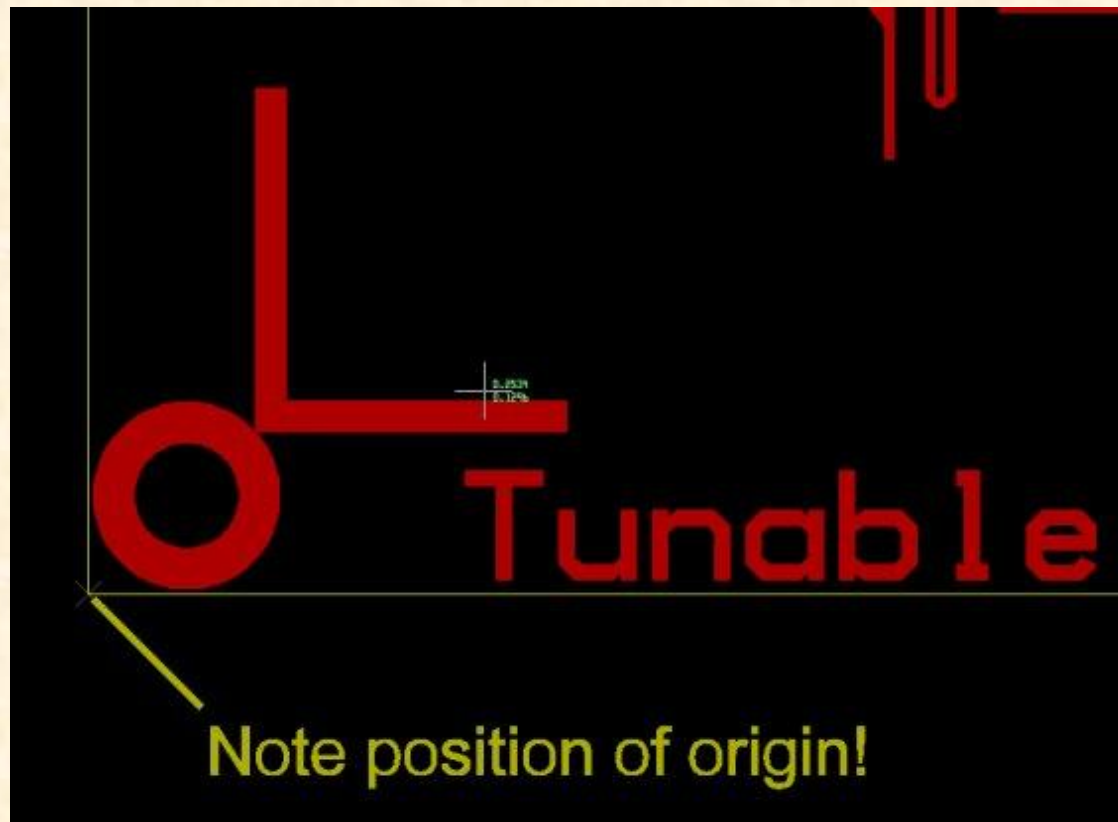
Gratton-converter-S1T.dxf

Substrate

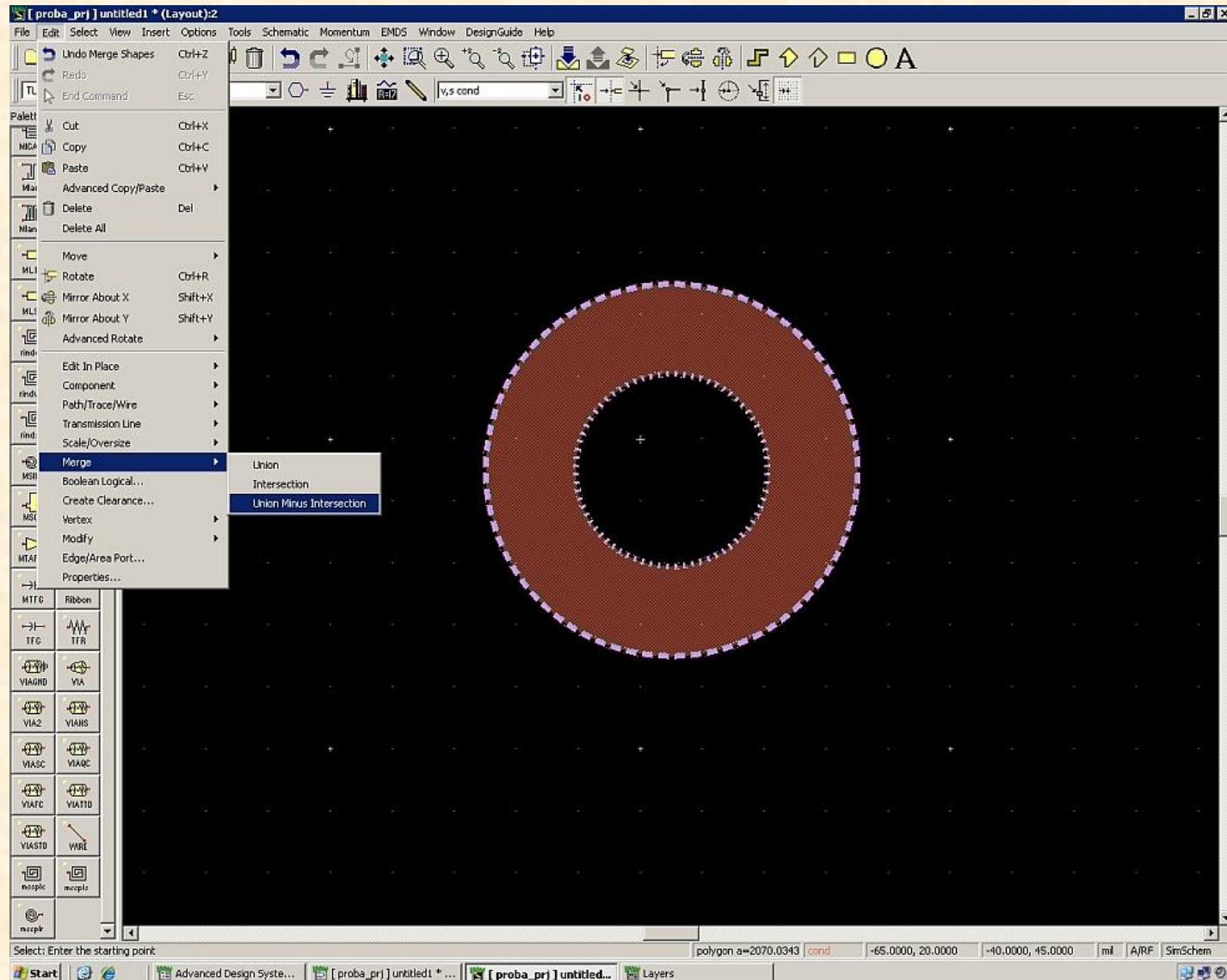
- Rogers Laminates <http://www.rogerscorp.com/acm/>
- 5870, 5880, 6002, 6006, 6010, 4000, 3000
- Order: rolled copper, 0.5 ounce thick (5R/5R)
- Sample is available on the web site, limited to 2 samples
- **Warning!** 5870 and 5880 substrates cannot be drilled using a laser beam. We cannot fabricate slots into these ones, also, you have to use standard hole sizes: 0.4, 0.5, 0.6, 0.7, 0.8, 1, 1.4, 1.6, 1.7, 2 mm diameter.

Useful hints

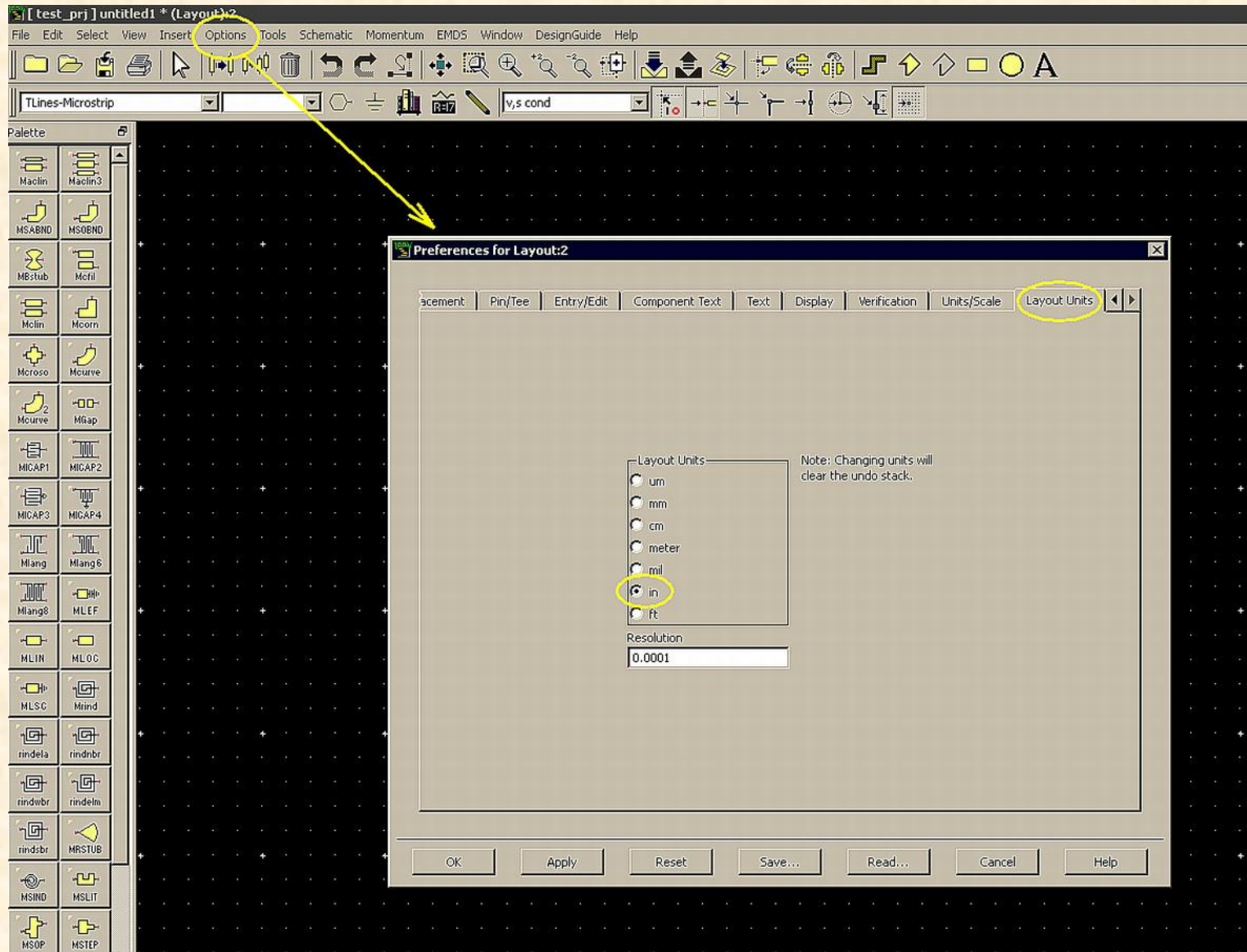
- Alignment donut and origin position
- Nothing has to be found at the left and under the x and y axis



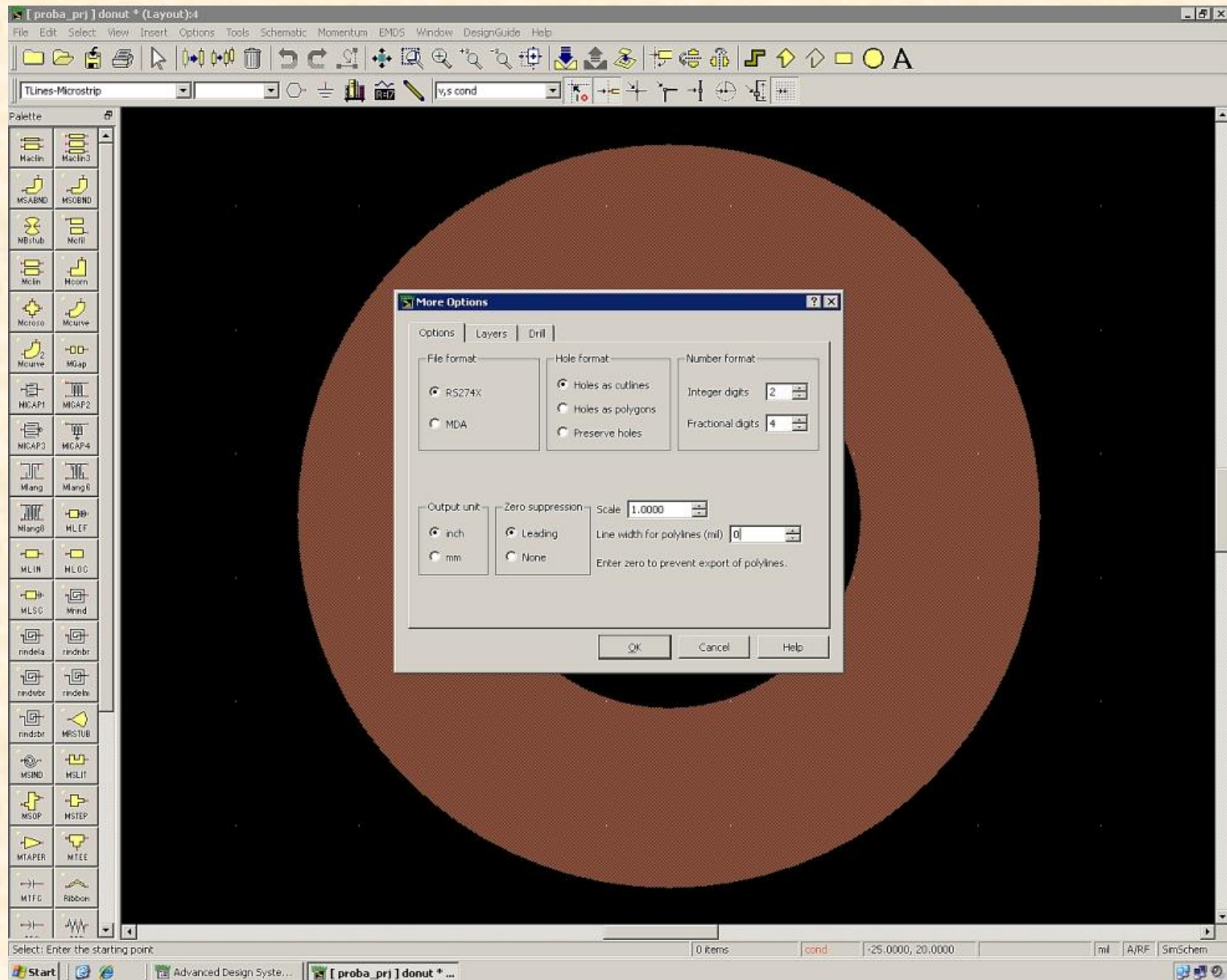
- How to make an alignment donut in ADS: make two, proper sized circles, align them and – Edit-Merge-Union minus Intersection



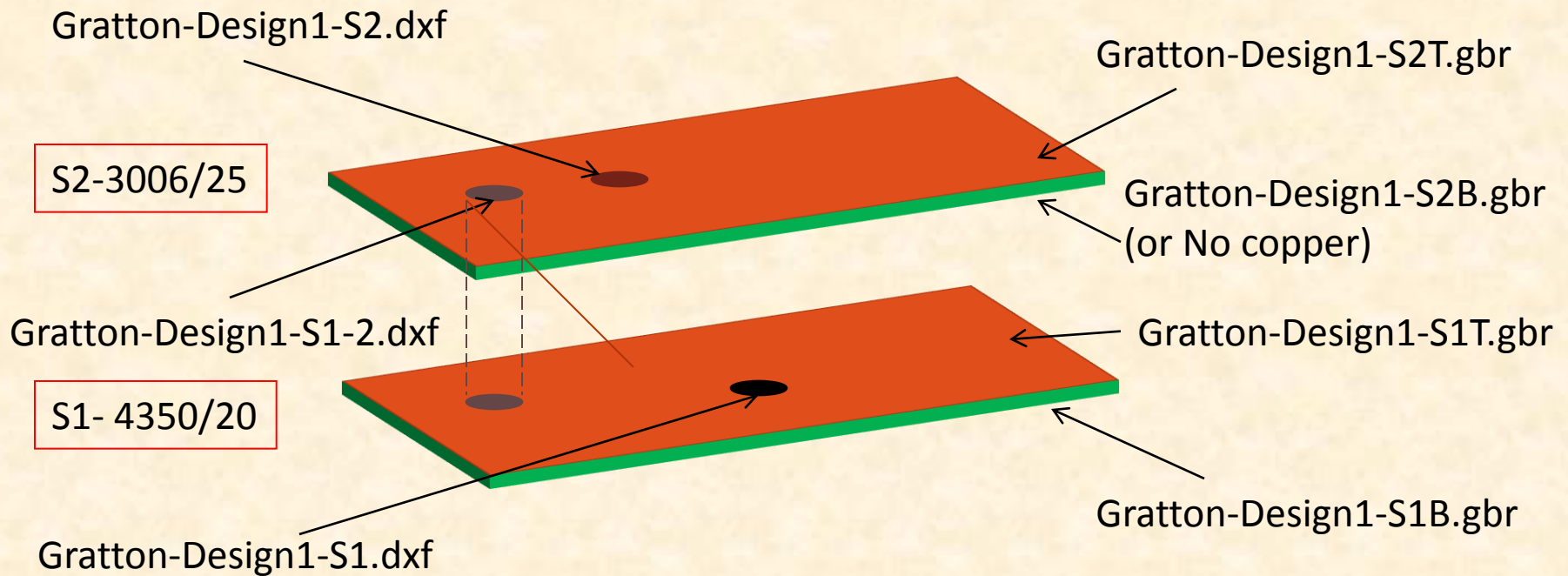
- Proper units in ADS: adjust the proper units before starting to work in the layout window (Options-Preferences-Layout Units-in)



• Gerber file export



- Multilayer drawing explanation



IMPORTANT!

Before sending any fabrication file, please take your time and pass the checklist below:

- Does the layout comply with the requirements stated in the document?
- Is the Gerber file exported in the proper format (RS274X, 2.4 precision)?
- Are units in inches?
- Is the DXF file for drilling in the proper units and format (inches, 2.4, dxf2000)?
- Has the DXF file only one layer?
- Are the holes circles, not segments?
- Did you include the substrate type and thickness?
- If you need a base, did you provide a 2D drawing in DXF? (your drawing software allows you to transform any 3D sketch in a 2D one)
- Did you include all extra information needed to fabricate the circuit and the base?
- Did you rename all your files (Gerber and dxf) correctly?

Please take note, that if your files don't comply with the requests, we are not able to process them, hence some major delays will occur.

Please send all your fabrication requests to: traian.antonescu@polymtl.ca