RF circuit fabrication rules

http://www.grames.polymtl.ca/PCB-Process/RF_circuit_fabrication_rules.pdf

Poly GRAMES

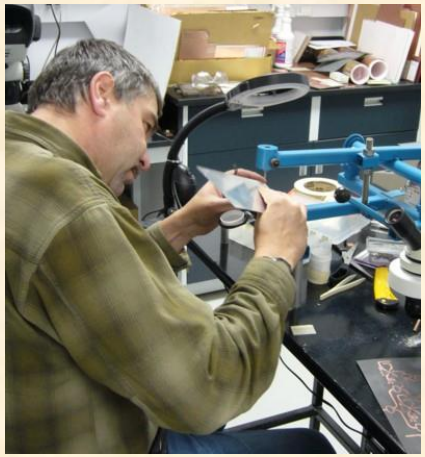
creer
centre de recherche
en électronique
radiofréquence

Content:

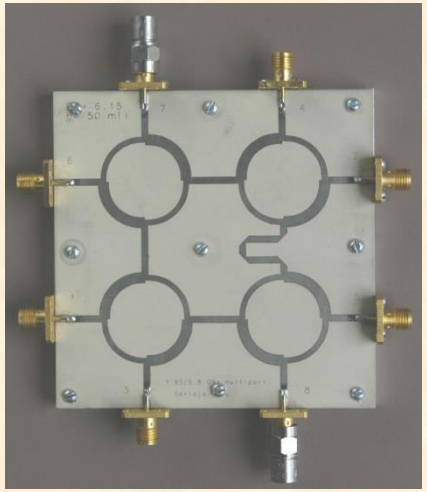
*You can use the underlined links to have direct access to the desired chapter

- Single layer (ref. page 4)
  - No vias (ref. page 4, 17-26)
  - With riveted vias (ref. pages 4,5,6, 17-26)
  - With plated vias (ref. pages 4, 5,7-11, 17-26)
  - SIW (ref. pages 4,5, 7-12, 17-26)
- Multilayer (ref. pages 4,5, 7-11, 13-16, 17-26)
- Appendix (ref. pages 17-26)

Team



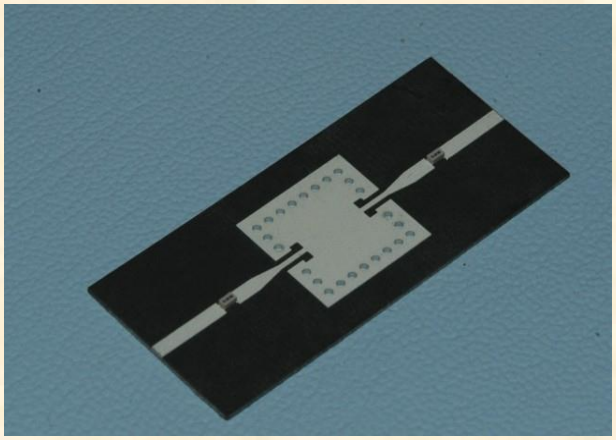
Single layer

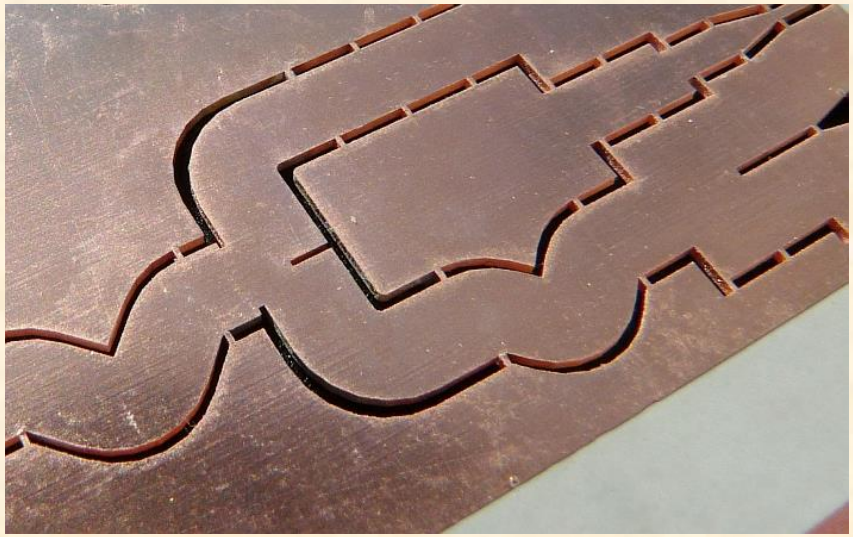


Specifications for circuits

- Max size 8”X 8”
- Min. line width 6 mils
- Min. gap between lines 6 mils
- 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 dimensions: 200x200x20 mils
- Origin at the lowest left corner of the layout (nothing outside the first quadrant)
- Connector spacing minimum 0.5”

Single layer with vias or slots

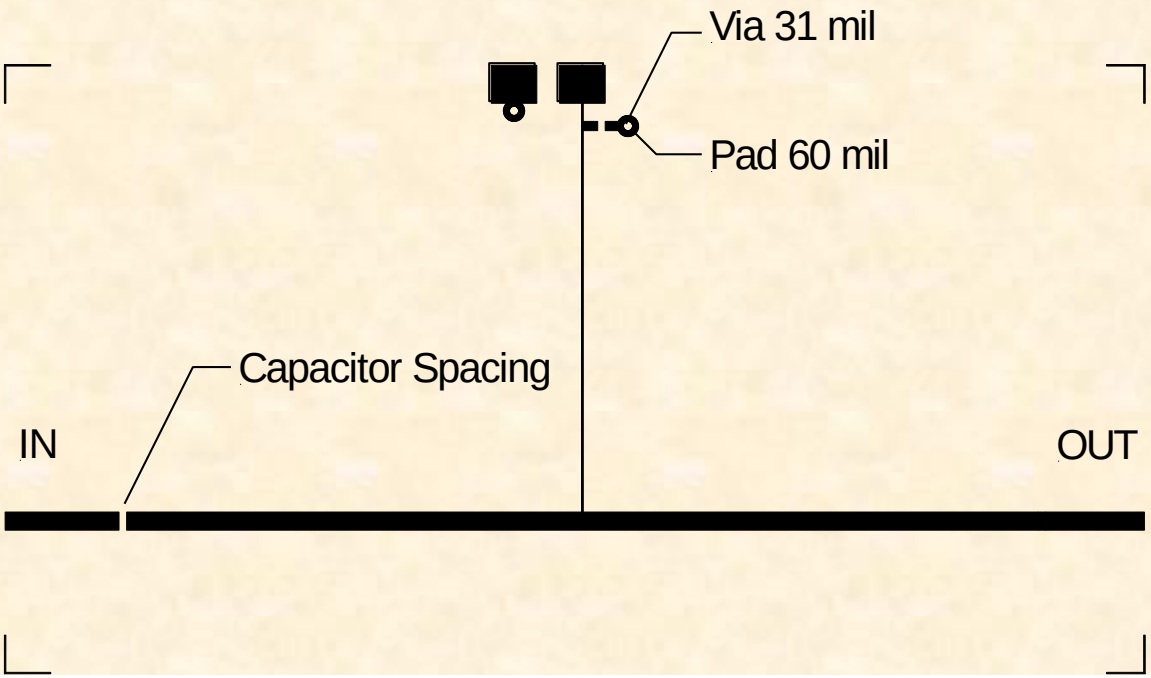




- Same as single layer +
- Riveted vias
  - Used only if circuit contains less than 15 holes
  - Hole 31 mils diameter only
  - Pad 60 mils diameter only
  - Center of pad uncovered in layout
- Plated vias or slots
  - Used for large quantity of holes on circuits or if hole size is critical
  - Min. aspect ratio of 1:2 (minimum hole diameter is half of substrate thickness)
  - Center of pad covered in 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 diameter



File name, Student/Professor Substrate/thickness

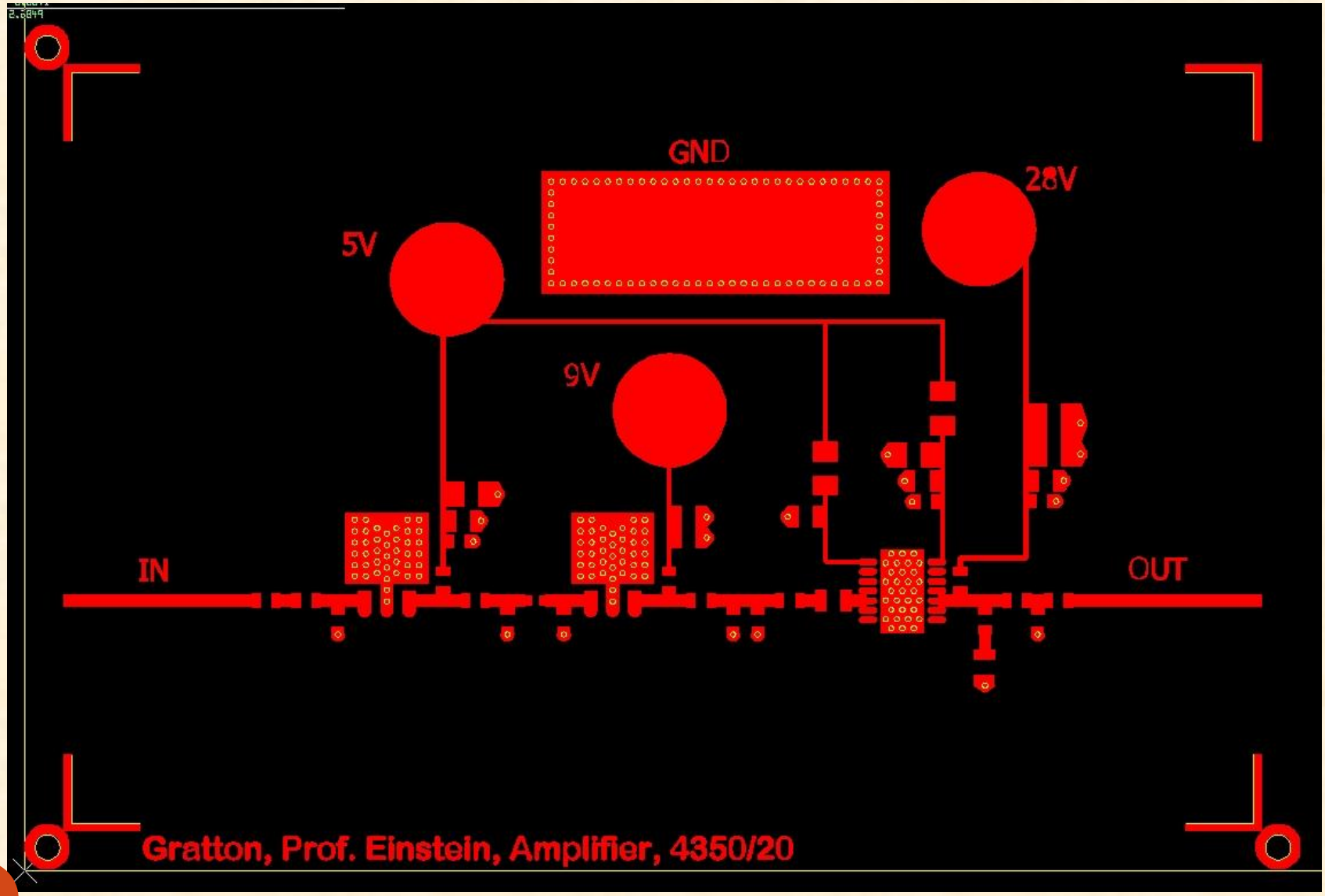
Plated Vias

- Layout file: Gerber RS-274x, precision 2.4 for Top (and) Bottom
- Hole file .dxf, 2.4, single layer
- Holes are filled on the layout (Gerber file)
- Three alignment holes of 66/120 mils diameter with donuts outside of layout
- Aspect ratio for plated hole is min 1:2 (minimum hole diameter is at least half of substrate thickness
- Clearance of 10 mils (between hole edge and copper edge)
- Max PCB size 6" x 6" for plated holes or slots (limited by plating machine size)

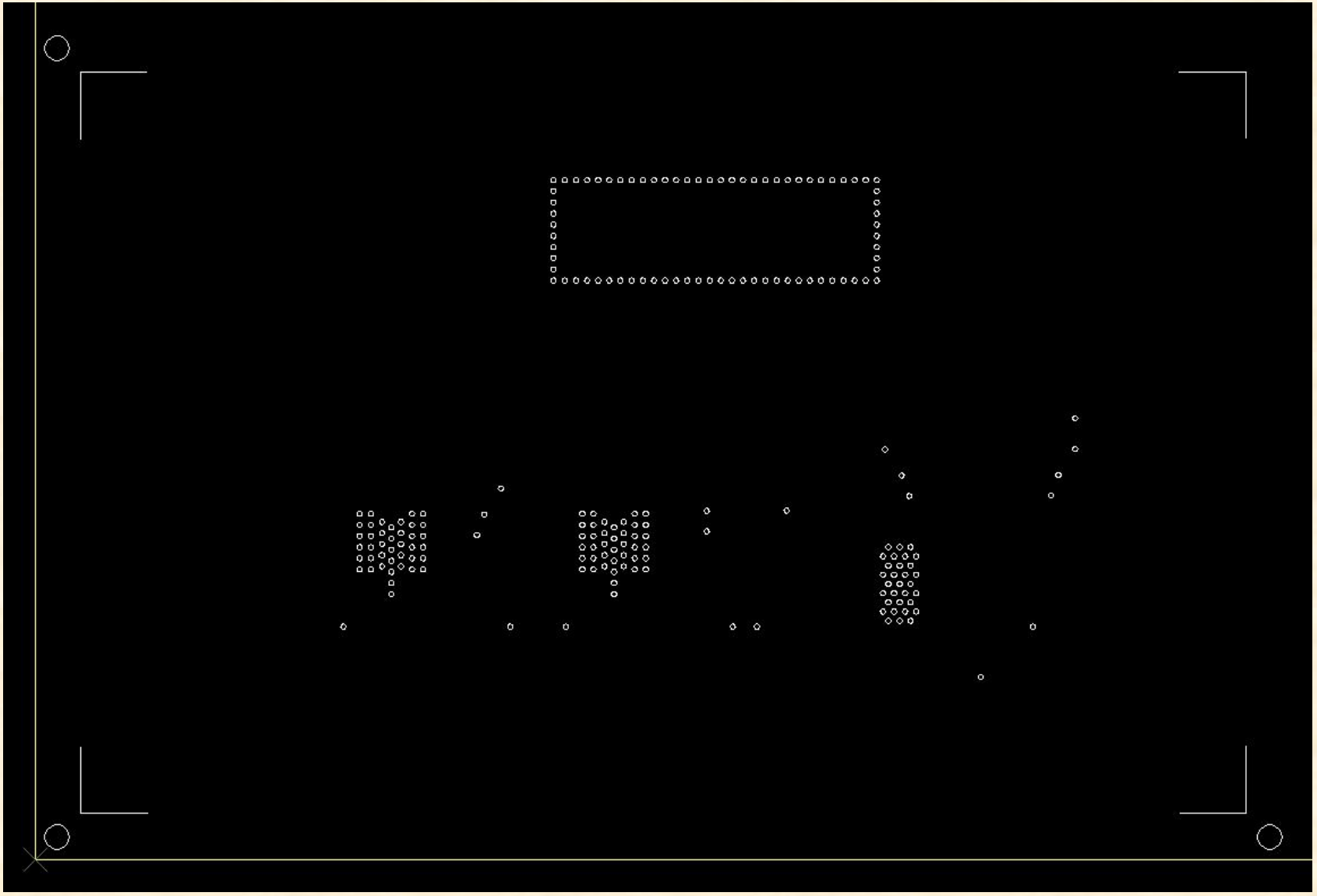
Laser drilling

- Holes need to be circles, not line segments or polylines
- Minimum space of 10 mils between 2 hole edges
- For soft substrates (RO5880) maximum thickness is 60 mils
- Minimum hole diameter 5 mils (Attention – aspect ratio)
- Cutting oversize is 2 mils (ex. 20mils diameter hole will become 22 mils

Plated Vias – Gerber and dxf files

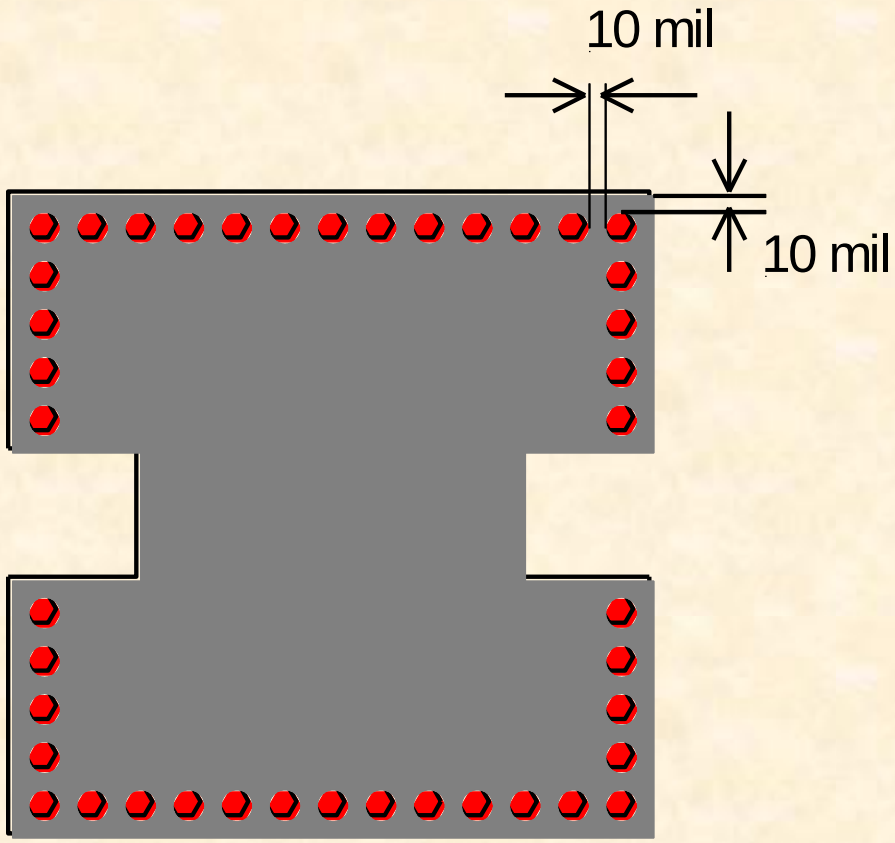


Plated Vias –dxf files

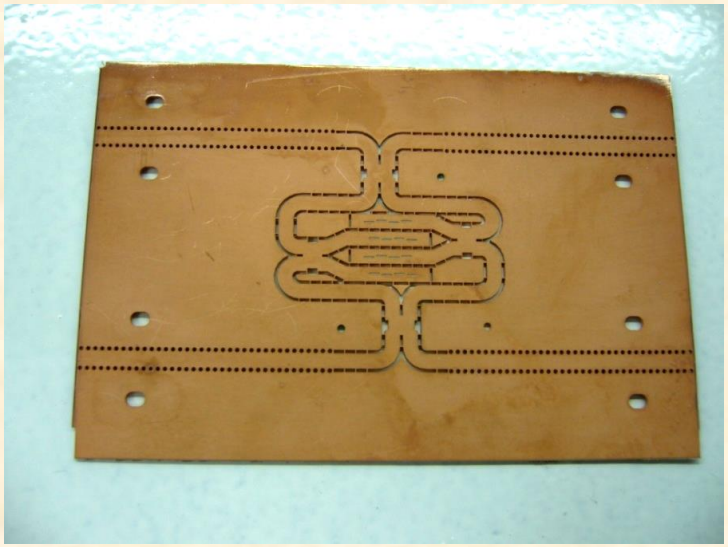


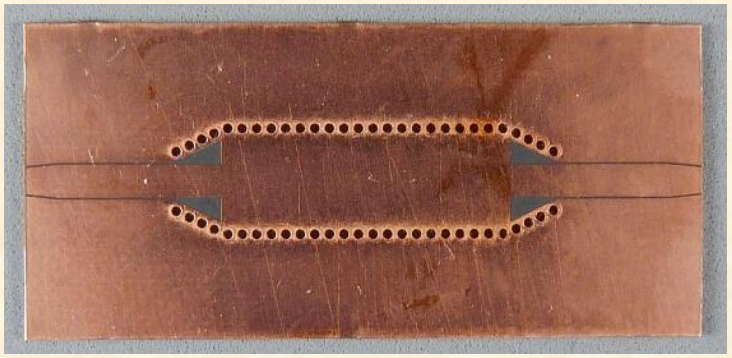
Clearance

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



Substrate-Integrated-Waveguide

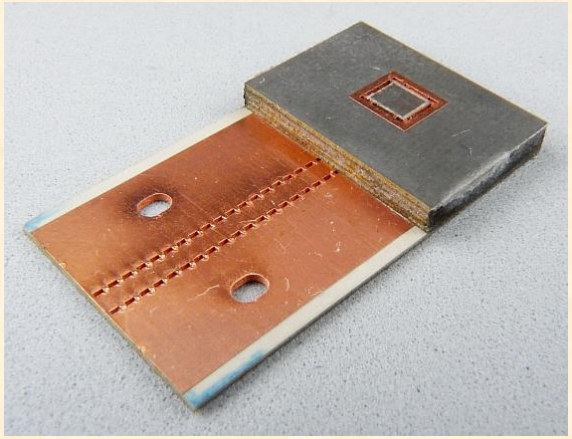


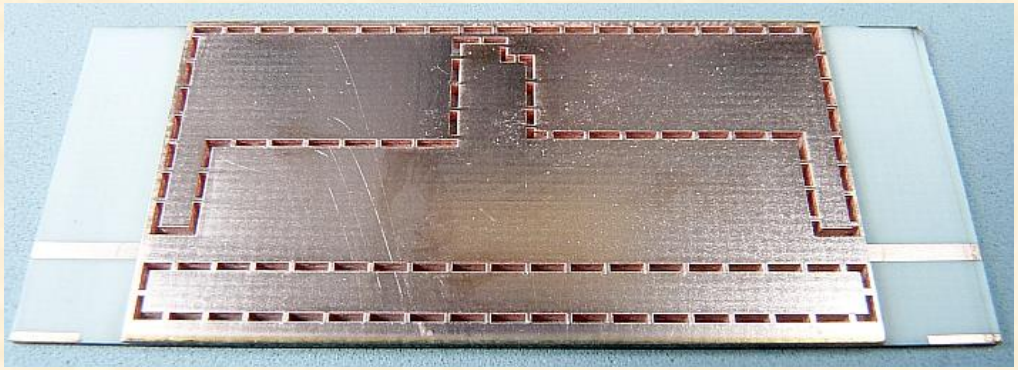


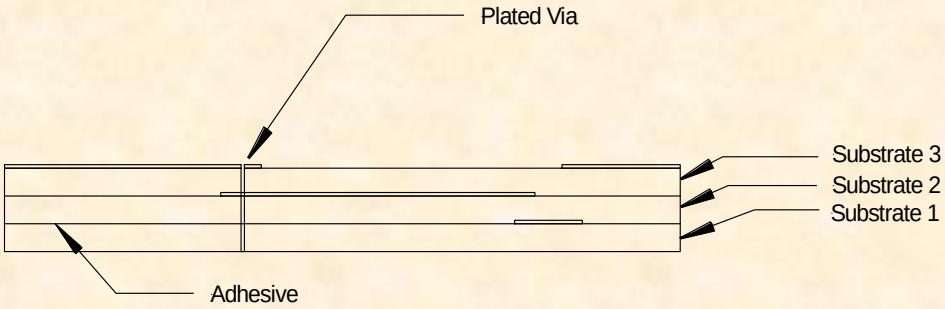
- Holes or slots are cut by laser
- Consider an oversize of 2 mils for laser cutting
- Laser cuts only through-hole
- Aspect ratio of minimum 1:2 (minimum hole diameter at least half of substrate thickness) has to be maintained for plating quality
- dxf file precision 2.4, single layer

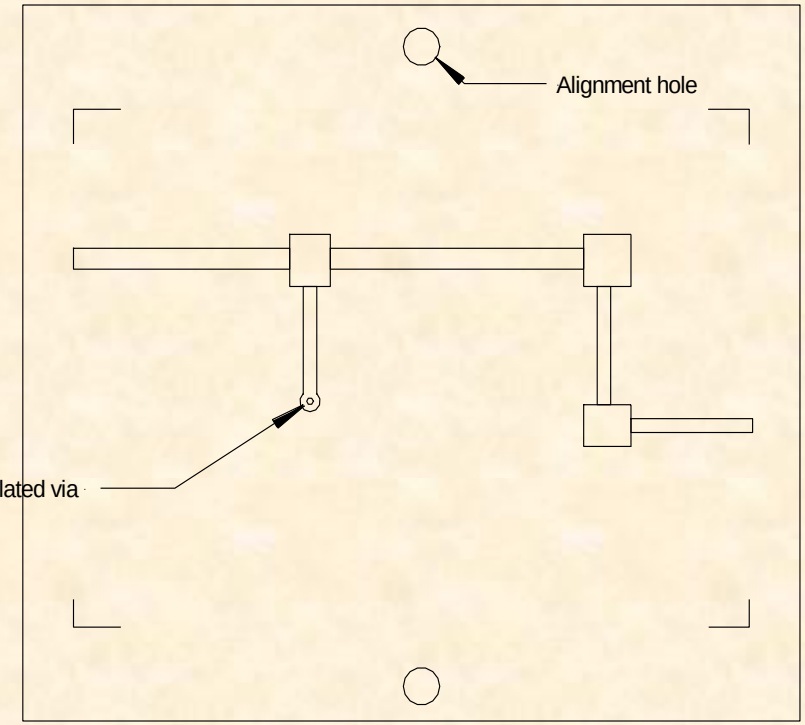
Multilayer

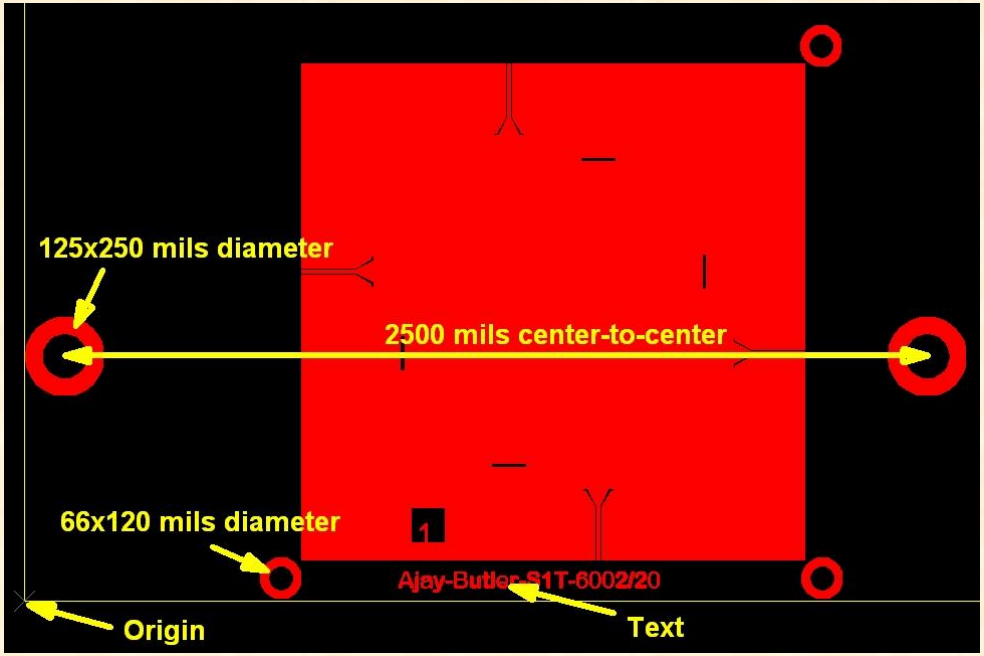
- In order to make complex RF structures, we developed a method to assemble several substrate layers.
- Epoxy glue, high temperature and pressure are used to combine the layers.
- The layers can be printed, have plated slots or holes
- The final assembly can also be printed, drilled, milled, plated









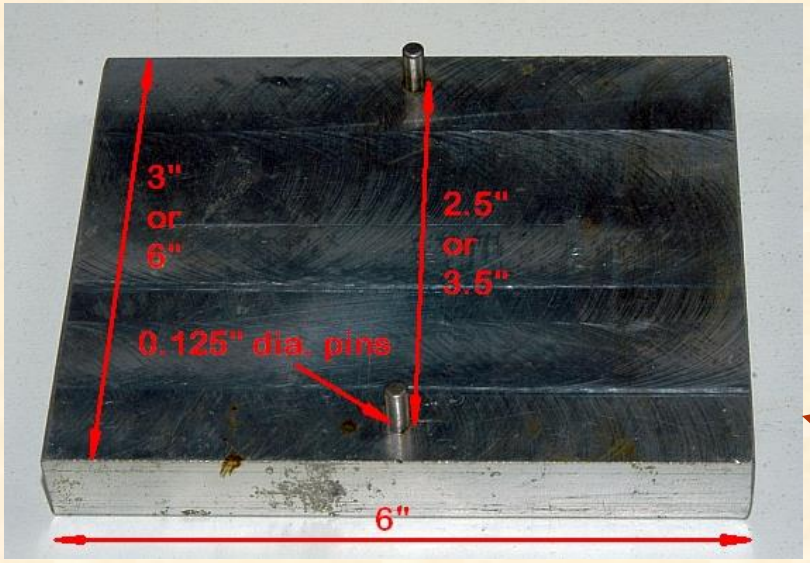


Side and top view of multilayer circuit

More information for multilayers……

- Need a 3 D drawing to represent the entire circuit
- Plated holes and inner connections are feasible
- If connection is required between two substrates, a plated via is required
- Max sizes are 3" X 6" and 4"x6"

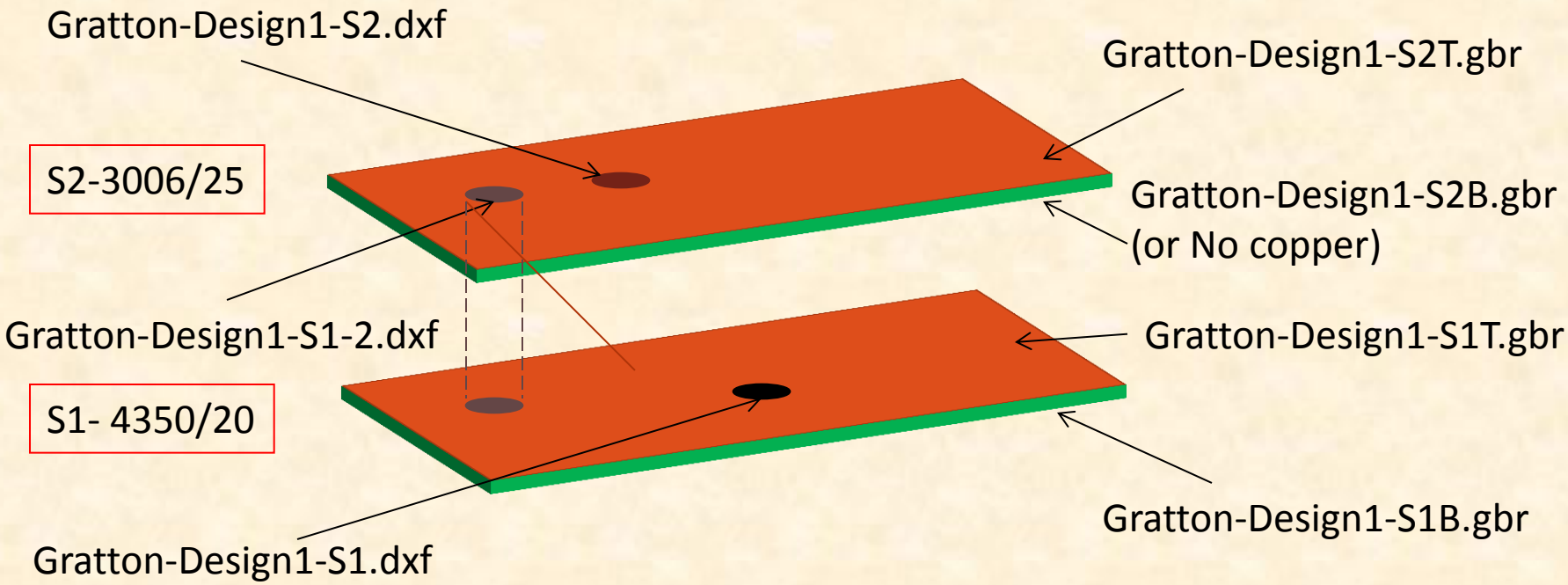
(please leave space for assembly pins)



Multilayer assembly fixture with dimensions

- First substrate is on the bottom
- Text indicating the layer name has to be printed on each layer
- Indicate on the drawing the top or bottom layer
- Ex: S1B (substrate 1 bottom)
- Gerber file for each layer
- Drill file is a .dxf
- The alignment between substrates is made using 0.125" diameter pins at 2.5" center-to-center distance
- The layout contains corresponding donuts of 125 mils internal and 250 mils external diameter

- Multilayer drawing explanation



To simplify your work, just replace the file names and substrate types, then save this page (document name has to contain your name) and send it along with the other files.

Useful software and links

- Gerber viewer: **ViewMate** (\\132.207.65.40\Exemples\PCB-Process\viewmate.rdp)
- Dxf editor and viewer: **Autosketch** (\\132.207.65.40\Exemples\PCB-Process\Autosketch.rdp)
- Unit converter: **Convert** (\\132.207.65.40\Exemples\PCB-Process\Convert.rdp)
- AutoCAD to Gerber translator, dxf file checker, duplicate remover, circle maker: **LinkCAD** (\\132.207.65.40\Exemples\PCB-Process\LinkCAD.rdp)
- Circuit schematic and printed circuit designer: **Altium Designer** (\\132.207.65.40\Exemples\PCB-Process\AltiumDesigner.rdp)

- Layout units adjustment in ADS2011
- Ajuster les unités de mesure en ADS2011
- Multilayer assembly drawing
- Transform polylines to circles
- Transformer des polygones en cercles
- Obtain Gerber files from HFSS

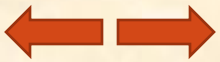
How to name the file

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

Gratton-converter-S1T-5880-20.gbr

Gratton-converter-S1B-5880-20.gbr

Gratton-converter-S1T.dxf

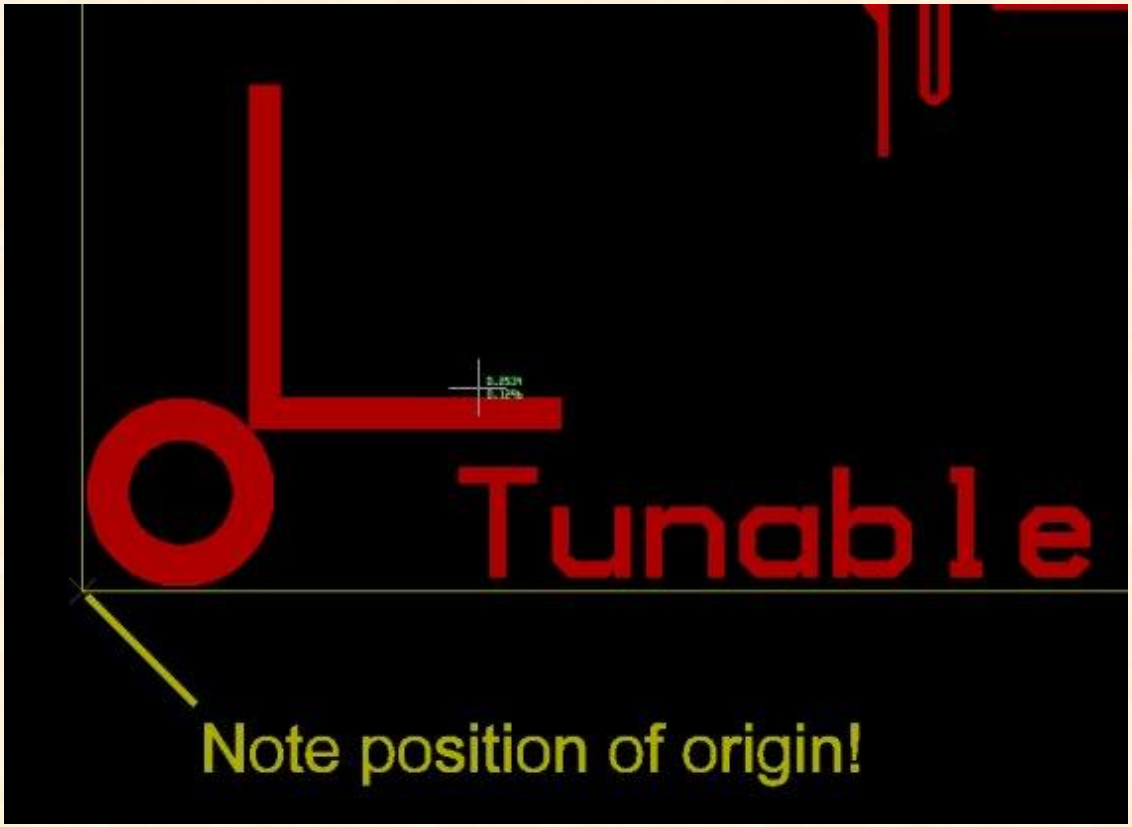


Substrate

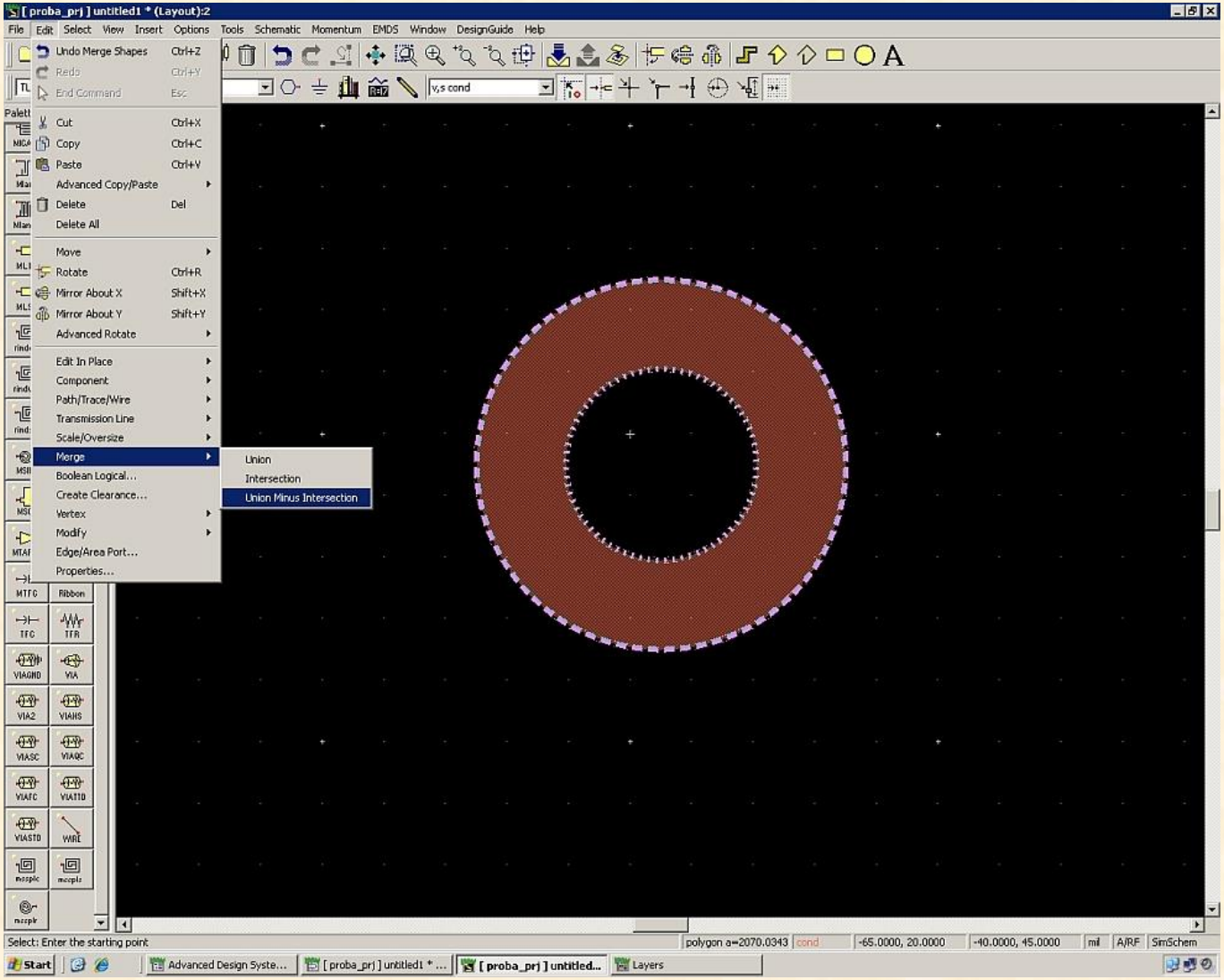
- Rogers Laminates http://www.rogerscorp.com/acm/
- 5870, 5880, 6002, 6006, 6010, 4000-series, 3000-series
- Due to laser-cutting constraints, it is impossible to cut into 5870 and 5880 substrates thicker than 62 mils.
- Please take into consideration that some substrates are very soft, hence less suited for certain circuits. If in doubt, please come to get an advice.
- Order: rolled or electrodeposited copper, 0.5 ounce thick (**5R/5R, or 5ED/5ED or HH**). It is important to respect the copper thickness for quality and manufacturing reasons.
- Samples are available on the Rogers web site, please ask for the correct copper thickness.

Useful hints

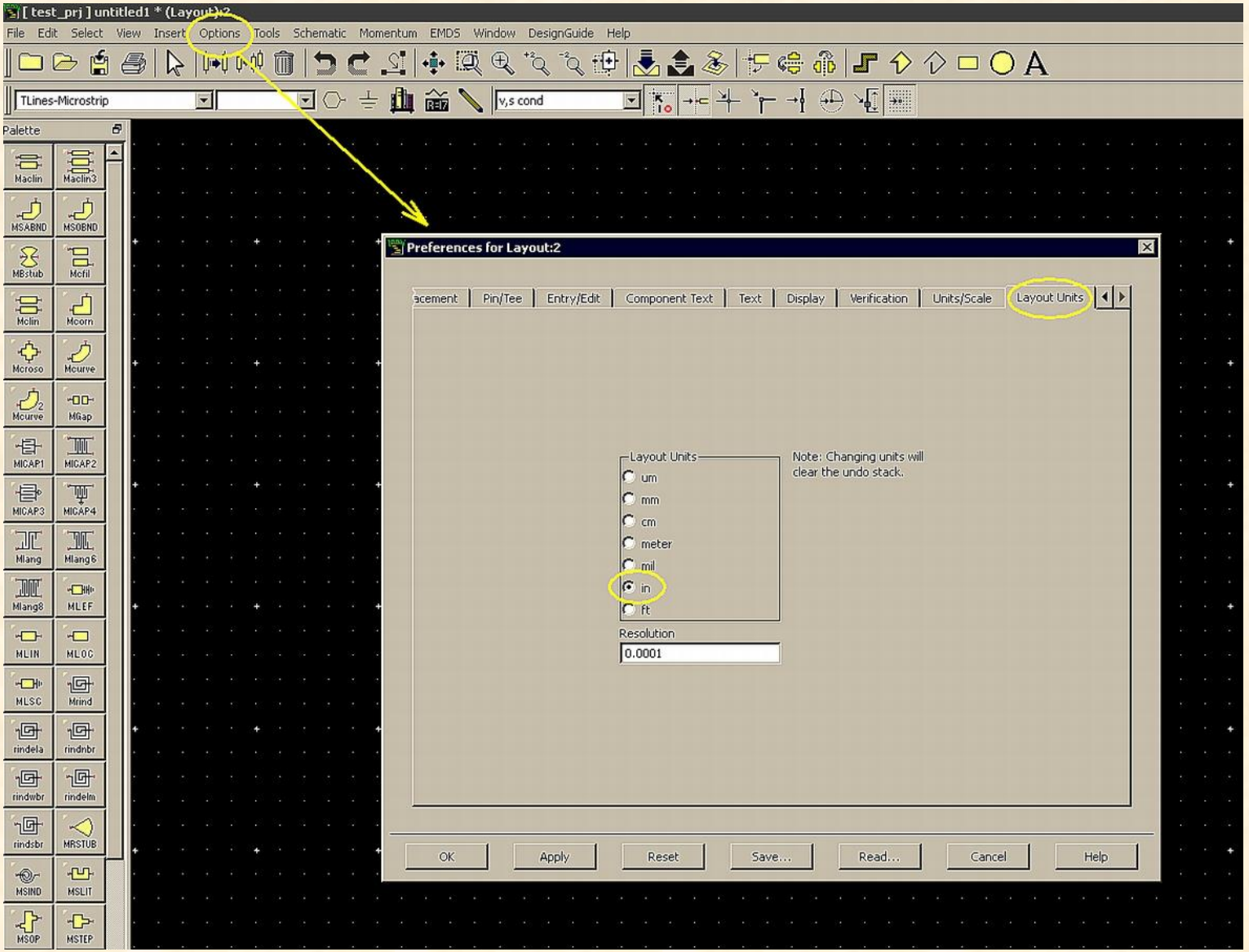
- Alignment donut and origin position
- Nothing has to be found on the left and under the x and y axis
- Everything has to be in the first quadrant.



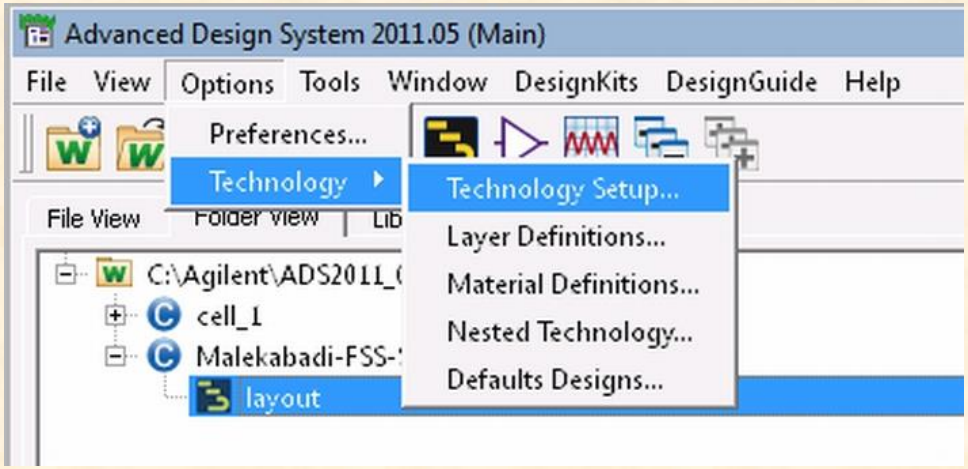
- How to make an alignment donut in ADS: make two, proper sized circles, align them and – Edit-Merge-Union minus Intersection

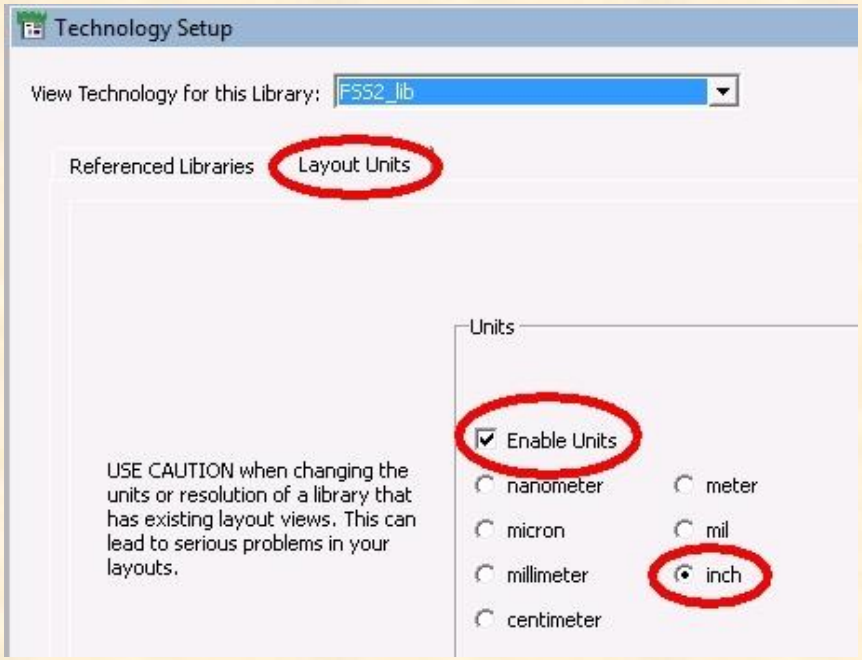


- Proper units in ADS2009: adjust the proper units **before** starting to work in the layout window (Options-Preferences-Layout Units-in)

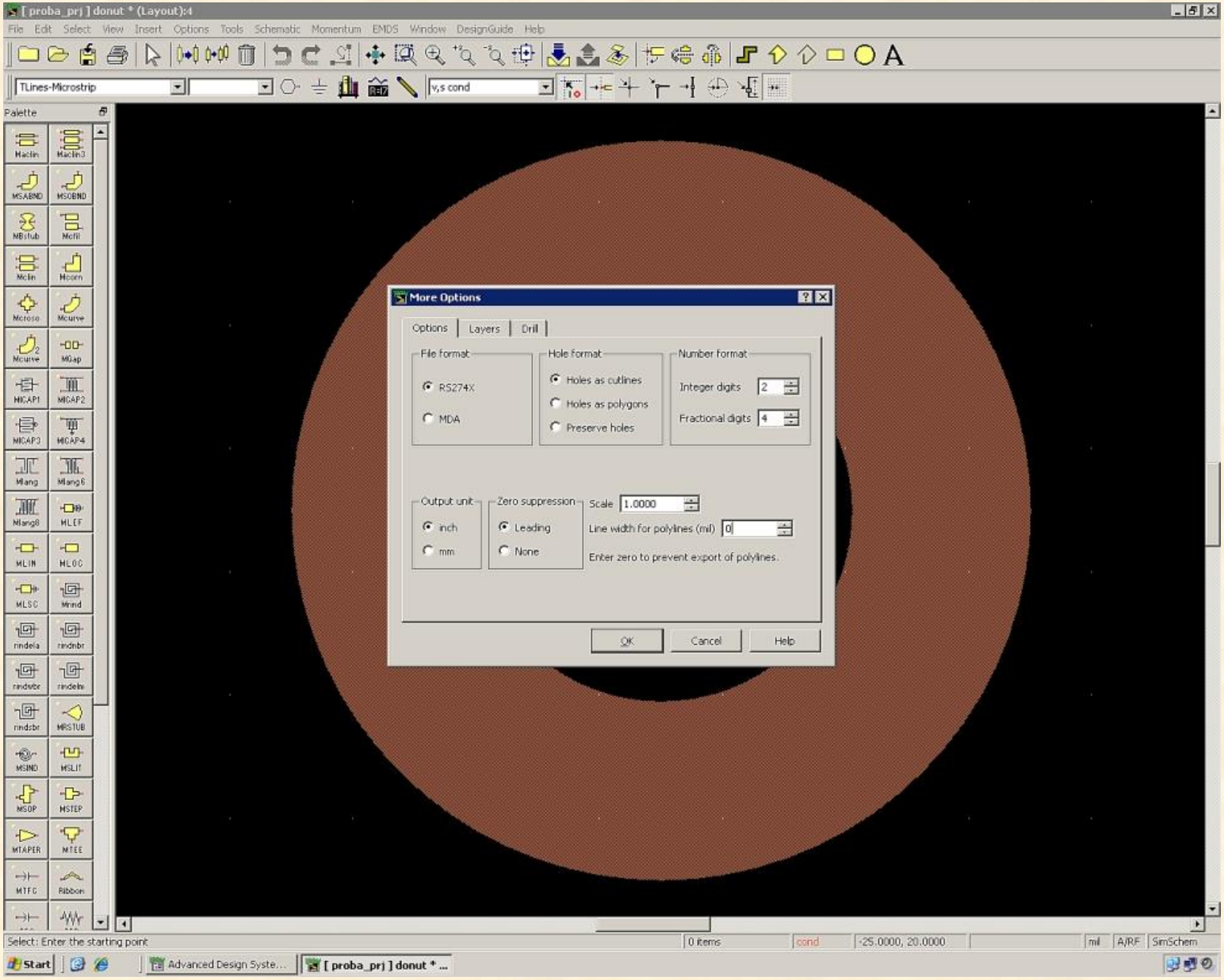


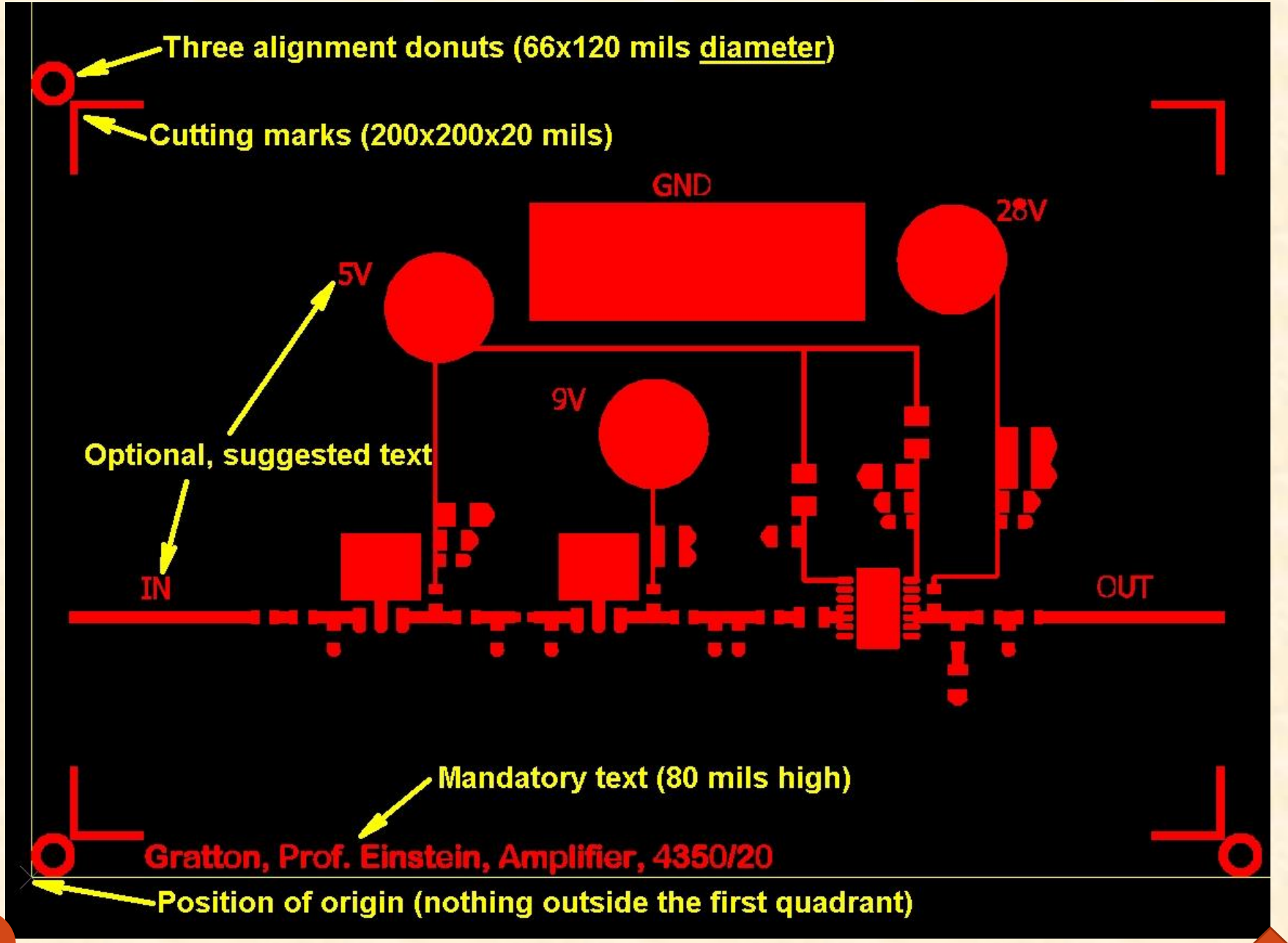
- Proper units in ADS2011: adjust the proper units **before** starting to work in the Main window (Options-Technology-Technology setup - Layout Units-in)





- Gerber file export




Three alignment donuts (66x120 mils diameter)
Cutting marks (200x200x20 mils)
GND
28V
5V
9V
Optional, suggested text
IN
OUT
Mandatory text (80 mils high)
Gratton, Prof. Einstein, Amplifier, 4350/20
Position of origin (nothing outside the first quadrant)

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, precision 2.4)?
- 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, is that layer named as the file?
- Are the holes circles, not segments in the .dxf?
- Did you include the substrate type and thickness?
- 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 use Viewmate to verify Gerber files: general circuit layout, correct position, correct size.
- Please use Autosketch to verify .dxf files: units, precision.

If you have any questions, please do not hesitate and ask.

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

Thank You!