Using “free” PCB design SW for Microwave PCB’s

Hamish Kellock

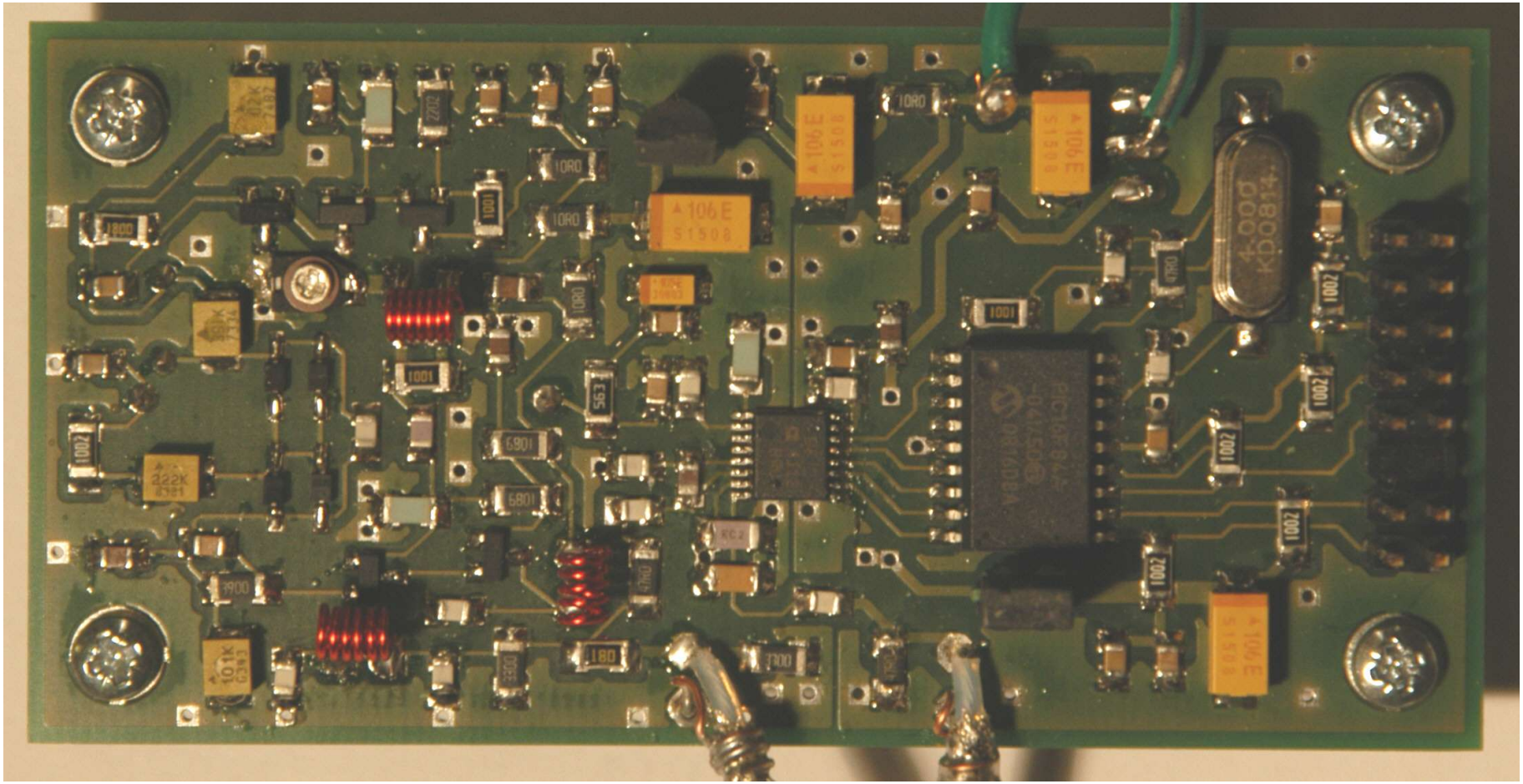
OH2GAQ

“Free” PCB Design SW

- There are several packages around which seem to be fully functional and “free”.
- Author has used the “Eagle” package for several years, so many of the details in the following comments apply to this package.
- But you can surely do the same thing with your favorite package as well.
- Eagle is particularly suited to the European environment, as several PCB manufacturers accept Eagle files directly.

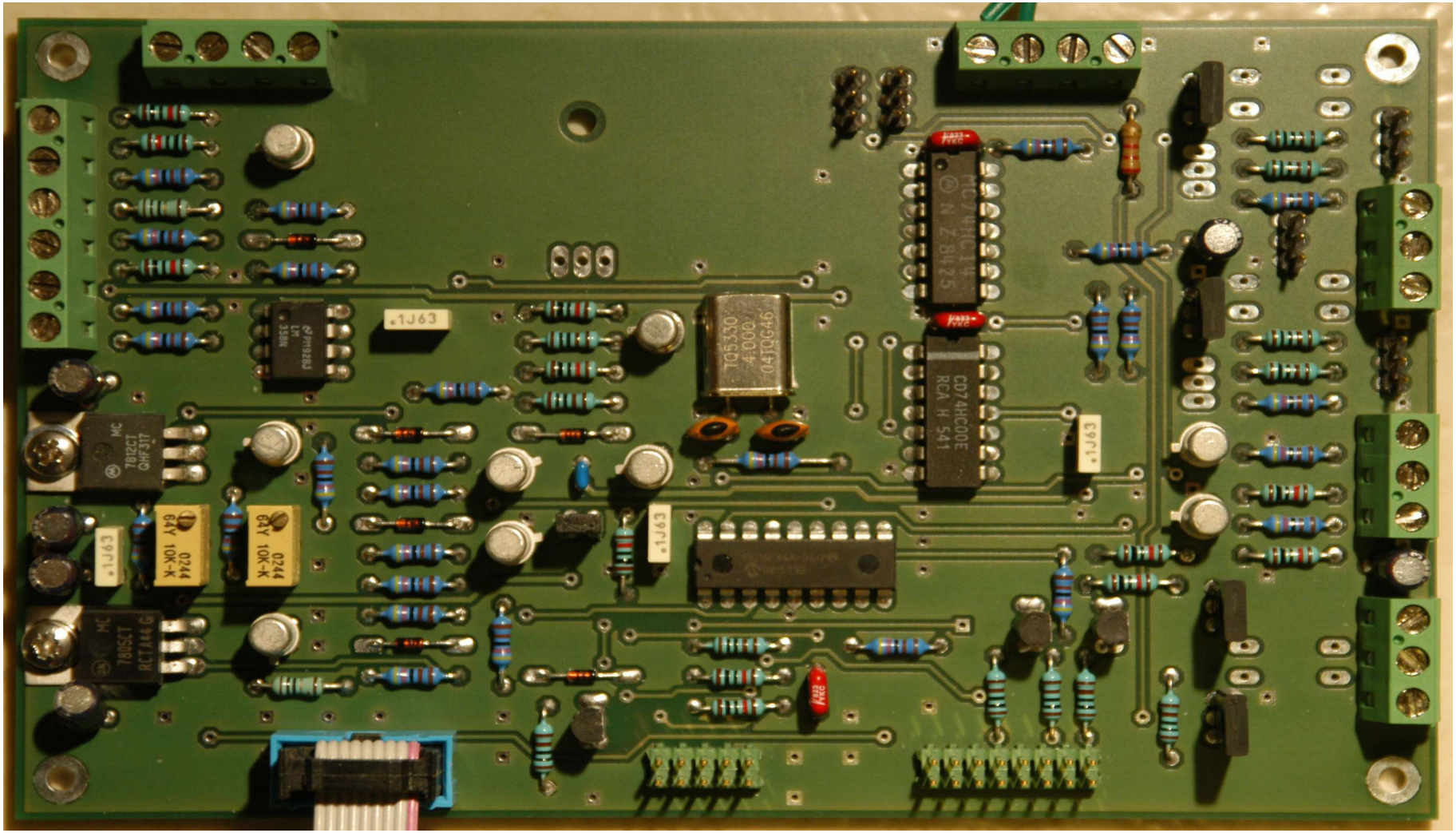
Conventional PCB Designs

Low Noise Xtal Osc. Locked by PLL



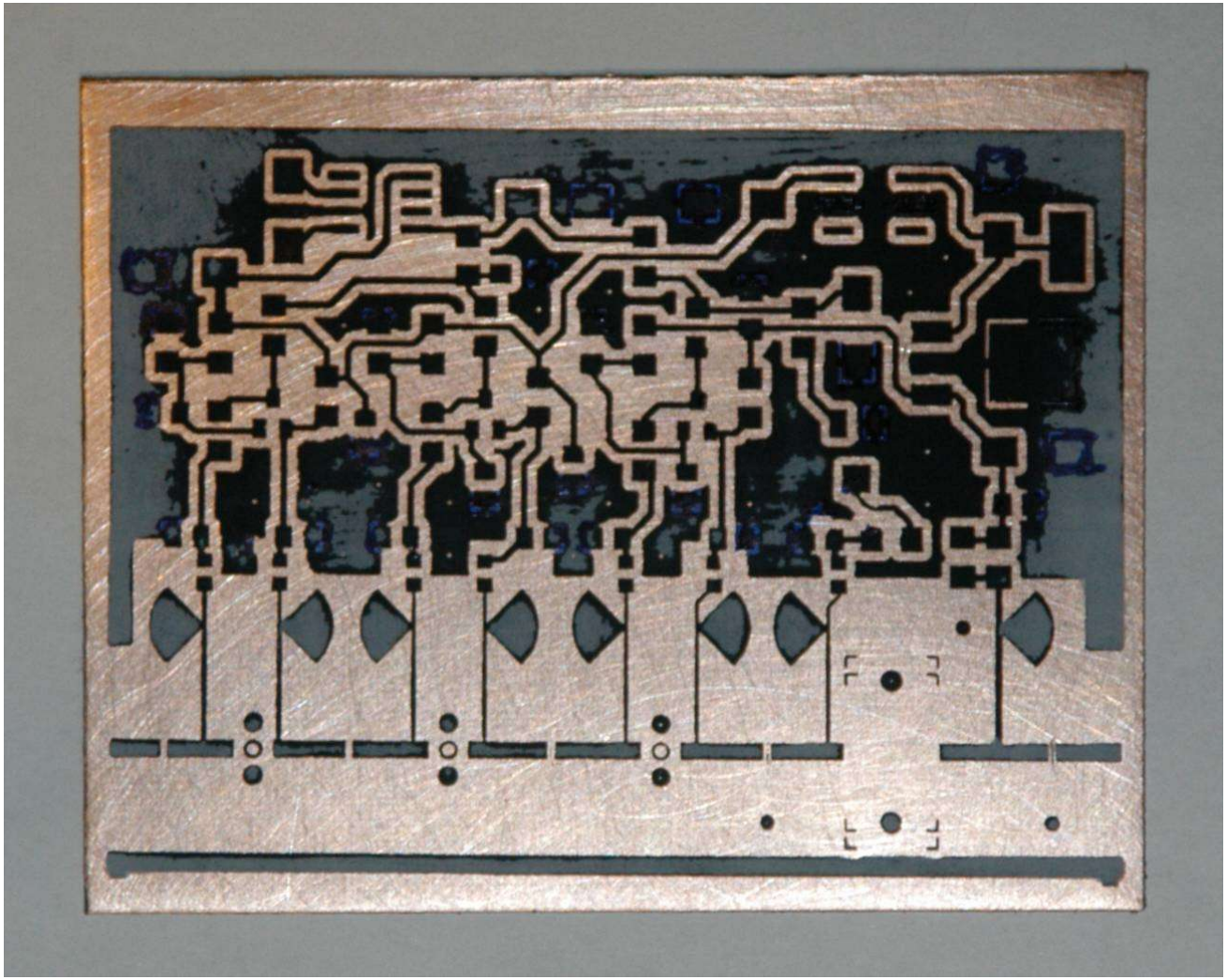
Conventional PCB Designs

Multi-Function Sequencer with Power Switch, Relay Drivers, and alarm processing



Microwave PCB Design

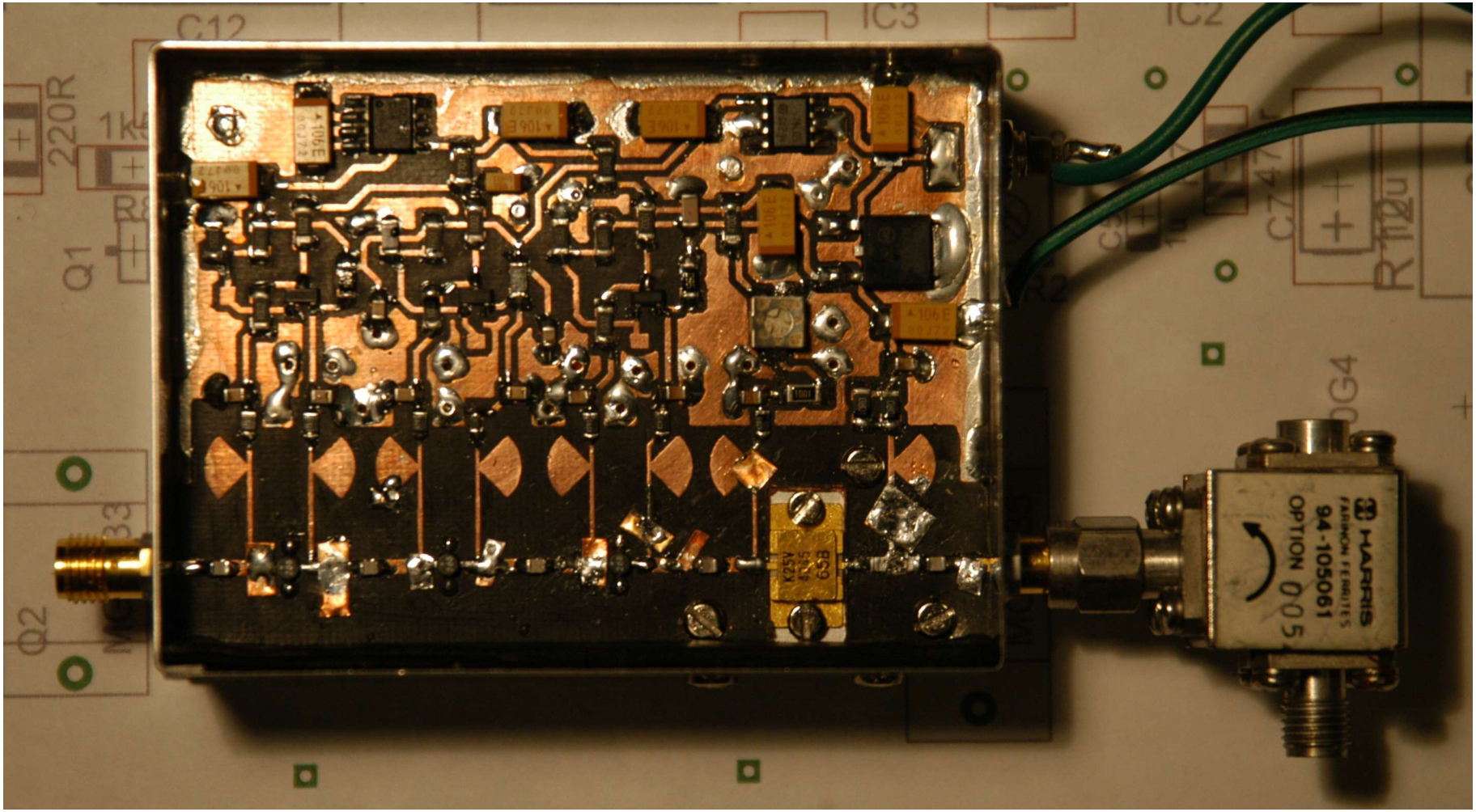
PCB on Rogers .015 inch (15 mil) 5880 material, using laser printer toner on photo paper as mask.



Finished 10GHz Amplifier

4 stages, Overall Gain 34 db, Pwr = +23 dbm

3 x ATF26884's, 1 x MGFK25



Approach adopted by OH2GAQ:

- Decide on the PCB board material to be used in microwave projects (Author uses Rogers 5880 15 mil material).

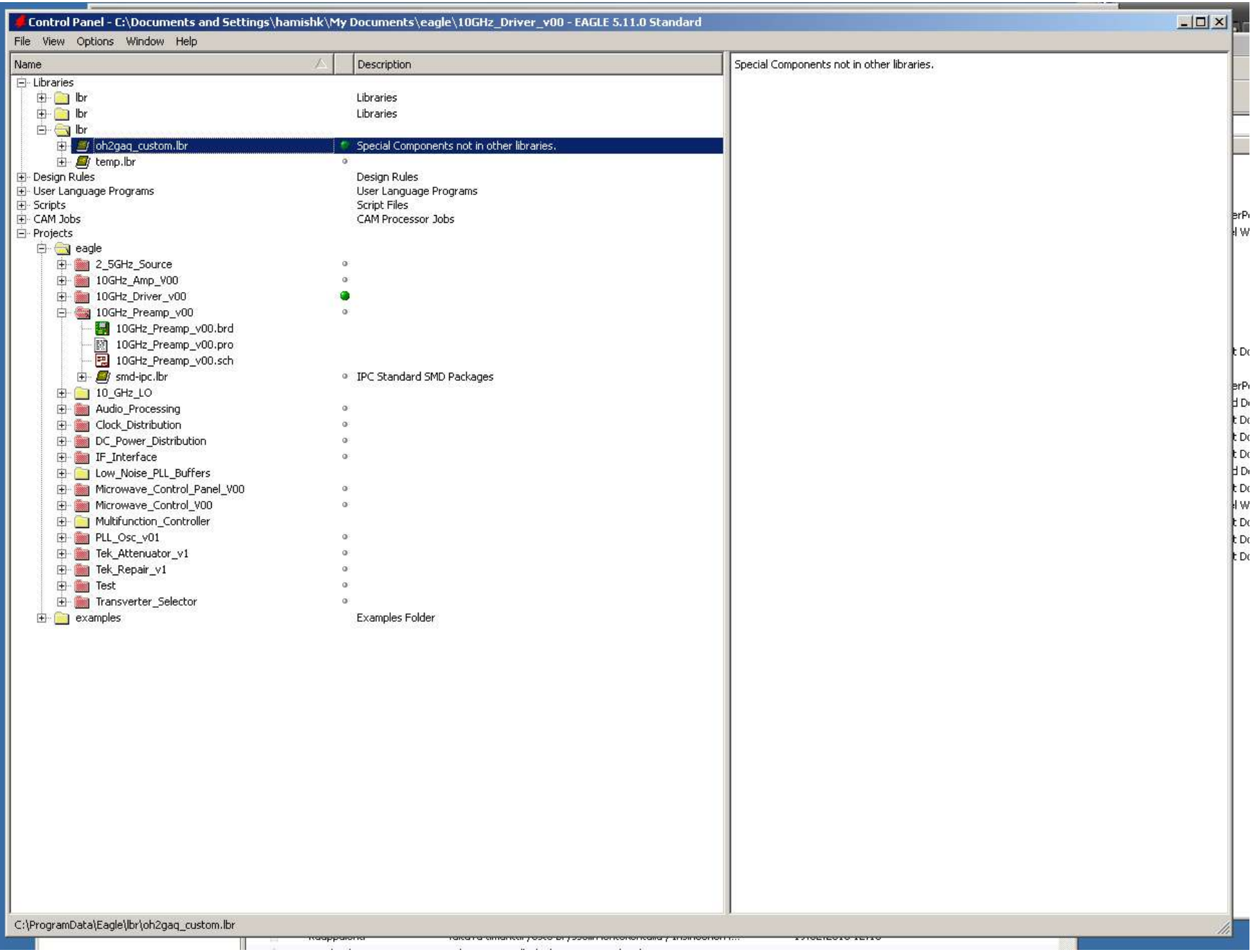
- Determine key parameters at the frequency of interest (Width of 50 ohm line, Wavelength in the PCB material, Losses if needed).

- Make symbols and component packages for the used semiconductors ensuring their connecting pads are the correct width for 50 ohms, also make single pads (50 ohms) and ¼ wave earthing stubs.

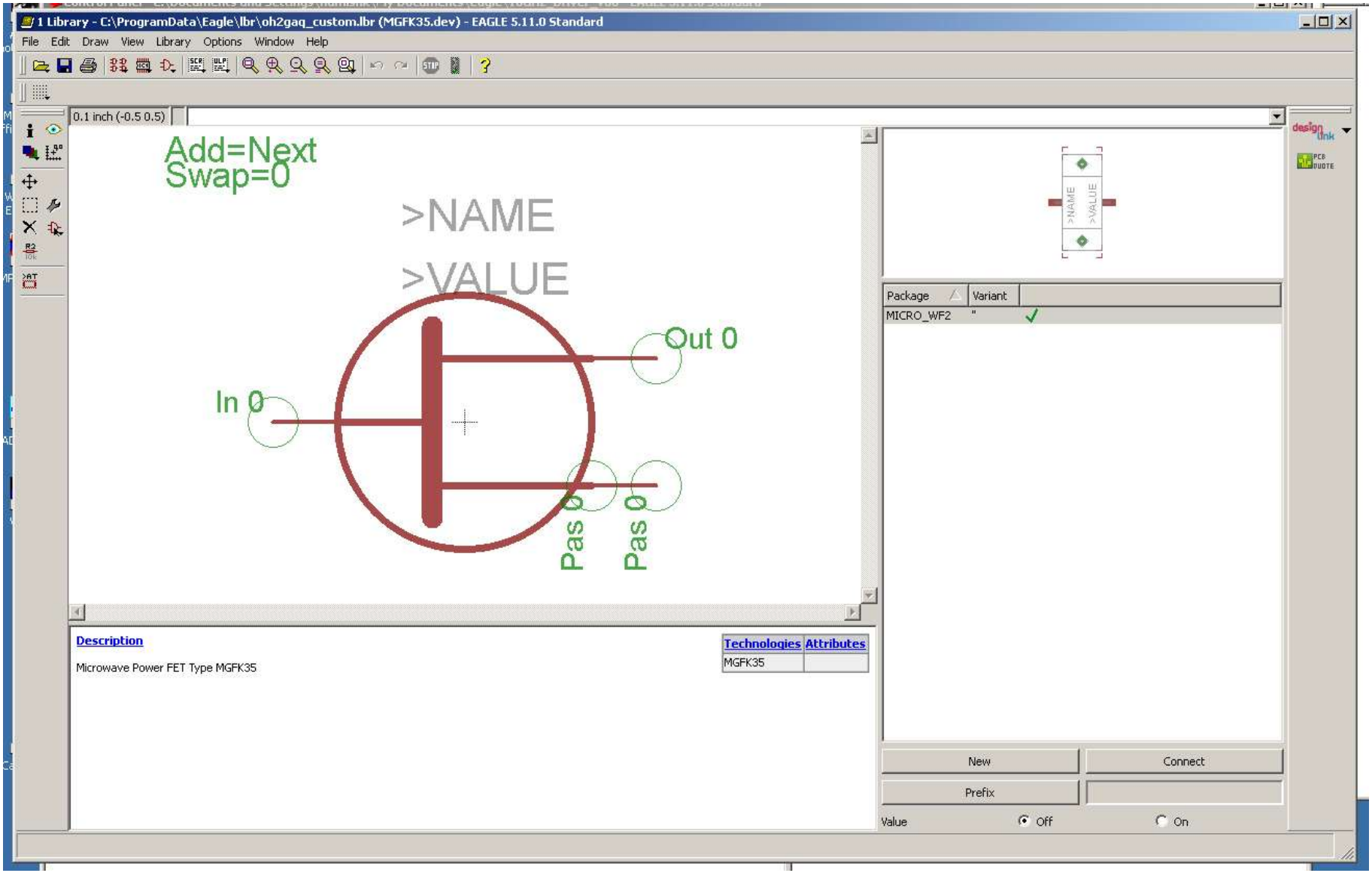
Approach adopted by OH2GAQ:

- Don’t bother with trying to do exact design of the matching networks, at least at 10 GHz and especially when using internally matched transistors designed for 14 GHz use. Use “snowflaking” after building the amplifier.

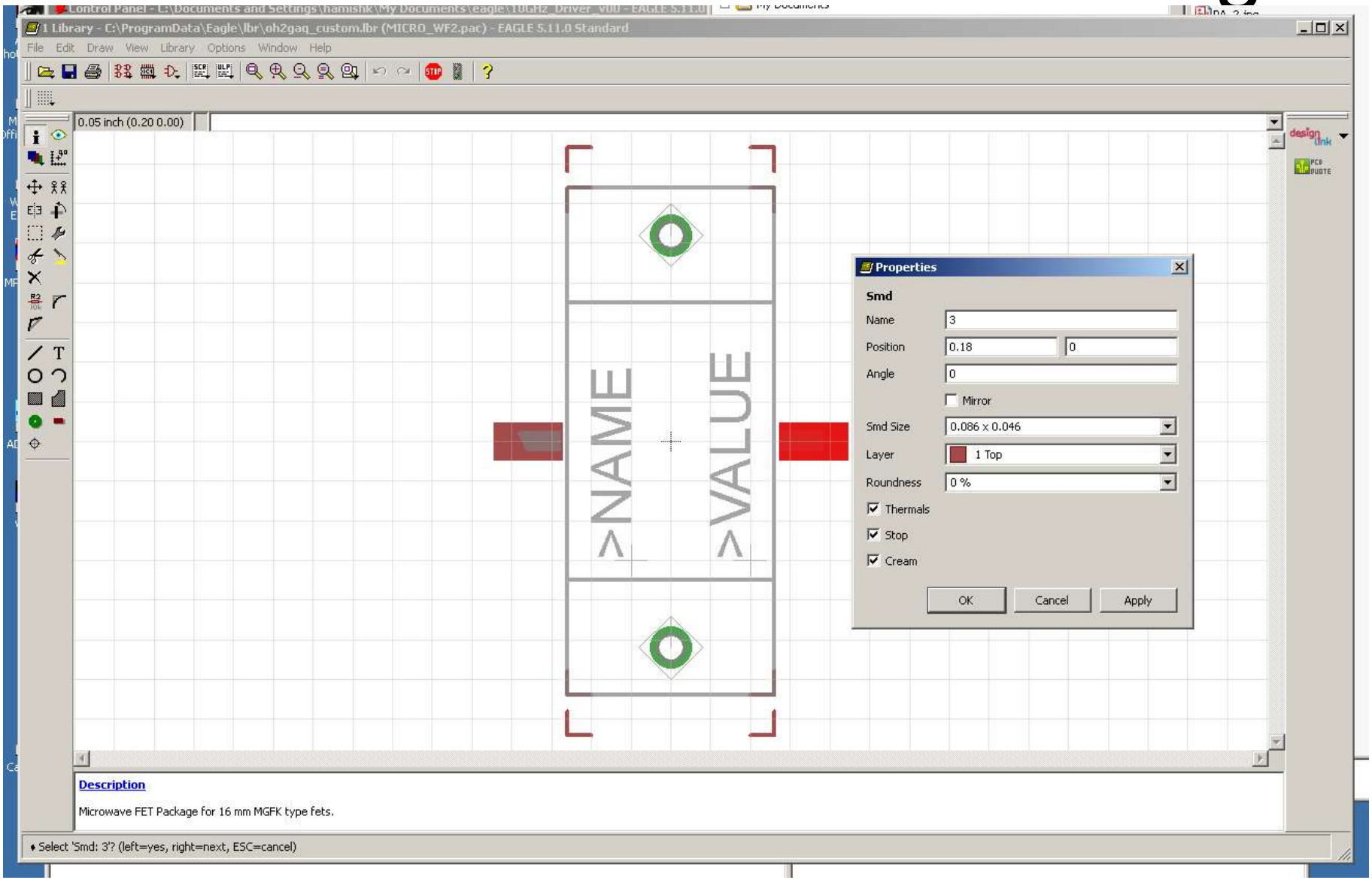
- Try and make reasonable length transmission line sections between the devices, to leave enough room for adding tuning stubs (“snowflakes”).


Control Panel - C:\Documents and Settings\hamishk\My Documents\eagle\10GHz_Driver_v00 - EAGLE 5.11.0 Standard
File View Options Window Help
Name
Description
Libraries
lbr
Libraries
lbr
Libraries
lbr
oh2gaq_custom.lbr
Special Components not in other libraries.
temp.lbr
Design Rules
Design Rules
User Language Programs
User Language Programs
Scripts
Script Files
CAM Jobs
CAM Processor Jobs
Projects
eagle
2_5GHz_Source
10GHz_Amp_V00
10GHz_Driver_v00
10GHz_Preamp_v00
10GHz_Preamp_v00.brd
10GHz_Preamp_v00.pro
10GHz_Preamp_v00.sch
smd-ipc.lbr
IPC Standard SMD Packages
10_GHz_LO
Audio_Processing
Clock_Distribution
DC_Power_Distribution
IF_Interface
Low_Noise_PLL_Buffers
Microwave_Control_Panel_V00
Microwave_Control_V00
Multifunction_Controller
PLL_Osc_v01
Tek_Attenuator_v1
Tek_Repair_v1
Test
Transverter_Selector
examples
Examples Folder
Special Components not in other libraries.
C:\ProgramData\Eagle\lbr\oh2gaq_custom.lbr

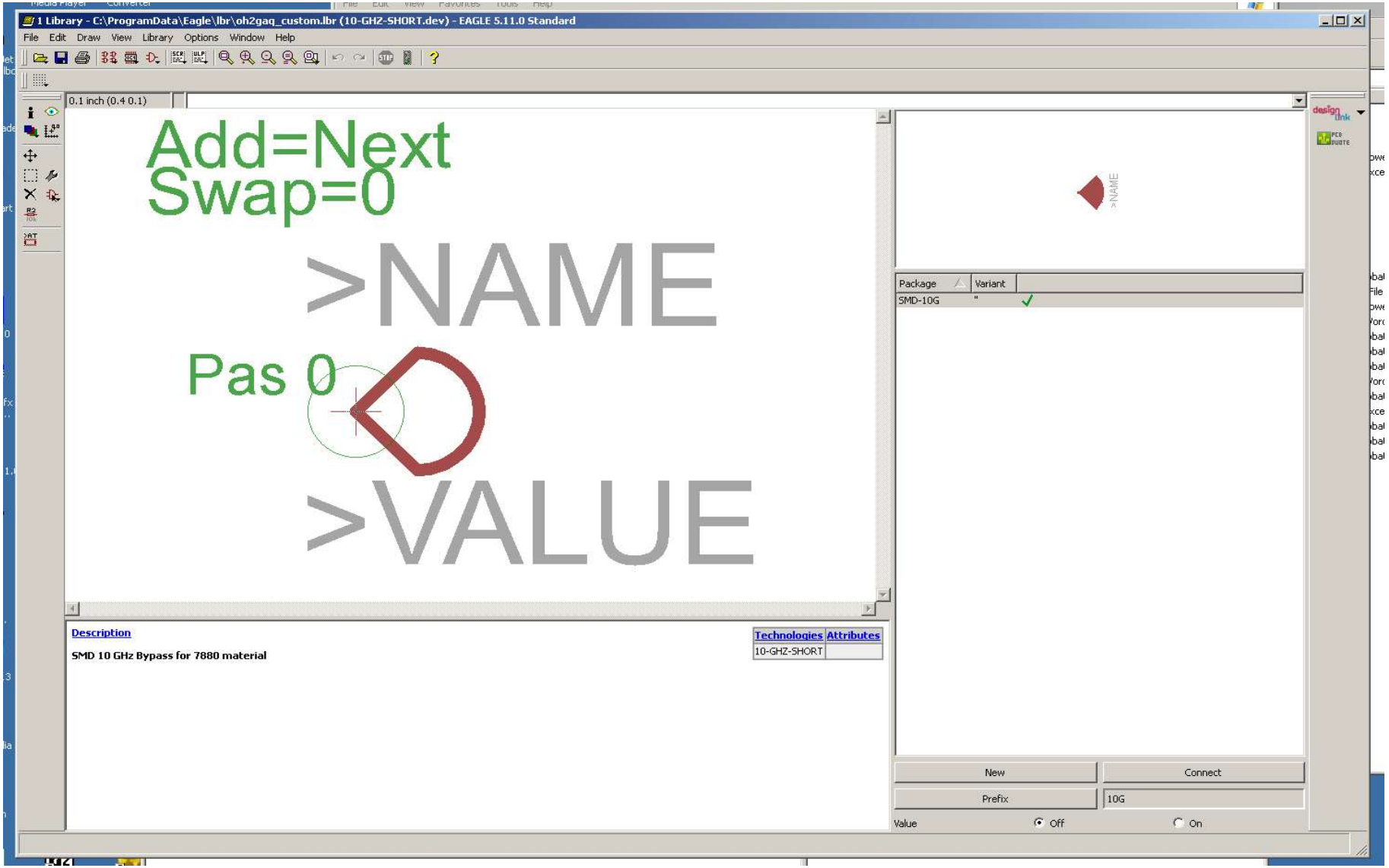
Microwave Power FET



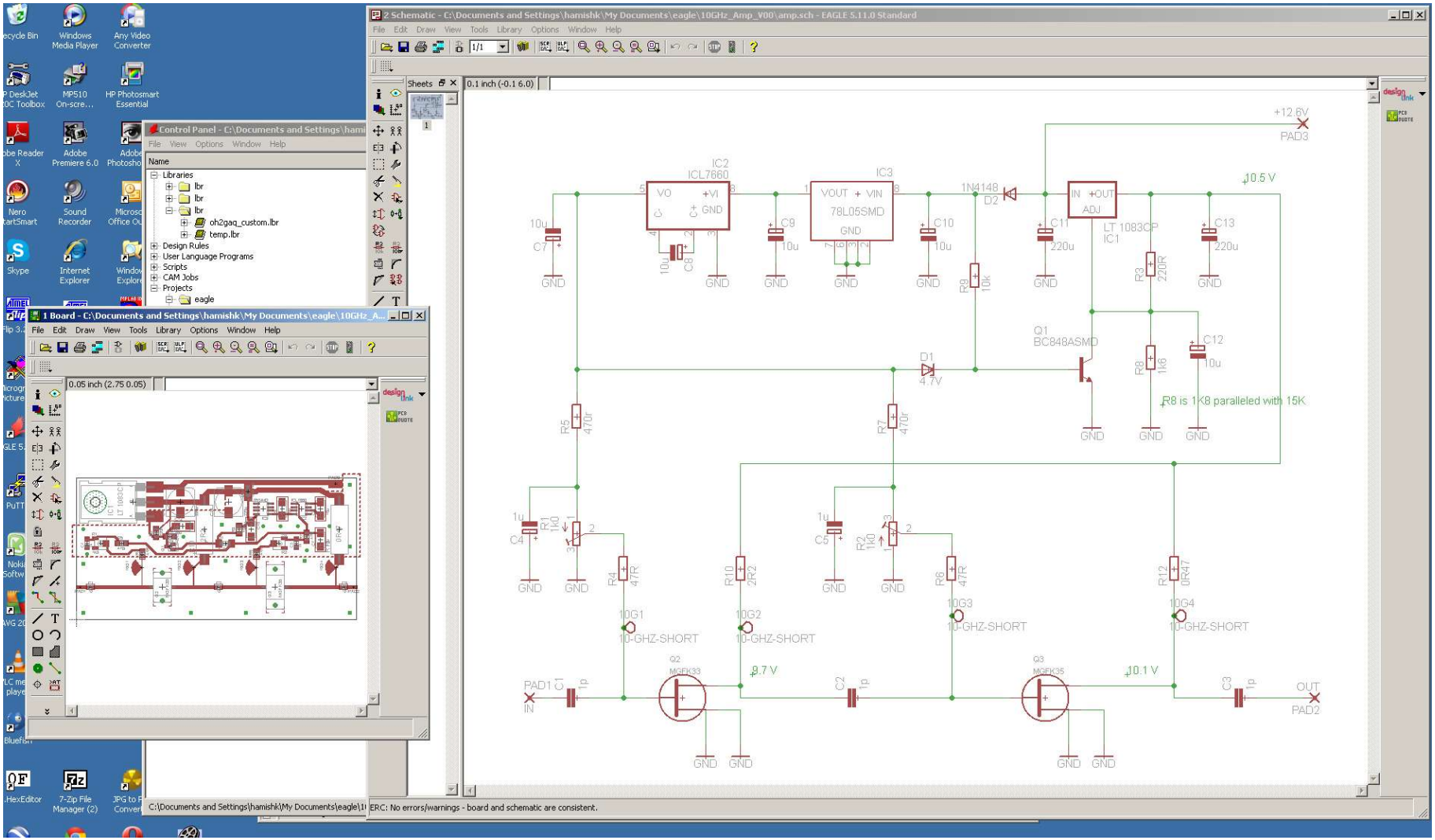
Microwave Power FET Package



¼ wave “short to ground”

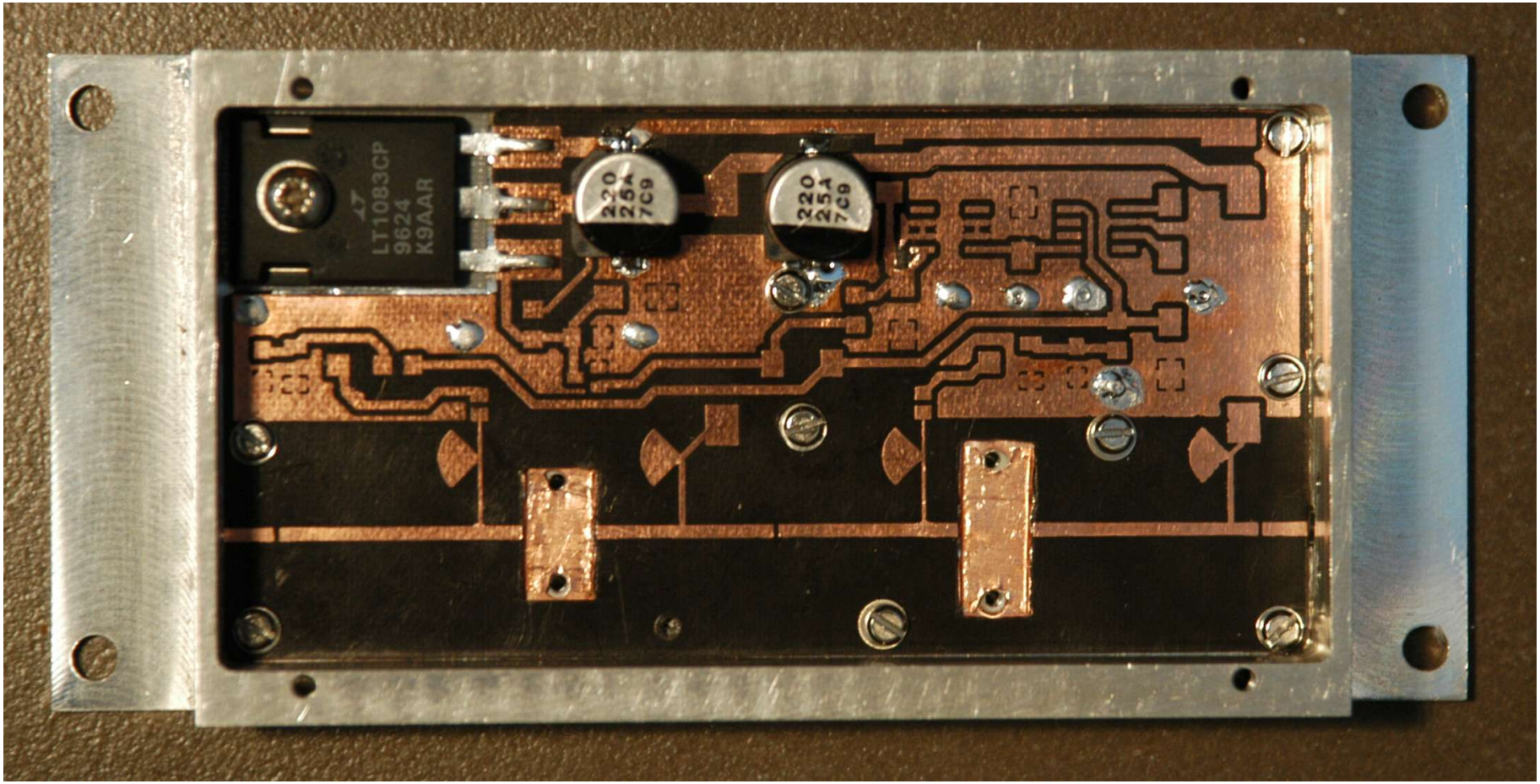


Schematic, Layout and General windows in Eagle



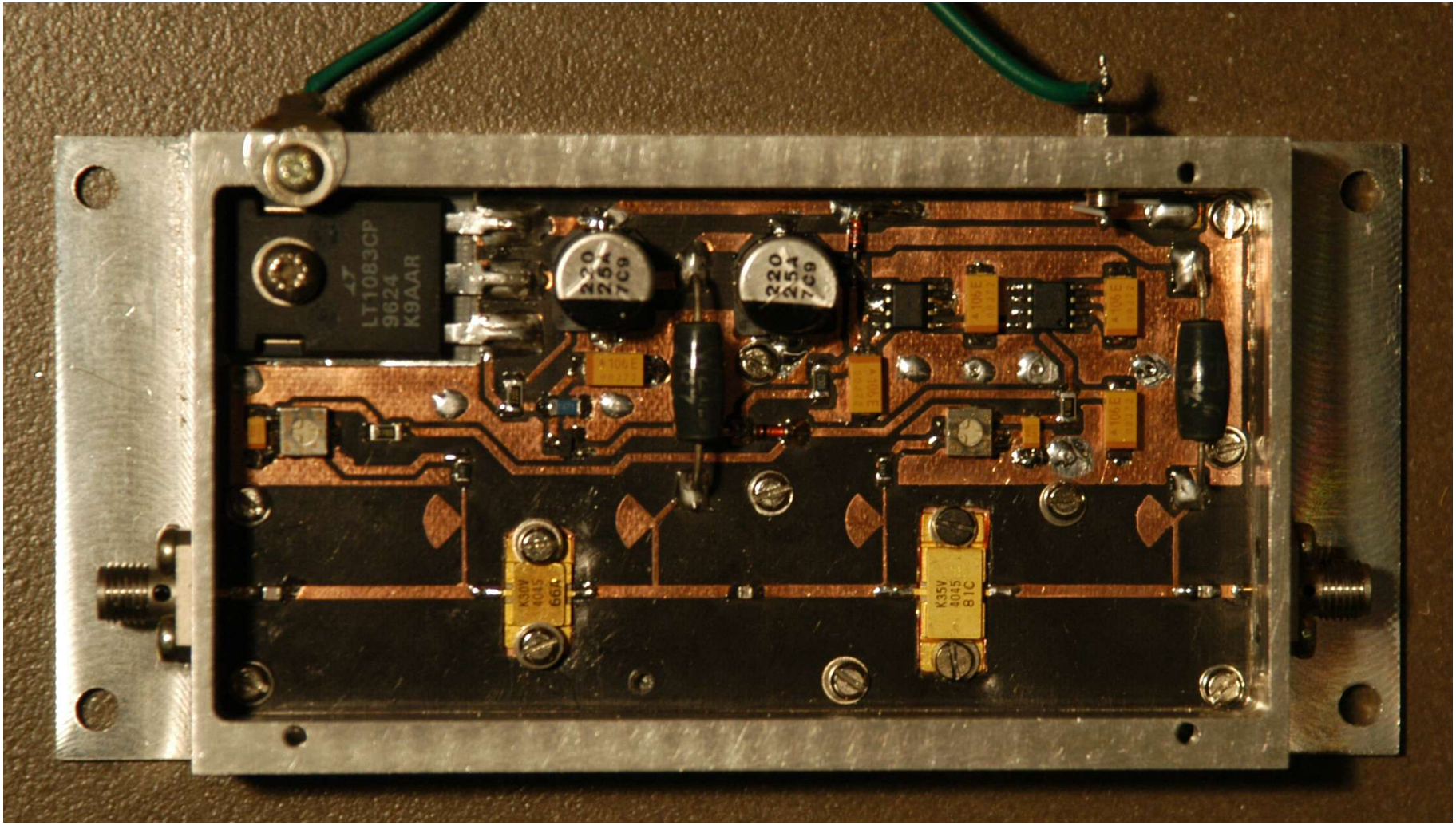
10 GHz Power Amplifier

before adding most components



10 GHz Power Amplifier

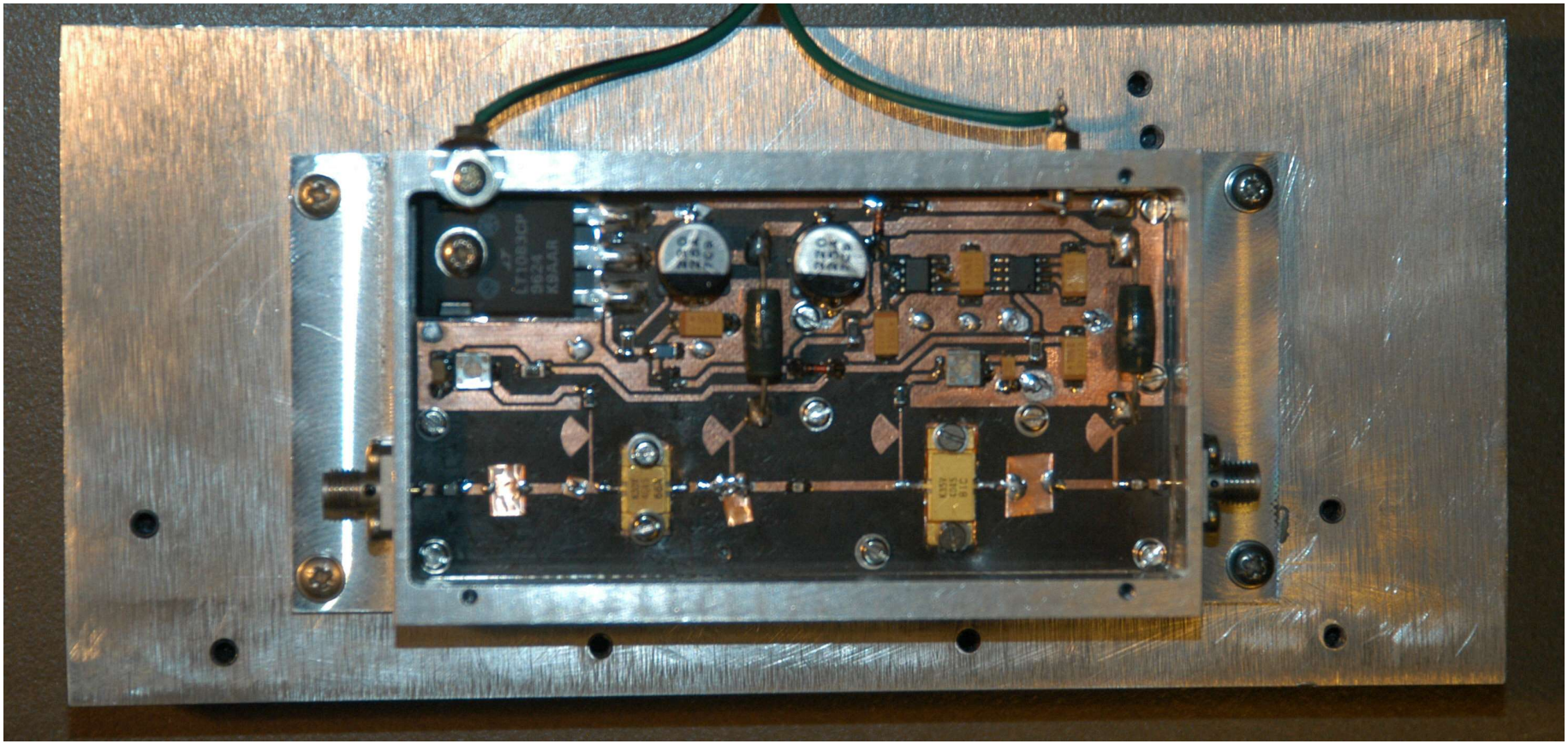
with all components in place, prior to tune up



10 GHz Power Amplifier

After tune-up.

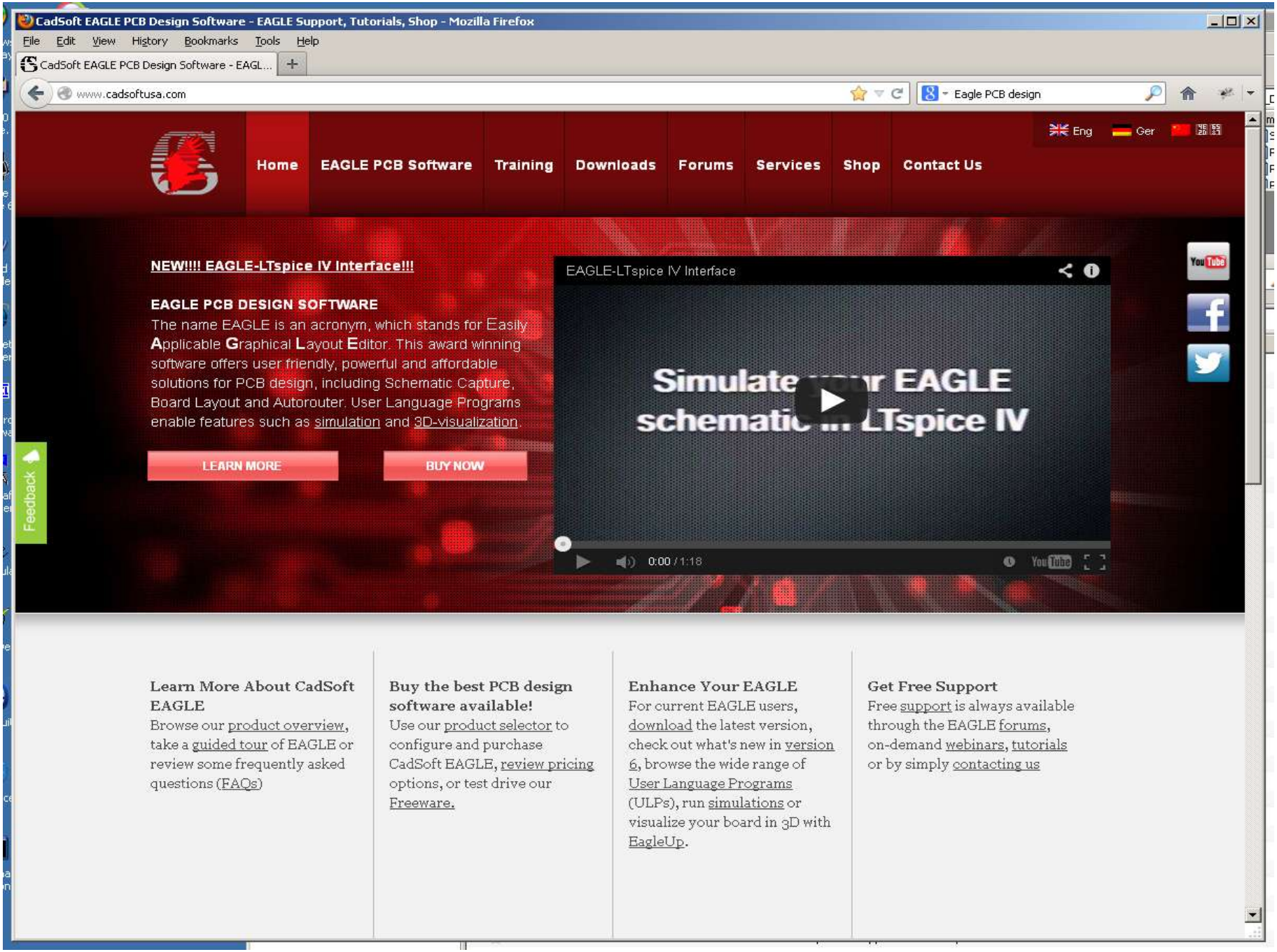
Overall gain +15.1 db, Pwr = +35.8 dbm



Eagle Licenses

- Freeware (100 mm x 80 mm, 2 layers), non-commercial use, 0€.

- Hobbyist (100mm x 160 mm, 6 layers), non-commercial use, 140€.


CadSoft EAGLE PCB Design Software - EAGLE Support, Tutorials, Shop - Mozilla Firefox
File Edit View History Bookmarks Tools Help
CadSoft EAGLE PCB Design Software - EAGL...
www.cadsoftusa.com
Eagle PCB design
Eng
Ger
Home
EAGLE PCB Software
Training
Downloads
Forums
Services
Shop
Contact Us
NEW!!!! EAGLE-LTspice IV Interface!!!
EAGLE PCB DESIGN SOFTWARE
The name EAGLE is an acronym, which stands for Easily Applicable Graphical Layout Editor. This award winning software offers user friendly, powerful and affordable solutions for PCB design, including Schematic Capture, Board Layout and Autorouter. User Language Programs enable features such as simulation and 3D-visualization.
LEARN MORE
BUY NOW
EAGLE-LTspice IV Interface
Simulate your EAGLE schematic in LTspice IV
0:00 / 1:18
Feedback
Learn More About CadSoft EAGLE
Browse our product overview, take a guided tour of EAGLE or review some frequently asked questions (FAQs)
Buy the best PCB design software available!
Use our product selector to configure and purchase CadSoft EAGLE, review pricing options, or test drive our Freeware.
Enhance Your EAGLE
For current EAGLE users, download the latest version, check out what's new in version 6, browse the wide range of User Language Programs (ULPs), run simulations or visualize your board in 3D with EagleUp.
Get Free Support
Free support is always available through the EAGLE forums, on-demand webinars, tutorials or by simply contacting us

Free EAGLE PCB Design Software, Layout Software - CadSoft Freeware - Mozilla Firefox

File Edit View History Bookmarks Tools Help

Free EAGLE PCB Design Software, Layout S...

www.cadsoftusa.com/download-eagle/freeware/?language=en

Eagle PCB design

Freeware

The EAGLE Light Edition can be used for **free**!

How do I get it?

Follow these steps to install the free EAGLE Light Edition on your system:

- Go to the Download area and get the file that contains the EAGLE program installation data for your operating system (Linux, Windows or Mac).
- Install the file you have downloaded onto your system.
- When you first start EAGLE, you will be asked whether you have a personalized license disk, or whether you want to run EAGLE as Freeware. To use the Freeware license select the "Run as freeware" button.

Limitations

The following limitations apply to the EAGLE Light Edition in general:

- The useable *board area* is limited to 100 x 80 mm (4 x 3.2 inches).
- Only *two signal layers* can be used (Top and Bottom).
- The schematic editor can only create *one sheet*.

Apart from these three limitations the EAGLE Light Edition can do anything the Professional Edition can do. You can even load, view and print drawings that exceed these limits!

The Freeware version of EAGLE Light adds these limitations:

- *Support* is only available via *email* or through our *forum* (no fax or phone support).
- Use is limited to *non-profit* applications or evaluation purposes.

What does "non-profit" mean?

Making the PCB's (1)

- **Here is one "recipe" for actually making the PCB's**

- 1. Cut a suitable piece of PCB material, at least 5 mm larger than the finished PCB. Clean your PCB material on the side you are going to use with a scouring pad (for example "Scotchbrite" or similar kitchen cleaning pads), using plenty of water and detergent.
- 2. Finish up the cleaning by using a colourless kitchen paper soaked in a solvent such as acetone or methy ethyl ketone (MEK) etc. Do not touch the surface of the PCB after this final cleaning.
- 3. Print up your artwork 1:1 using a **laser printer** and a **glossy photographic paper**, that is not plastic based. Presently the author has found that Canon Glossy Photo Paper "Everyday Use" type GP-501 seems to work pretty well. The paper is intended for use with Ink Jet printers. Print mirror image if required. Use the darkest toner setting on the laser printer, and set it to print on transparency material, if it has this setting. This is not the correct way to use a laser printer and using this kind of paper can void the warranty.
- 4. Place a sheet of clean paper on sheet of 20 to 30 mm thick flat and stable wood. Place the cleaned PCB material with the clean face upwards in the middle of this combination. After cutting the artwork to about 5mm larger than the active artwork area, place it face down on the PCB material. Place two (2) sheets of clean paper on the top of the stack of materials. Make sure everything is still aligned – the artwork is still centered over the PCB material.
- 5. Use an ordinary domestic clothes iron, make sure it is drained of water if it is a steam iron. Set the temperature to a high setting, but not the highest. You should aim for a temperature of between 160 Celsius to 180 Celcius. If too high, it is difficult to impossible to remove the photo paper without destroying the image in places. If too low, the toner from the laser printer will not transfer correctly. Let the iron reach a stable temperature, then lower it onto the stack of paper and PCB. If the PCB is smaller than the iron, just apply a small pressure and wait about 30 to 45 seconds. If it is larger than the iron, lift the iron after about 30 seconds and place it down again over a different area of the PCB. Cover all the PCB area this way, ensuring that all areas have had at least 30 seconds to 45 seconds of heating.

Making the PCB's (2)

- 6. Remove the PCB with attached photo paper and let it cool. Careful, it will be hot !
- 7. Fill the washbasin or sink with clean hot water. Place the PCB with attached photo paper in the sink, and go and do something else for about 15 minutes.
- 8. Peel off the backing paper using your fingers to "roll" it off the underlying paper. You can then rub off the remainder of the backing paper until all that is left is the toner-covered board, plus probably a whitish tinted layer in some places.
- 9. Gently "scrub" the board with a medium texture toothbrush, until the whitish deposit is removed from all copper areas where etching is required. See Slide 5 of this series as an example.
- 10. If there are areas where the tracks separate, either start over again, or repair the area with a special PCB resist pen. Normally if the cleaning step 2 succeeds, all will go well.
- 11. As we are talking about RF boards, normally the other side of the board will be full copper. Cover it with either paint or tape, duct tape works well here. Make sure there are no leak paths into the interior of the PCB surface.
- 12. Etch the board in a warmed etchant bath. Works best if the board is upside-down, so the area where the material has to be etched away is facing downwards. The author uses Ferric Chloride etchant.
- 13. After etching, wash thoroughly with running water and detergent. Remove the tape from the backside of the board.
- 14. Before removing the resist (laser toner plus any repair ink), ensure that all holes are marked on the copper either by etching or if not etched, use a sharp scribing tool to mark the holes. Don't use an automatic centre punch, especially if you are using soft microwave substrates. Mark the board outline and any cut-outs for transistors also with a scribing tool.
- 15. Remove the resist with acetone or MEK. Drill the holes, cut out the transistor holes, and cut the board to size.
- 16. Clean both sides of the board with a scouring pad, and mount it in it's case. Mount the components, without touching the board if possible with your fingers. Clean the finished assembly with cleaning alcohol (Note you may have to leave off some components like coils and trimmer resistors and capacitors at this stage). Spray with a "solderable" resist material, available from electronic component suppliers. Add any components excluded from the cleaning process.