

MCC125 - HARDWARE DESIGN

Vessen Vassilev

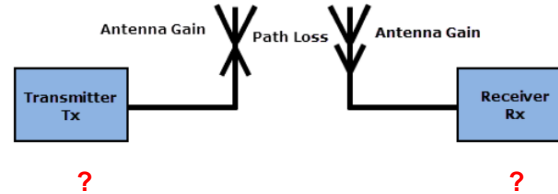
Offices: C621,



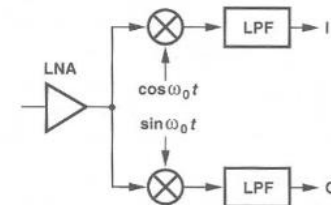
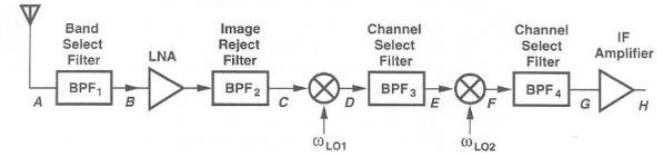
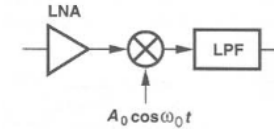
HW Design Steps

1. Transceiver topology selection
 - Type of mixers (IQ) or DSB
 - Choice of RF/LO at Tx/Rx, IF frequency,
 - Choice of RF/IF/LO amplifiers – you need to provide certain gain and output power
 - Tx/Rx block diagram
2. Components selection
 - all active and passive components and their packages
3. PCB design
4. RF Measurements
 - Gain, output power, saturation levels etc. of Tx/Rx are measured
5. Test your wireless link

Step 2: Transceiver topology selection



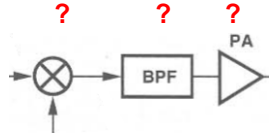
- Several Topologies exist. All have their pros and cons.
- Check **"RF microelectronics"** by Behzad Razavi. **Chapter 5**. Available in Chalmers Library.
- Present your Tx/Rx topology designs and motivate your choice in the "Link Budget" assignment.



HW Design Steps

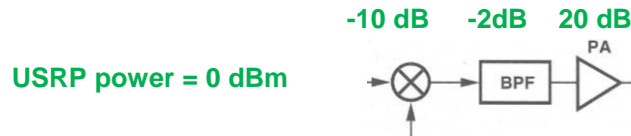
1. **Link budget to system specifications**
2. **Transceiver topology selection**
3. **Components selection**
4. PCB design
5. RF Measurements

Step 2: Components selection



- Replacing blocks with commercial products
- An inventory list of available products is uploaded in Canvas with links to datasheets
- Extra components can be ordered if needed
- Make sure the overall performance matches your link budget analysis:

Components		
Item	Description	Quantity
General amplifiers		
HMC311ST89E	Gain block amplifier, DC to 6 GHz	13
HMC789ST89E	MMIC amplifier, 0.7 to 2.8 GHz	10
Low noise amplifiers		
HMC374E	Low noise amplifier, 0.3 to 3.0 GHz	6
MGA-86563	Low noise amplifier, 0.5 to 6.0 GHz	10
Power amplifiers		
HMC414MS8GE	Power amplifier, 2.2 to 2.8 GHz	10
HMC455LP3E	0.5 W High IP3 amplifier, 1.7 to 2.5 GHz	6
MGA-31689	0.5 W High Gain Driver Amplifier, 1.5 to 3.0 GHz	10
HMC415LP3E	Power amplifier, 4.9 to 5.9 GHz	6
HMC406MS8GE	Power amplifier, 5 to 6 GHz	6
HMC407MS8GE	Power amplifier, 5 to 7 GHz	6
HMC408LP3E	1 W Power amplifier, 5.1 to 5.9 GHz	4



How to order components not available in the inventory





- Go to: <https://www.digikey.se/en>
- Find your component and chose the one you need, consider availability, minimum quantity and package. Same Amp is available in different packages!

Make sure it is available

Make sure you can order few pieces

Make sure you this is the package you consider in your PCB layout

This is the component you want: it is in stock and fewer pieces can be ordered

Compare Parts	Image	Digi-Key Part Number	Manufacturer Part Number	Manufacturer	Description	Quantity Available ?	Unit Price SEK	Minimum Quantity	Package / Case	Supplier Device Package
<input type="checkbox"/>		1127-3543-ND	HMC374SC70E	Analog Devices Inc.	IC RF AMP GP 300MHZ-3GHZ SC70-6	591 - Immediate	kr48.28000	1	6-TSSOP, SC-88, SOT-363	SC-70-6
<input type="checkbox"/>		1127-1828-2-ND	HMC374SC70ETR	Analog Devices Inc.	IC RF AMP GP 300MHZ-3GHZ SC70-6	0 Standard Lead Time 13 Weeks	kr37.01957	5 000	6-TSSOP, SC-88, SOT-363	SC-70-6
<input type="checkbox"/>		1127-2948-ND	HMC374E	Analog Devices Inc.	IC RF AMP GP 300MHZ-3GHZ SOT26	0 Standard Lead Time 13 Weeks	kr57.18000	1	SOT-23-6	SOT-23-6
<input type="checkbox"/>		505-HMC374-ND	HMC374	Analog Devices Inc.	IC RF AMP GP 300MHZ-3GHZ SOT26	0 Standard Lead Time 13 Weeks	kr84.24000	1	SOT-23-6	SOT-23-6

How to order components not available in the inventory

- Use the function "Cart Share"
- Copy the link in the pop-up window
- Enclose this link instead of BOM
- Upload the link to "Canvas" together with your layouts.
- **OBS! You need this step only if you need special components not available in the course inventory!!**

The screenshot shows the Digi-Key Electronics website interface. At the top, there's a navigation bar with the Digi-Key logo, a search bar, and a user account link. Below the navigation bar, there's a section for "MCC125 Gr1" with a shipping alert: "Shipping Alert - Due to extremely high order volumes your order may take 1-3 business days to ship." A red 'X' is drawn over the "Checkout" button in the top right corner. The main content area shows a table with product details for "1127-3543-ND HMC3745C70E Analog Devices Inc. IC RF AMP GP 300MHZ-3GHZ SC70-6". The table has columns for Quantity, Availability, Unit Price, and Extended Price. The quantity is set to 2, and the extended price is 96,56 kr. A red 'X' is also drawn over the "Add to Cart" button. On the right side, there's a "Summary" section with fields for Web ID, Access ID, and Subtotal. Below the summary, there's a "Cart Tools" section with links for "Add Carts", "Add BOM", "Add Quote", "New Cart", "Schedule Shipments", "Delay Entire Order", and "Resume Cart". At the bottom right, there's a "Subtotal: 96,56 kr" and a "Checkout" button. A red 'X' is drawn over the "Checkout" button.

Cart Share

This link will create a new cart containing the same products, quantities and customer references for anyone who opens it. No other attributes of this cart will be shared. Copy the link and paste it anywhere you would like it shared.

<https://www.digikey.se/short/c4wz2v>

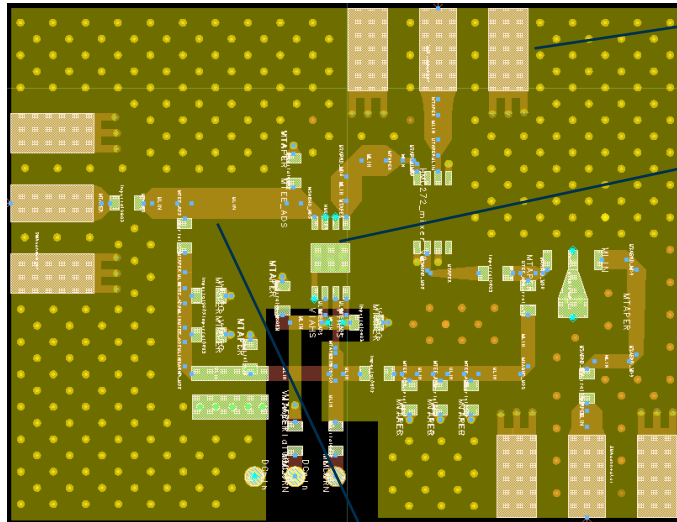
HW Design Steps

1. Link budget to system specifications
2. Transceiver topology selection
3. Components selection
4. PCB design
5. RF Measurements

Step 3: PCB Design

Step 3: PCB Design

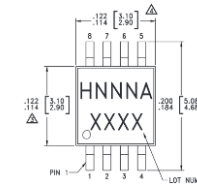
- How to design the PCB?
 - Layout footprints of components and connect them with transmission lines
 - RF lines carrying the signal should have 50 Ohms impedance (Use ADS LineCalc tool to calculate the width of 50-Ohm lines)
 - DC lines width should be chosen so that it handles current flowing through it.



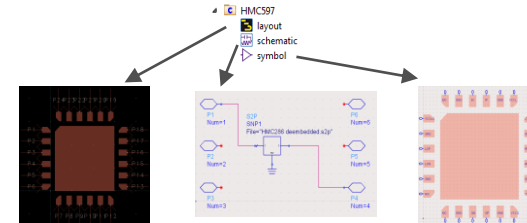
Footprint of
SMA connector



Footprint of
selected mixer
component
(From datasheet)



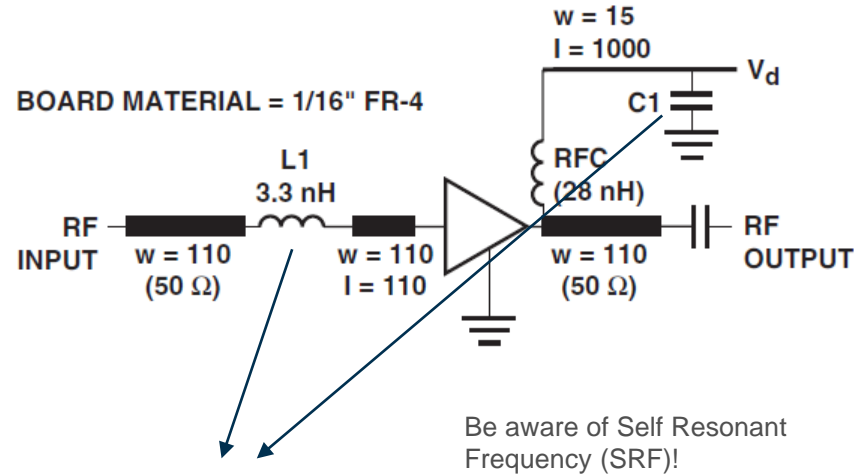
Most components you
need to draw yourself



50-Ohm Transmission
line

Step 3: PCB Design

- Bias network (Check datasheets of components for proper DC biasing)



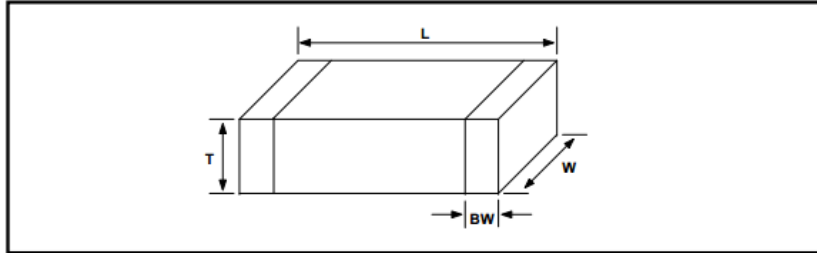
Step 3: PCB Design – component package size

Inch Case Code	Inch (Metric) Case Code	Metric Dimensions	Inch Dimensions
008004	008004 (0201 Metric)	0.25 x 0.125	0.010 x 0.005
009005	009005 (03015 Metric)	0.30 x 0.15	0.012 x 0.006
01005	01005 (0402 Metric)	0.40 x 0.20	0.016 x 0.008
0201	0201 (0603 Metric)	0.60 x 0.30	0.020 x 0.010
0402	0402 (1005 Metric)	1.00 x 0.50	0.040 x 0.020
0603	0603 (1608 Metric)	1.60 x 0.80	0.060 x 0.030
0805	0805 (2012 Metric)	2.00 x 1.25	0.080 x 0.050
1008	1008 (2520 Metric)	2.50 x 2.00	0.100 x 0.080
1206	1206 (3216 Metric)	3.20 x 1.60	0.125 x 0.060
1210	1210 (3225 Metric)	3.20 x 2.50	0.125 x 0.100
1806	1806 (4516 Metric)	4.50 x 1.60	0.180 x 0.060
1812	1812 (4532 Metric)	4.50 x 3.20	0.180 x 0.125
1825	1825 (4564 Metric)	4.50 x 6.40	0.180 x 0.250
2010	2010 (5025 Metric)	5.00 x 2.50	0.200 x 0.100
2512	2512 (6332 Metric)	6.30 x 3.20	0.250 x 0.125
2920	2920 (7451 Metric)	7.40 x 5.10	0.290 x 0.200

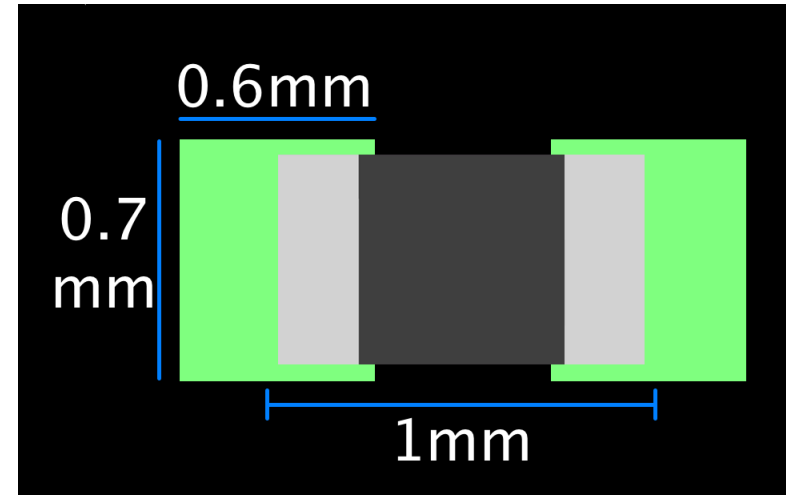
Step 3: PCB Design – component foot print

0402	0402 (1005 Metric)	1.00 x 0.50	0.040 x 0.020
------	--------------------	-------------	---------------

APPEARANCE AND DIMENSION



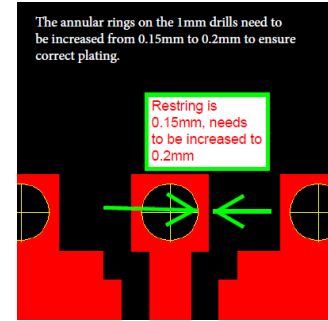
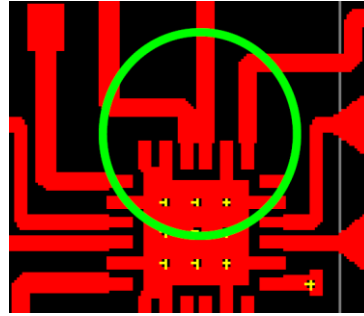
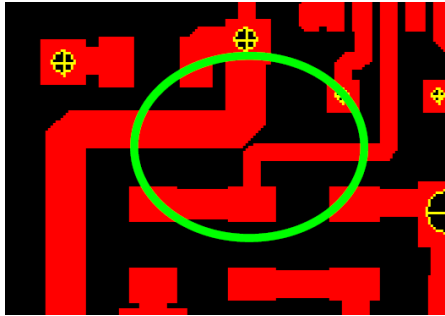
CODE	EIA CODE	DIMENSION (mm)			
		L	W	T (MAX)	BW
03	0201	0.6 ± 0.03	0.3 ± 0.03	0.33	0.15 ± 0.05
05	0402	1.0 ± 0.05	0.5 ± 0.05	0.55	0.2 +0.15/-0.1
10	0603	1.6 ± 0.1	0.8 ± 0.1	0.9	0.3 ± 0.2
21	0805	2.0 ± 0.1	1.25 ± 0.1	1.35	0.5 +0.2/-0.3



<https://randy-clemmons.blogspot.com/2016/09/when-does-0402-capacitor-footprint.html>
<https://www.worthingtonassembly.com/perfect-0402-footprint>

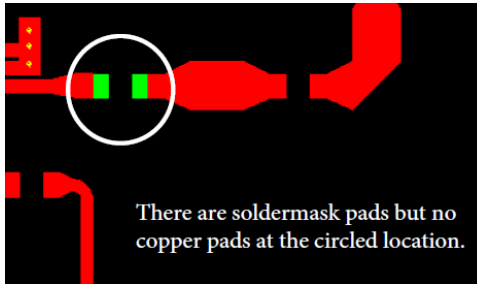
Step 3: PCB Design

- Common DRC issues

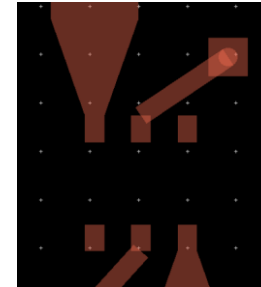
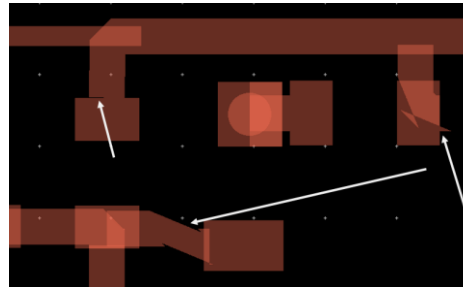


The annular rings on the 1mm drills need to be increased from 0.15mm to 0.2mm to ensure correct plating.

Restring is 0.15mm, needs to be increased to 0.2mm



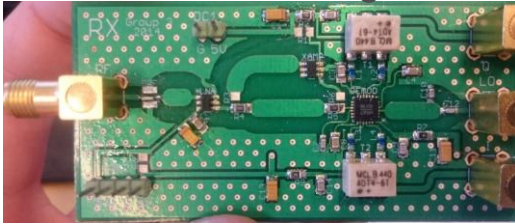
There are soldermask pads but no copper pads at the circled location.



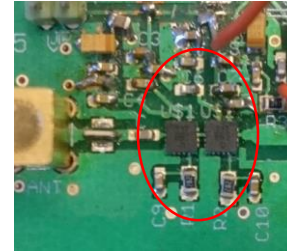
Step 3: PCB Design

- Advices

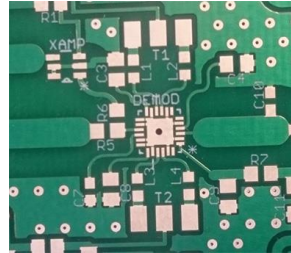
Modular design



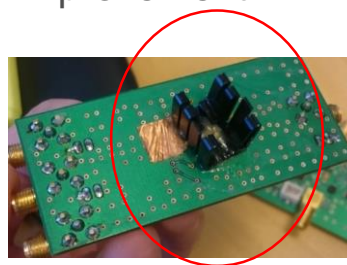
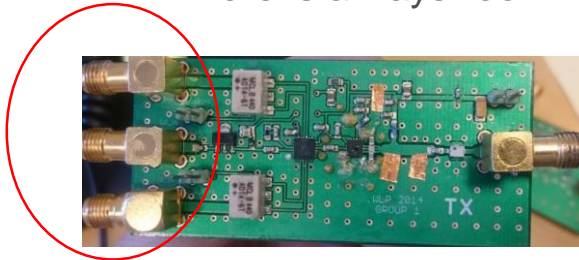
Don't place components too close



Via holes!



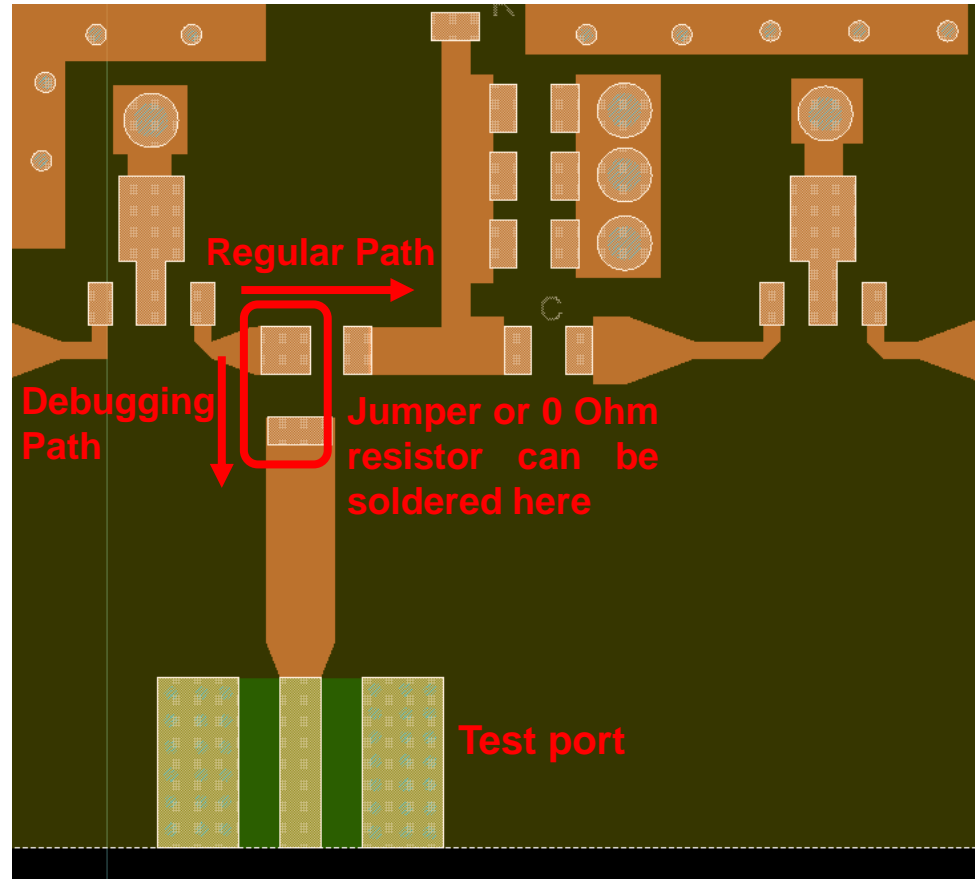
There is always room for improvement



Step 3: PCB Design

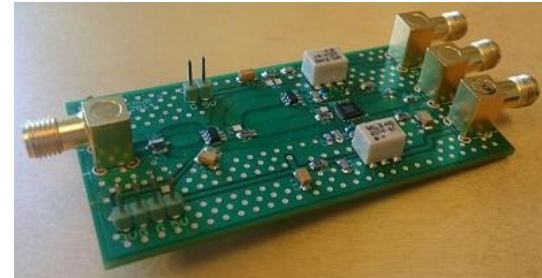
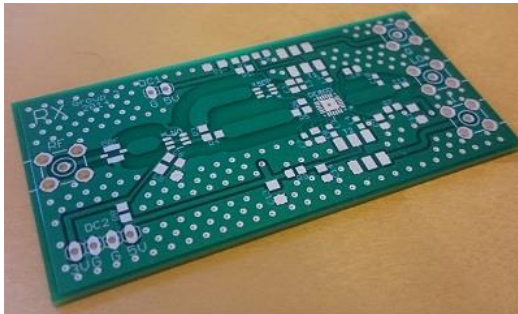
- Plan how to de-bug your circuits

SMA breakouts for de-bugging



Step 4: PCB Design

- Assembly of PCB
 - "Tricky" components (e.g. QFN packages) need to be reflow soldered
 - Passives and connectors can be hand soldered, but utilize the reflow oven as much as possible!
 - [SMD Soldering tips](#)
 - The student lab has:
 - 2 Soldering stations + 1 reflow oven
 - 1 Microscope
 - 1 Dremel (for drilling, cutting, etc)
 - Most components are expensive - don't take more than you need!



HW Design Steps

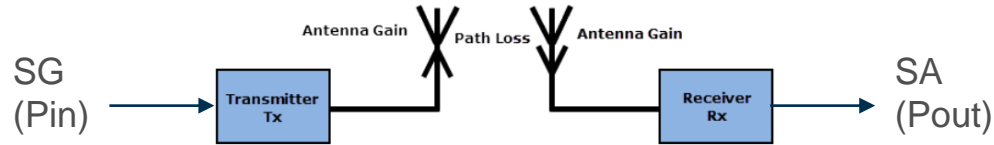
1. Link budget to system specifications
2. Transceiver topology selection
3. Components selection
4. PCB design
5. RF Measurements

Step 4: RF Measurements

- Verify your circuits draw the correct DC current
 - Verify the gain and saturation power of both Tx and Rx using lab instruments such as signal generators (SG) and spectrum analyzer (SA)
 - Well-equipped lab
 - Power supply
 - Handheld Digital Multimeter (DMM)
 - Oscilloscope
 - Spectrum Analyzers
 - Signal Generators
 - Vector Network Analyzer (VNA)
 - Laptop (for portable measurements)
- } Stored in locker!
- Consult tutors if you need to access instruments or need help with operating them

Step 5: Test your wireless link

- Make sure you can transmit a signal with your Tx and receive it
- Verify the level of the received signal



- Test the link with the modulated signal from the USRP
- Try to receive your message with the USRP

Step 5: RF Measurements

First measurement attempt usually doesn't work

Don't Panic

- Before measuring – think through your setup
 - Are the power levels reasonable? I.e. will anything blow up?
 - Correct biasing?
 - Consult tutors if unsure
- RF connectors are sensitive, handle with care!
 - Align connector properly and finger-tighten it. Use torque wrench for last bit of tightening.
 - Never use anything except a torque wrench to secure RF connections!



Part II:

PCB Design and Submission Instructions

PCB Fabrication: Eurocircuits Standard pool

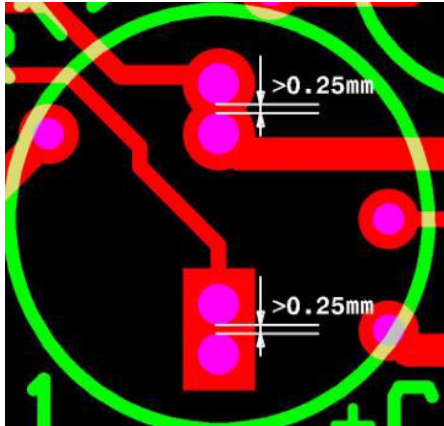
Dielectric FR4 1mm thick

Description of Design rules at:

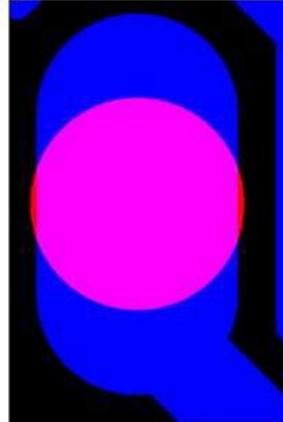
<https://www.eurocircuits.com/pcb-assembly-standard-pool/>


See also PCB Design Guidelines at:

<https://www.eurocircuits.com/pcb-design-guidelines/>



NOT ACCEPTED
(OAR < 0.000mm - Breakout)



https://www.eurocircuits.com/pcb-assembly-standard-pool/ Wireless link project Översätt Wikipedia	
<div>  <div> HOME OUR SERVICES TECHNICAL GUIDELINES STUDENTS & TEACHERS </div> </div>	
Legend Type/Colour	Pooling – White on one or both sides Non-Pooling – Black & White PIL
Min. Track Width/Spacing	Pooling – 0.100mm Non-Pooling – 0.090mm
Min. Finished Hole Size	PTH – 0.100mm NPTH – 0.200mm Non-Pooling – others available in the online calculator
Minimum Outer Layer Pad Diameter = Finished Hole Size + listed Value	PTH – 0.300mm NPTH – 0.200mm
Minimum Inner Layer Pad Diameter = Finished Hole Size + listed Value	PTH – 0.350mm NPTH – 0.250mm
Minimum Copper to Board-edge Clearance – Outer Layers	Routed – 0.250mm V-cut – 0.450mm

Layers convention:

ADS layer/Eurocircuits layer name:

cond – Top copper

cond2 – Bottom copper

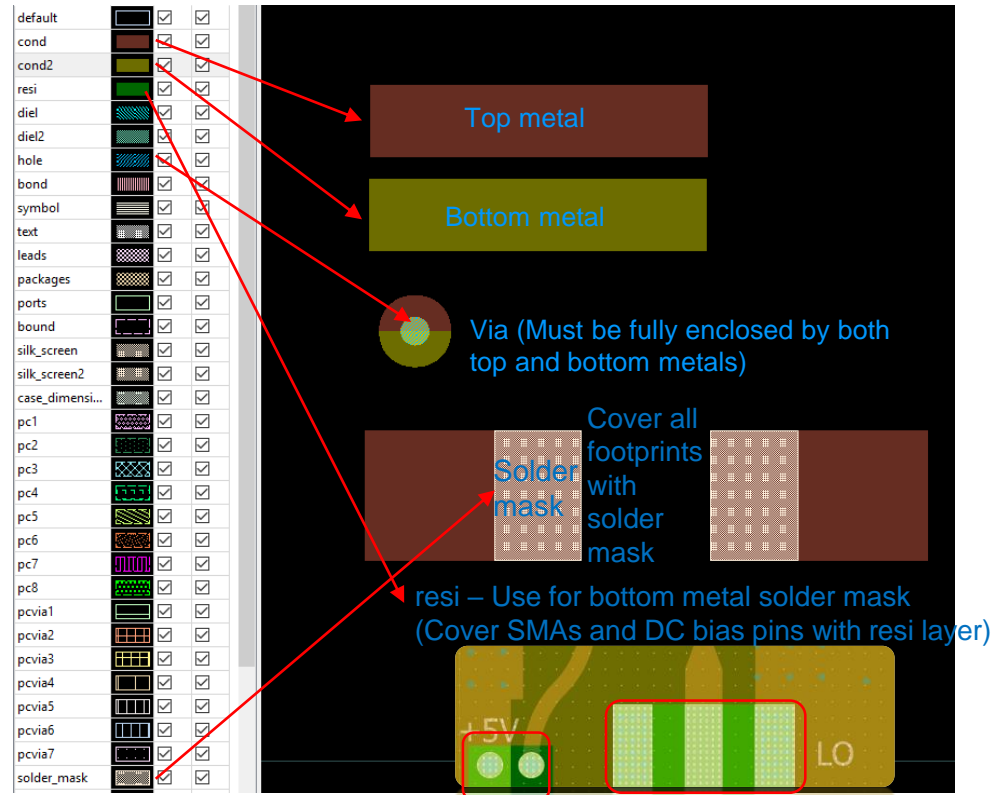
solder_mask – Top soldermask

hole – Plated drill

resi – Bottom soldermask

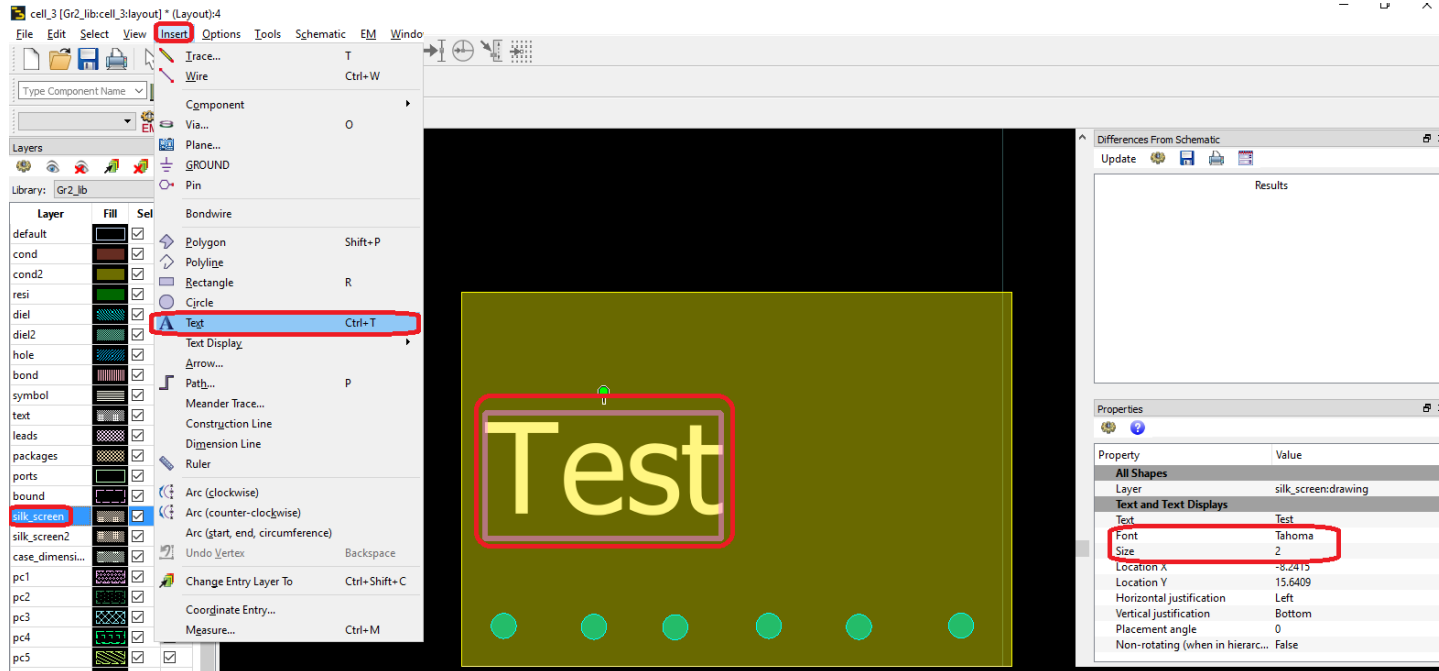
silkscreen – Top legend

case_dimensions- Rout



Layers

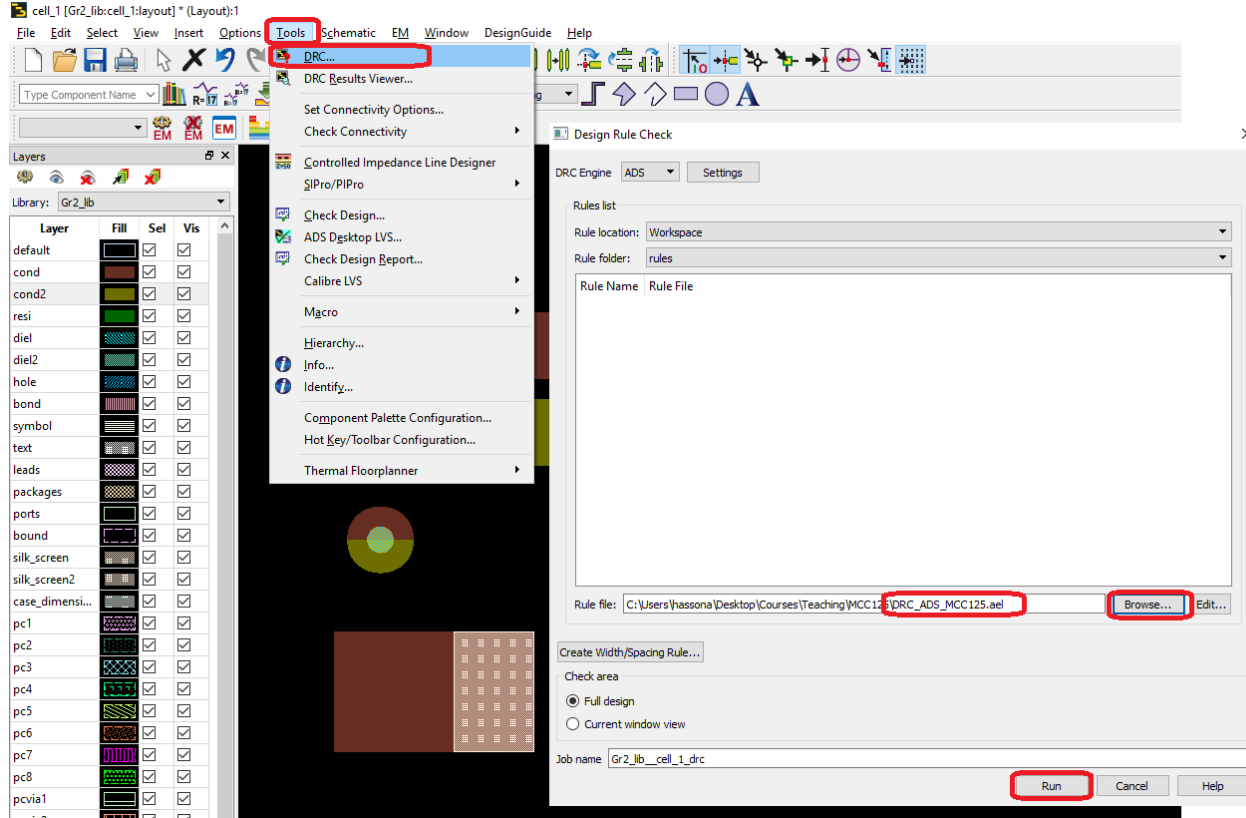
- Extra Features: Silk_screen is used to write text on the board (you can use it to write component numbers, your group name, Tx/Rx ...etc)
- Please use large clear fonts as shown below



Design Rules

- The PCBs will be manufactured on a FR4 substrate with a thickness of 1.0 mm.
- It will be a two-layer board with 1 oz. (35 μm) thick copper traces (Electroless Nickel Gold finish).
- The minimum trace width/spacing is 0.125 mm.
- Drill holes are at least 0.25 mm in diameter with a minimum 0.25 mm clearance to neighboring holes or traces (edge to edge).
- Rules Summary:
 - Substrate FR4 Substrate thickness 1.0 mm
 - Trace thickness 35 μm copper w. ENIG finish
 - Minimum trace width 0.125 mm
 - Minimum trace spacing 0.125 mm
 - Minimum drill hole size 0.25 mm
 - Minimum drill hole clearance 0.25 mm
 - Minimum through hole pad 0.3 mm larger than drill hole size
- More details about the rules can be found in the following: [Link](#) (Class 7D)

DRC



Final DRC

- Final DRC check has to be performed through the online DRC engine at <https://be.eurocircuits.com/>
- Register an account at eurocircuits (one per group)
- Upload your zipped gerber files and create an “order”
- You can now visualize your order using the “PCB Vizualizer” tool

EURO CIRCUITS

Home Cart 0 English Vessen Vassilev Logout

Running orders

[Modify order](#)
[Cancel order](#)
[Order eC-stencil](#)
[View details](#)
[View delivery](#)
[Status details](#)
[History](#)
[Download production data](#)
[Exception details](#)
[View PCB image](#)
[Files](#)
[Track your package](#)
[Request pre-production approval](#)

[Edit administrative details](#)
[Add tracking info](#)
[Ask question](#)

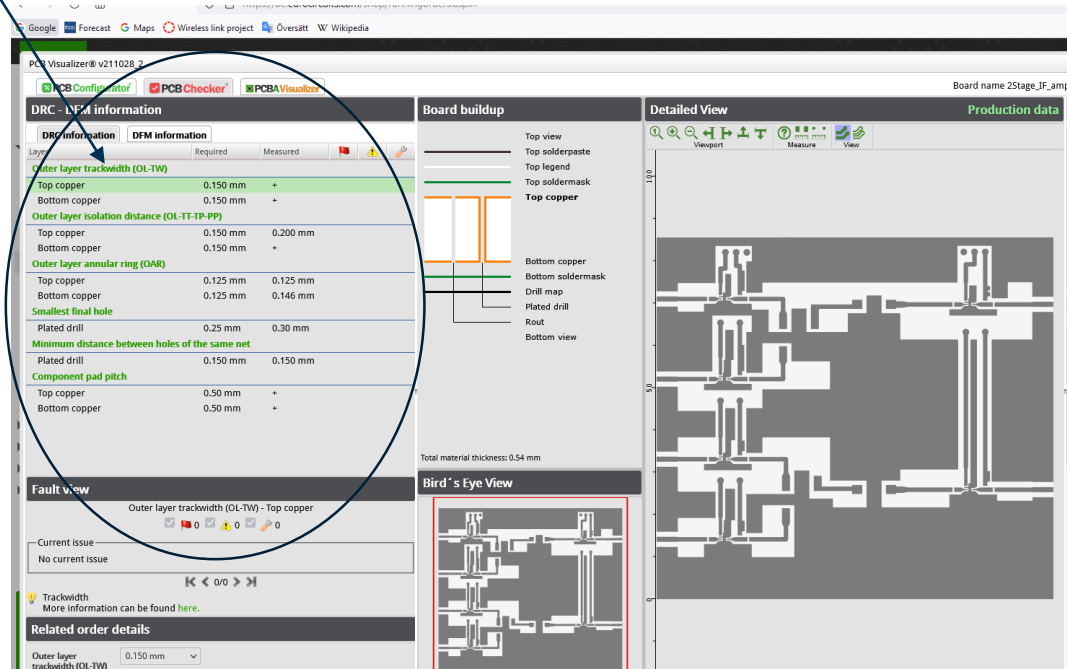
Order no. PCB name Purchase ref Project ref Article ref Order type Status Order date Order qty Panel qty

Order no.	PCB Visualizer®	Assembly Visualizer	Remarks	PCB name	Purchase ref	Project ref	Article ref	Order type	Status	User name	Order date	Shipment date	Panel qty	Order qty
<input type="checkbox"/> E1360177	PCB Visualizer®			2Stage_JF_amp_IF_Board_stencil				Stencil Service	SI prep	vesсен.vassilev@chalmers.se	28 Oct 2021	03 Nov 2021	1	1
<input type="checkbox"/> E1359273	PCB Visualizer®	Analyse BOM & CPL		2Stage_JF_amp_IF_Board				RF pool	Panelise	vesсен.vassilev@chalmers.se	25 Oct 2021	02 Nov 2021	0	3

View running orders 2

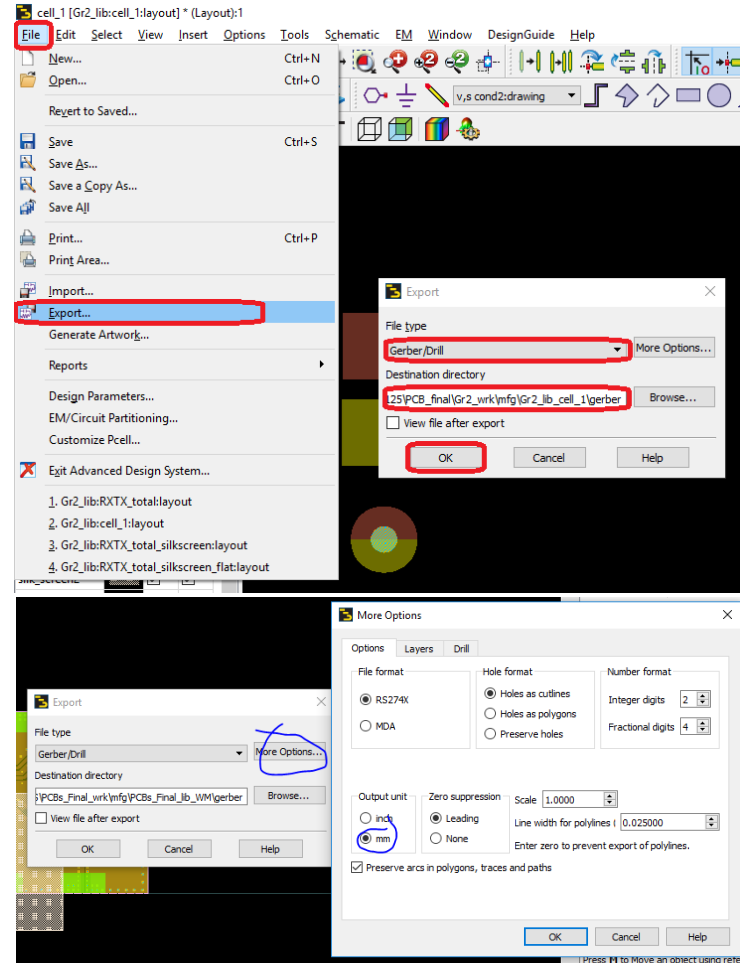
Final DRC

- Go to PCB checker, a list of your errors will be displayed
- A DRC free design is required before layout submission
- For more details check a video uploaded at the course page in Canvas, for better quality download the video on your computer.

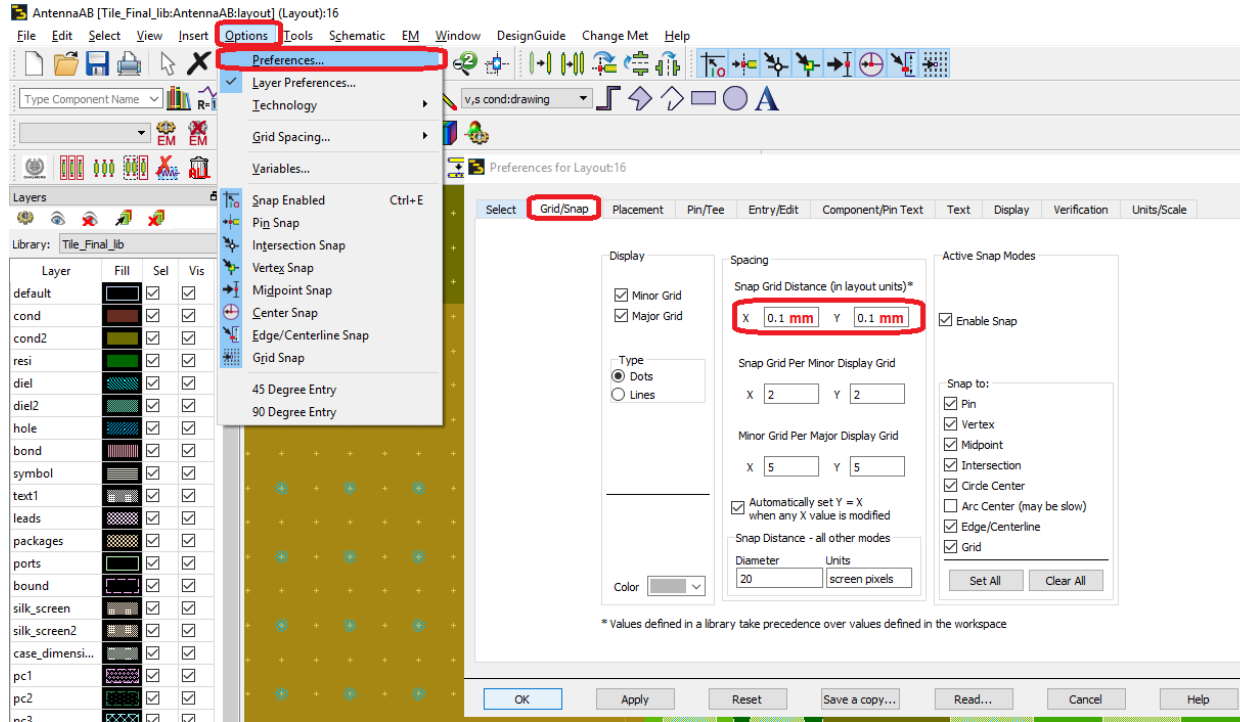


Generating output files

- Manufacturer only accepts Gerber format
- In Gerber format you generate one file for each layer
 1. cond
 2. cond2
 3. hole
 4. solder_mask
 5. resi
 6. silk_screen
- After you export the files, re-import them in ADS and check that everything looks ok.
- **Make sure that you export dimensions in mm not inches**
- **Flatten and merge layout before exporting**
- For more details check a video uploaded at the course page in Canvas, for better quality download the video on your computer.

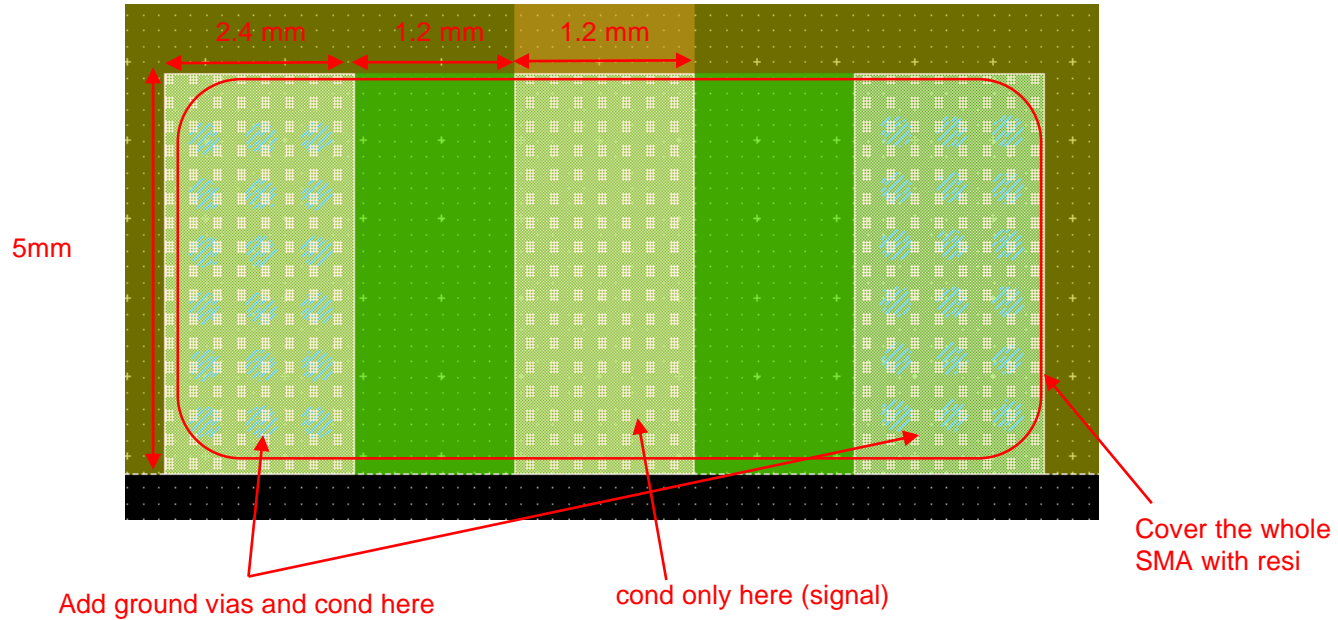


Grid settings



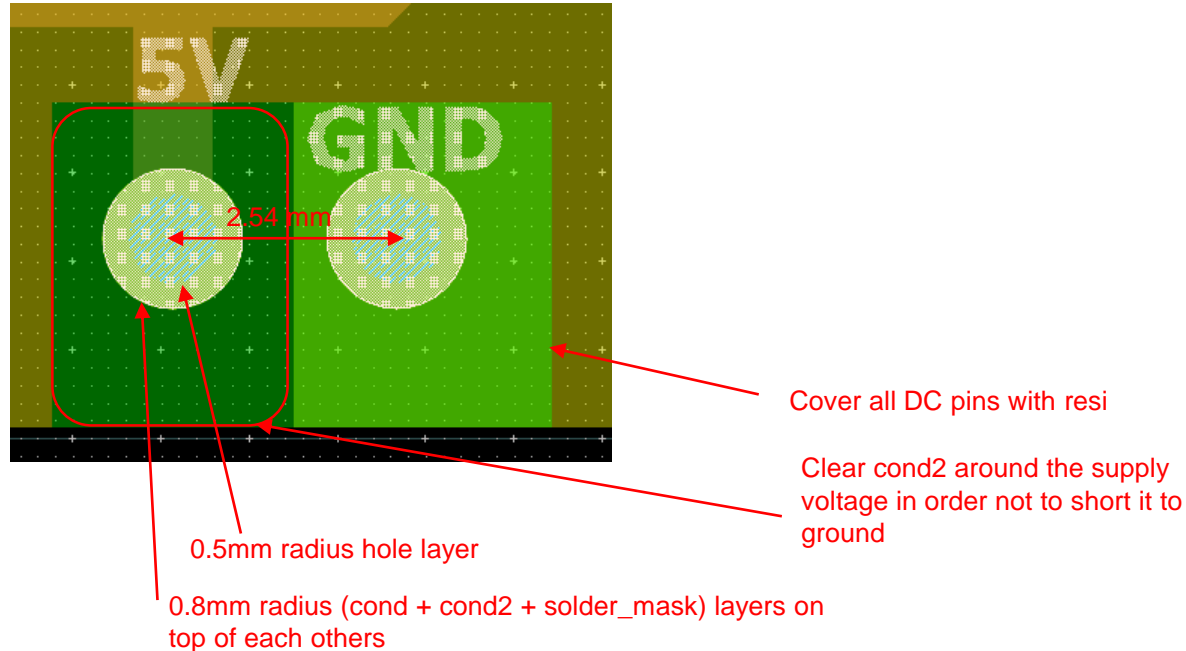
SMAs

- Used for RF, LO and IF connections



DC bias pins

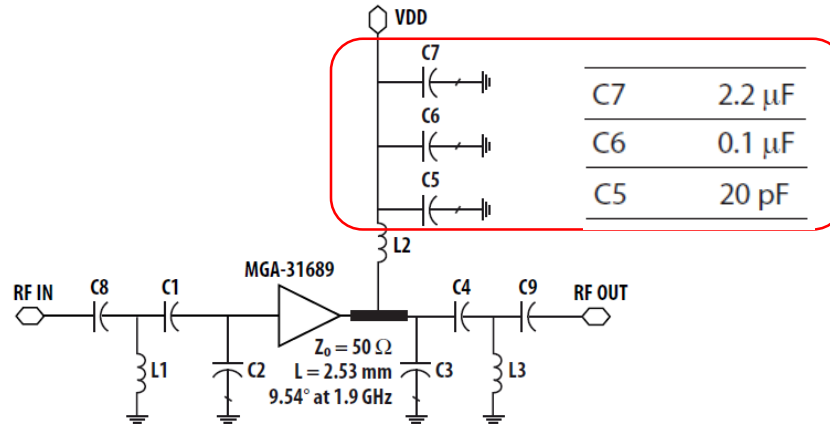
- You can have any number of DC pins separated by 2.54 mm (center-to-center) and having the dimensions shown below



- Note: Try to combine the DC biasing as much as possible to limit the number of bias cables needed.

Decoupling capacitors

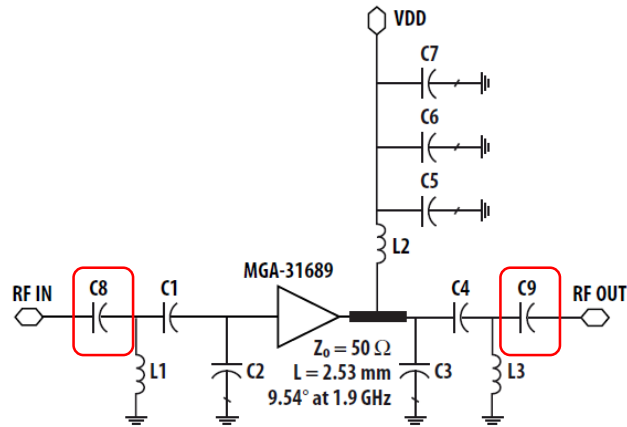
- Used to filter any RF signal on supply network
- Usually three of them are used on the supply of each active RF block
- Use the following values unless otherwise is specified in the application circuit



DC Blocking capacitors

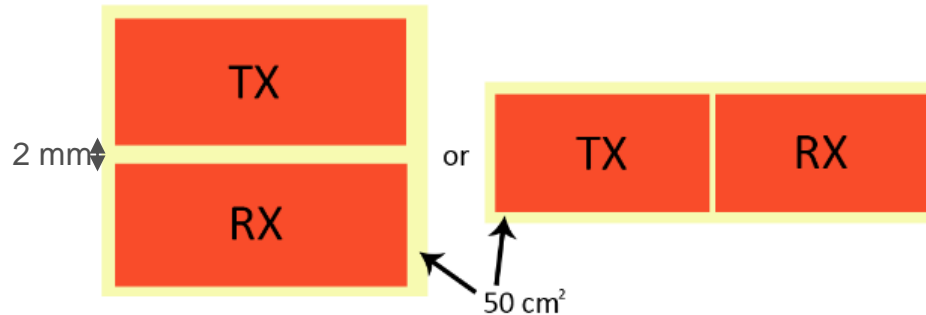
- Used to separate DC bias between cascaded RF blocks
- Note:** Capacitor value should be chosen so that it has low impedance at your frequency of operation

$$X_C = \frac{1}{\omega C} = \frac{1}{2\pi f C}$$



Allocated size for each group

- The maximum allowed board size is 50 cm² for both the transmitter board and receiver
- Submit both in the same layout (Separate them by at least 2mm)
- It's up to you how to orient them in the layout as long as they don't exceed the specified area.



More Information

- <https://www.eurocircuits.com/pcb-design-guidelines/#introduction>



The screenshot shows the Eurocircuits website interface. At the top, there is a navigation bar with links: HOME, PCB SERVICES AND SOLUTIONS, and GUIDELINES. Below the navigation bar, there is a file list table with columns: Name, Size, and Type. The file list contains the following entries:

Name	Size	Type
Layer_1.gbr		
Layer_3.gbr		
Layer_4.gbr		
Layer_5.gbr		
Layer_6.gbr		
Layer_7.gbr		
filter_v1_0.inf	4 KB	Setup Information
filter_v1_0.gb4	3 KB	GB4 File
filter_v1_0.gb3	2 KB	GB3 File
filter_v1_0.gb2	10 KB	GB2 File
filter_v1_0.gb1	5 KB	GB1 File
filter_v1_0.gb0	1 KB	GB0 File
filter_v1_0.drl	2 KB	DRL File

Below the file list, there are four design guidelines:

4. **DO NOT scale** your data. All data provided must be scale 1/1 (100%).
5. Make sure that your Gerber files **DO NOT** contain apertures with a zero-size (size = 0.00mm or inches) and
6. Use the same offset for all your Gerber layers and the Excellon drill data. Preferably use no offset at all.

Below the guidelines, there are two images of PCB layouts. The left image shows a PCB layout with a red background and green outlines, with a caption below it: "All layers have the same offset →". The right image shows a PCB layout with a red background and yellow outlines, with a caption above it: "→ All layers have a different offset!".



CHALMERS

UNIVERSITY OF TECHNOLOGY