Design Checklist for Electronic Schematics and PCB Layouts

Doc. Version 2011-11-01-3.0

Abstract: *Guideline and checklist for development and drawing of electronic circuits, schematics and PCB layouts. Focus is on issues critical for system reliability, stability, testability and production often neglected and rarely covered by text books. Some specials regarding the usage of the CAE tool EAGLE[1] are also touched.*

Keywords: *device models, PCB, library, ratings, JTAG, IEEE1149.1, jumper, designators, values, numbers, net class, placement, alternative fitting, DFT, SMD, THT, reset, power-up, power cycling, FPGA, CPLD, pull resistors, unused pins, floating pins, ESD, vector font, prototype, solder stop lacquer, shorts, track width, temperature, thickness, drill size, connectors, via, pad, reference marks, German, English, translation, DRC, ERC, polygons, airwire*

1 Introduction

By providing this checklist I let my customers participate in my expertise I have been collecting over years as design and test development engineer. Some points of this work probably do not appear in most textbooks.

I do not intend to address academic problems of electronics but fairly simple rules that get neglected or forgotten during schematics and PCB design. Taking those rules serious much wasting of time and money can be avoided.

The line spacing throughout this document is left intentionally large to give the reader space for personal notes.

Special thanks to A. Zaffran at CadSoft Computer GmbH at http://www.cadsoftusa.com for the tables in the appendix. They have been written in German. A translation table from German to English keywords can also be found there.

I appreciate all of critics to improve the quality of this document !

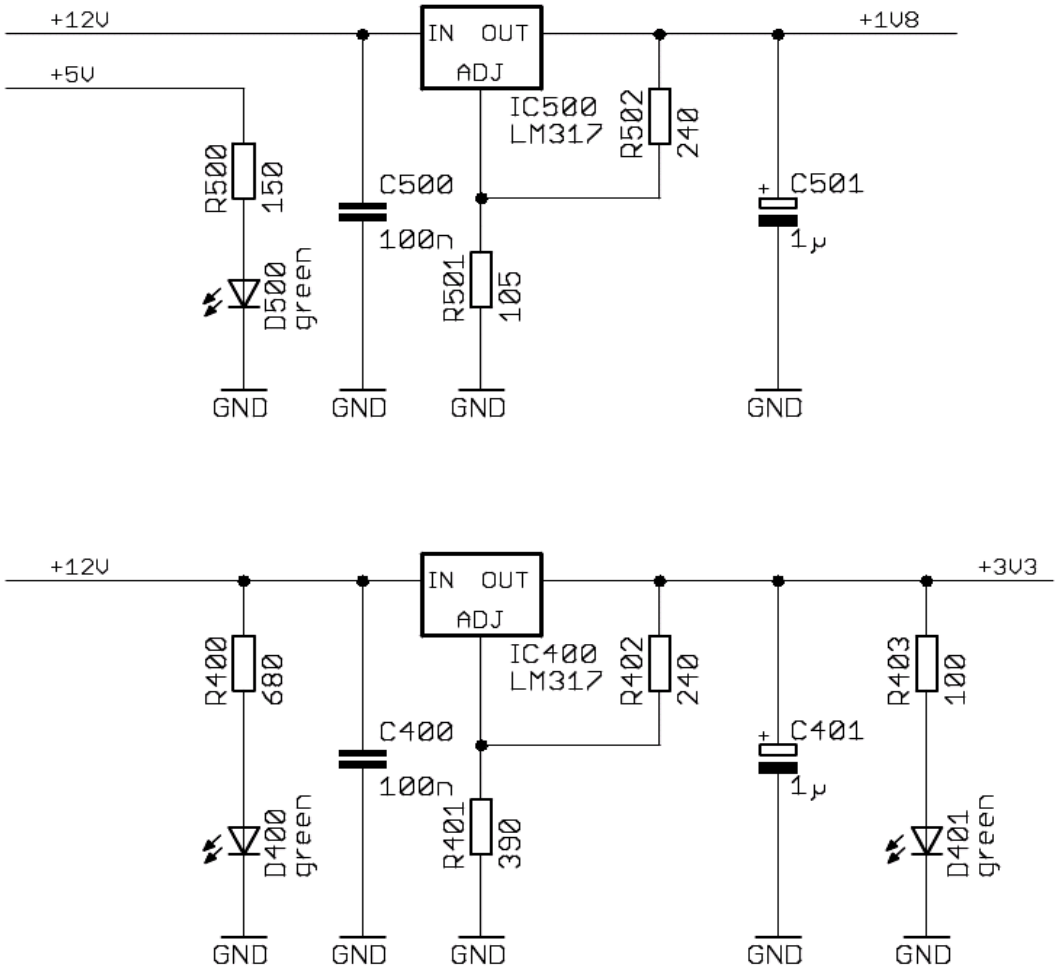
1 EAGLE is a registered trademark of *CadSoft Computer GmbH*.

Contents

2 Schematic

1. Verify the pinout of your device models in the CAD library !
   - Manufacturers of CAD / CAE systems usually do not guaranty 100% correctness!
   - Please be cautious when importing third party device models.

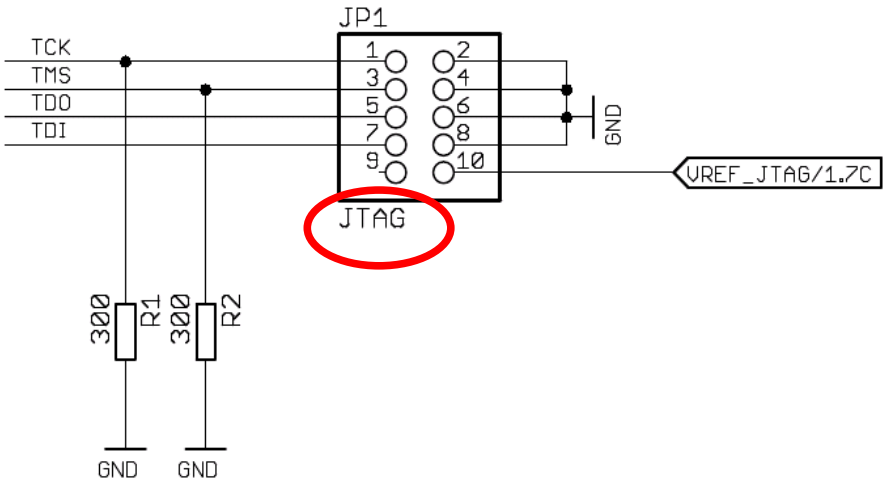
2. Do the device numbering group wise in relation to their function !
   - This eases the later process of PCB design and hardware debugging.
   - See following example: Devices needed to provide the +1,8V supply are numbered with R5xx, C5xx or IC5xx, whereas devices for the +3.3V supply have numbers greater than 400.



Drawing 1: numbering devices

3. Give all devices unambiguous values like 100R/2W or 10µF/35V ?
   - By mere copying of devices wrong or ambiguous values may get spread around the schematic.
   - Especially the **maximum allowed voltage** of capacitors or the **power rating** of resistors gets special attention this way. For example a series resistor for an LED operated at 12V may dissipate up to 0,2W heat !
   - Avoid white-spaces in device values. They may confuse CAD data importing tools for JTAG/IEEE1149.x or other ATE. CAD data importing processes frequently understand a white-space as separation character. *EAGLE* does not forbid white-spaces in values ![2]

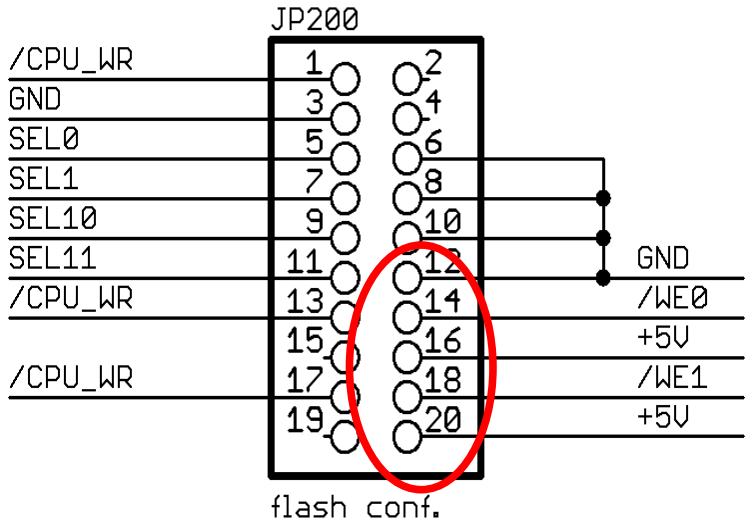
4. Give connectors unambiguous values !
   - E.g. the pin head JP1 should get a value that speaks for its purpose like „JTAG“.



Drawing 2: connector values

2 I recommend to *CadSoft* to output an ERC warning in such cases.

5. Do the connections to pinheaders so that there can **not** be a damaging jumper position !
    - In the example below an inadvertent dangerous short from +5V to GND is impossible.

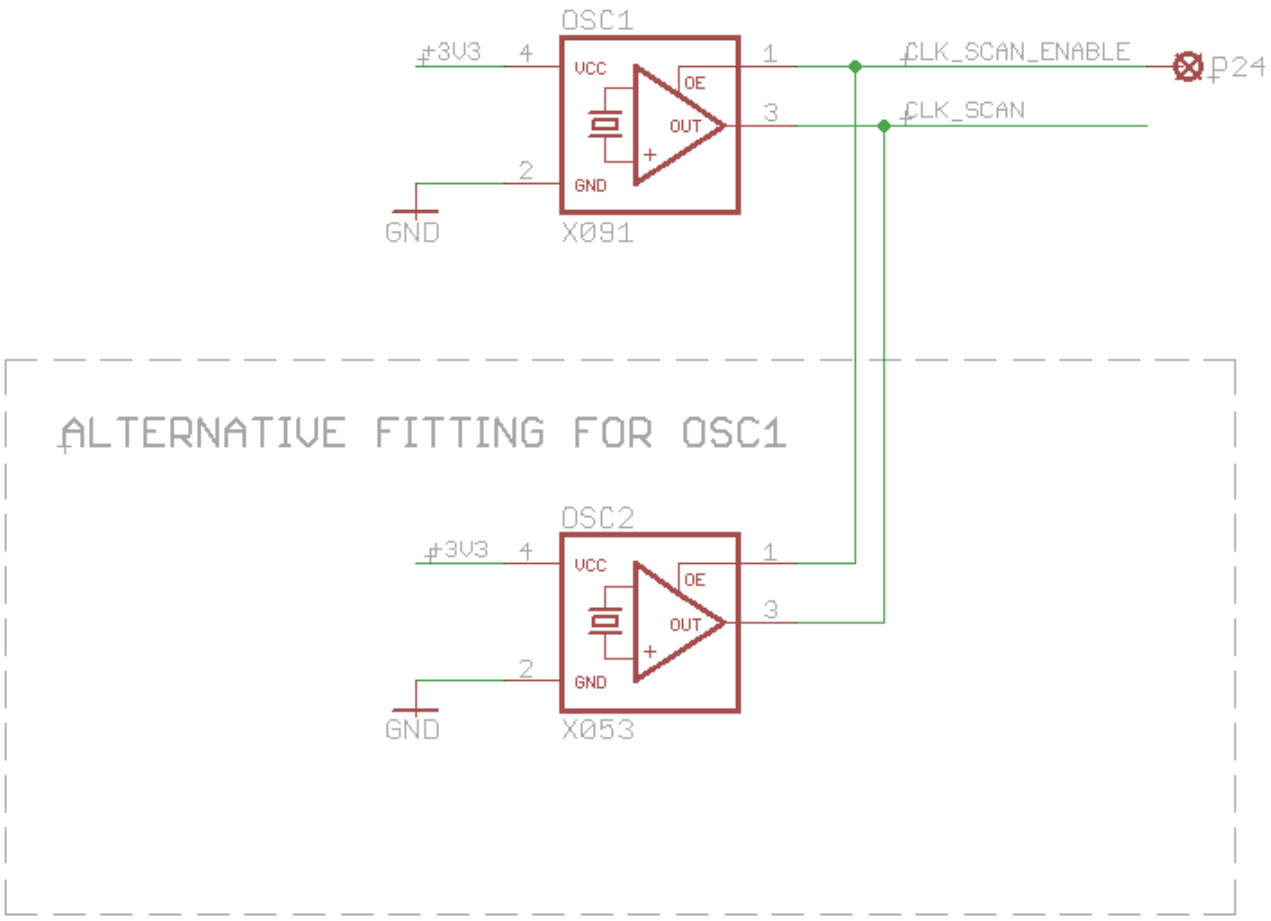


Drawing 3: no danger of power shorts at jumpers

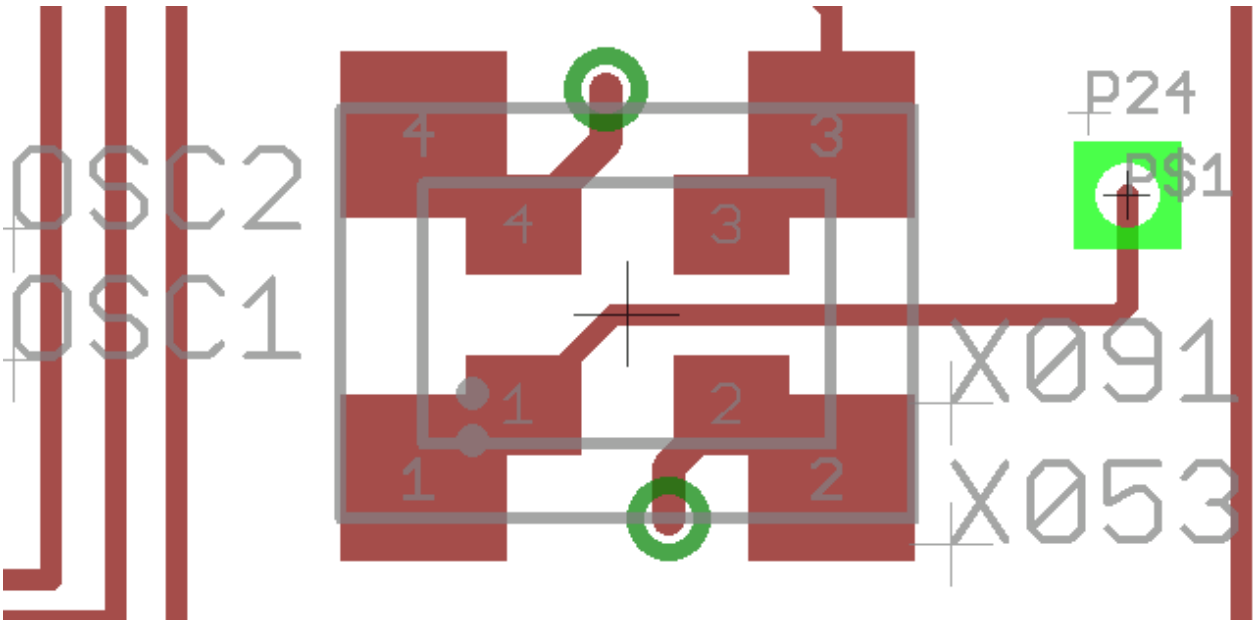
6. Give unambiguous designators to your devices !
    - E.g. the designator „555“ for an IC is not always clear. An LM555 is not necessarily the same as an NE555 (compare datasheets).

7. Take precautions for alternative fitting of devices !
    - Time is running fast today. The production of a *top, super fast, cheap, high performance, ...* IC made for the mass market may become discontinued after 5 years so that you should have a *plan B* ready without the need to change the PCB layout.
    - Component manufacturers notify about the device production status by words like **full production, not for new designs, obsolete, ...**
    - Be also ready for delivery shortages of components.
    - In case the alternative components has a footprint differing from the original one: Draw in your schematic the alternative device parallel to the original

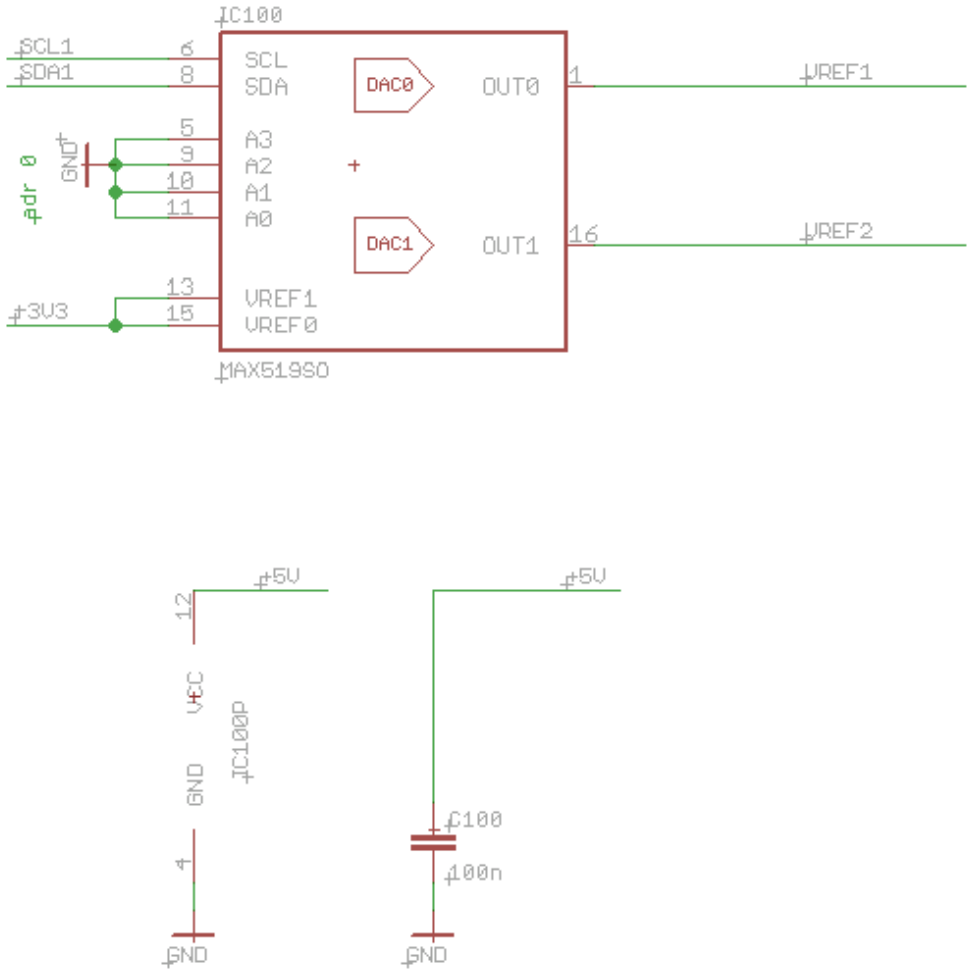
device and leave a text note for your colleges (see Drawing 4 to 8).



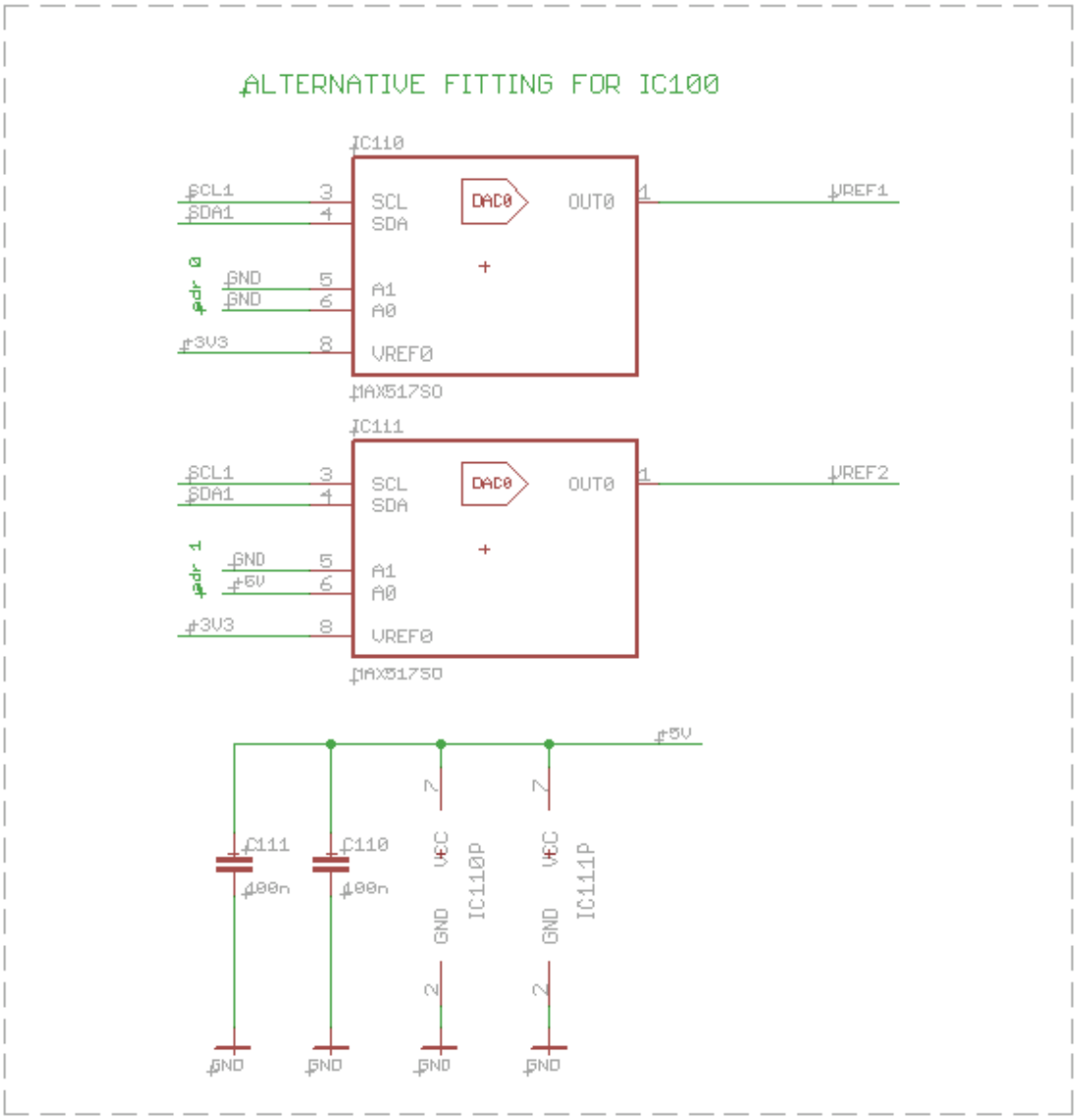
Drawing 4: alternative fitting in schematic



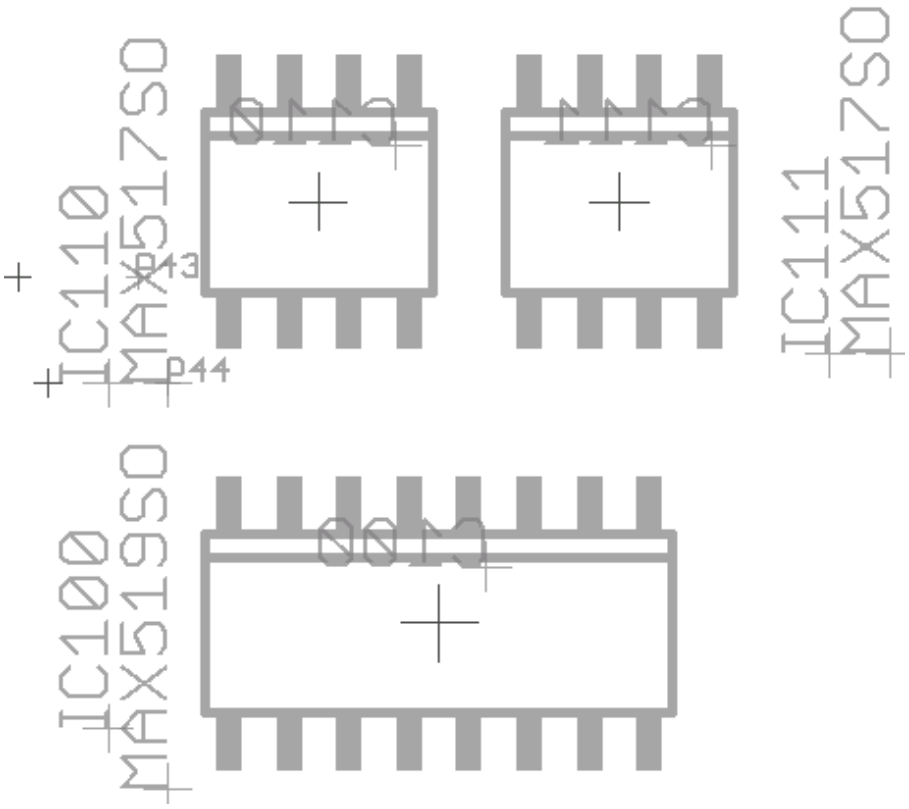
Drawing 5: alternative fitting on PCB



Drawing 6: alternative fitting in schematic



Drawing 7: alternative fitting in schematic



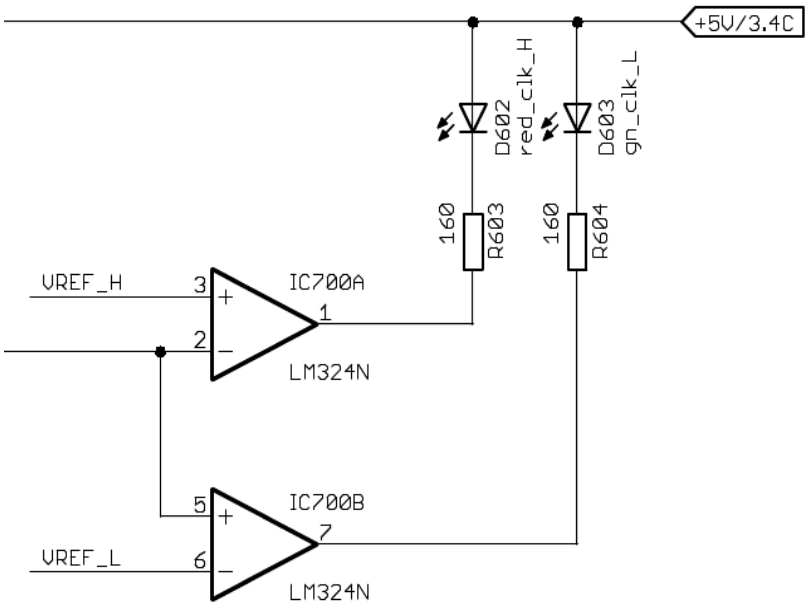
Drawing 8: alternative fitting on PCB

8. If you intend to have the board assembled manually please consider the person doing the job !
   - Use as **less different** values as possible. For example assembling a board with 20 resistors of 10kOhms each is faster and less error prone than a board having 3 x 8,2k , 7 x 12k, 5x 9,5k and 5x 10k.
   - This eases ordering and purchasing the components (**Prices** fall as the ordering amount increases usually.).

9. Does your circuitry also work stable and reliable if
   - optional components are not fitted ?
   - connectors are open for a period of time ? Inputs of ICs may **float free**.
   - Do **not** entirely rely on the ERC (Electrical Rule Check) of your design tool in this case !

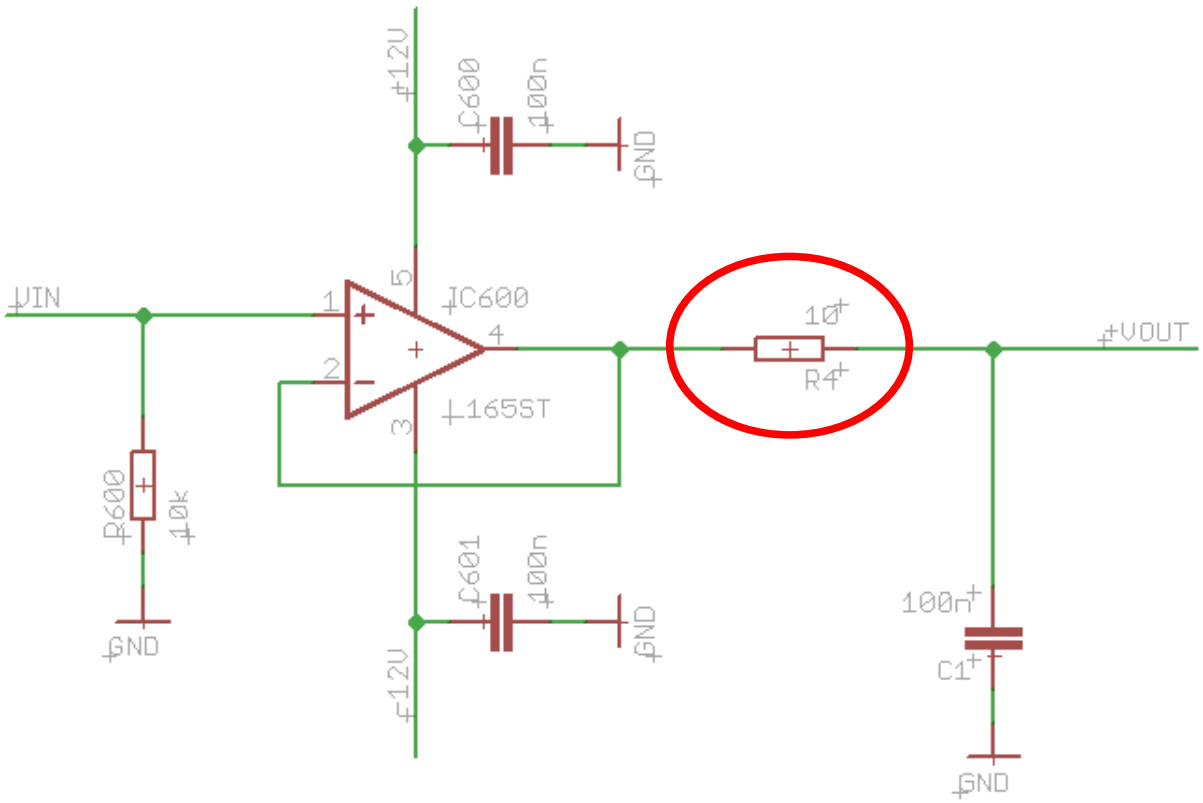
10. Does your circuitry behave predicable during reset or power cycling ?
    - **Note:** During reset the I/O pins of CPLDs, FPGAs, processors and controllers go HIGH-Z or may drive HIGH ! In this phase nets may **float** or cause **dangerous conditions** you have never thought of before like a motor driver bridge shorting VCC and GND.
    - While in reset state or during power cycling collisions may occur. Outputs may drive against each other. Bus contention on wide bus systems does pose stress to driving devices and leads to increased thermal load on them.
    - Does your circuitry perform a proper reset after power-up ?
    - Do capacitors, especially of the reset circuitry, have a path to discharge after power-down ?
    - Do supply voltages reach their nominal value withing the maximum allowed time ?
    - Does the power-up sequence of supply voltages matter ?
    - What is the behavior of **not-programmed** ICs like CPLDs, FPGAs after power-up ? For example the I/O pins of the Xilinx Coolrunner II are pulled high by internal resistors of minimal 40kOhm against VCC_IO.
    - What is the behavior of **programmed** ICs like CPLDs, FPGAs after power-up ? In the fitter options of ISE at www.xilinx.com the power-up value of registers may be set.

11. Inputs and Outputs of ICs
    - Is the load at outputs within safe range – even in case of alternative fitted components ? Even if there is no overload, the voltage at outputs changes under load. Digital downstream inputs may not read the state properly in such cases as shown in the example below: The outputs of the operational amplifiers fall to around 1V sinking the LED currents (provided a single supply voltage of the Op-amp). 1V is hard to be interpreted as L signal for downstream devices.



Drawing 9: op-amp sinking current

- Do (non-FET) operational amplifiers have DC current paths from their inputs to GND ?
- Are drivers (like CMOS drivers) capable of driving (multiple) inputs of downstream ICs. Especially mixing of digital device families may lead to “misunderstandings” among them. (See datasheets for output/input voltage ranges, fan out/in) ?
- Operational amplifiers may drive **negative** voltages (when supplied symmetrical) to (digital) downstream devices which may get stressed or damaged in such a case.
- Avoid **capacitive loads** at op-amp outputs ! Op-amps with feed back loops tend to instability if capacitive loaded. Workaround: Add a series resistor right after the op-amp output to isolate the downstream capacitors (Drawing 10).

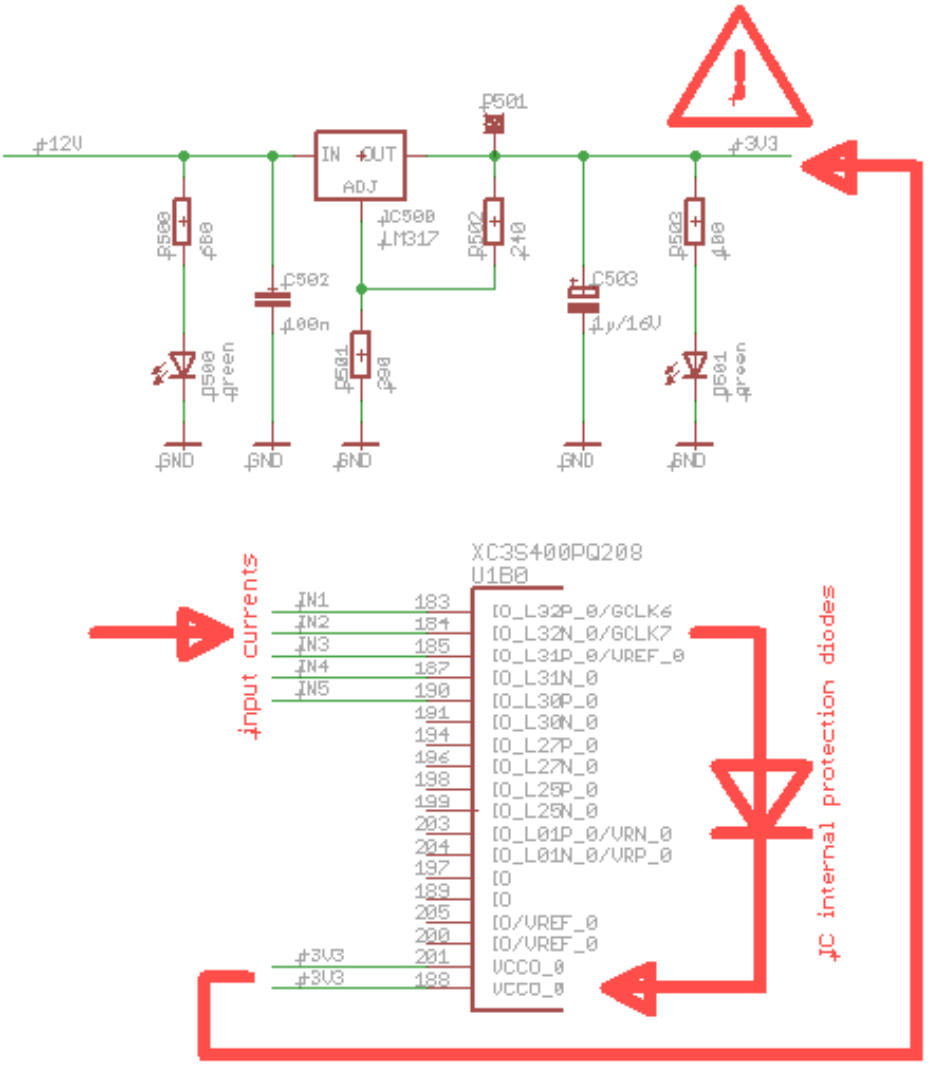


Drawing 10: op-amp with isolated capacitive load

- Are LH or HL edges of digital signals short enough ? Digital devices show **miraculous** behavior if fed with signals changing to slow. For example if you drive a digital clock input of an XC9572 CPLD by a slow operational amplifier like the well known LM324 you will encounter very strange effects...
- In harsh industrial environments an output (at a connector) may get shorted inadvertently. Does the driver survive this load at all ? Please take your time to consider this scenario. Replacing the whole unit at customers site may get more expensive than the time you spend thinking on this matter.
- Probably ESD (electrostatic discharging protection) is an issue for you. *NXP* provides outraging solutions at http://standardproducts.nxp.com/ .

12. Backdriving

- If ICs inputs are driven with voltages exceeding the specified maximum ratings given by the manufacturer, backdriving occurs. More or less current flowing into the inputs will accumulate and drain into the power rail, thus raising the voltage there.
- Most voltage regulators do **not** sink current into their outputs. So there is danger of **damaging** other devices "living" from the affected power rail.



Drawing 11: backdriving into power rail

13. Make sure digital ICs are fed with patterns clearly allowed by the manufacturer !
    - Example: The pattern HLL driven into the inputs A, B and C of an IC causes the device to behave **unpredictable** on its outputs. If you can't avoid such a situation, does it impair the system performance or cause dangerous effects ?

14. Has a minimum of address pins of (FLASH)-EEPROMs been connected ?
    - Example: When programming or just ID code reading of a W29C020 a succession of 5555h, 2AAAh, ... on the address bus is issued by the processor on at least 15 address lines ! If you tied address pins 15..12 low, because you don't need them in normal mode, the device could not be programmed in-system later on ![3]

15. JTAG / IEEE 1149.x
    - Provide termination resistors to GND on TCK and TMS. I recommend from experience 300 Ohms each (see Drawing 2 page 4).
    - Do the boundary scan system (or JTAG-system) drivers have sufficient strength to make a reliable H level on TCK and TMS when loaded with 300 Ohms ?
    - Provide an optional pull-up resistor on TDO of each IC on the UUT/ Target-System.
    - May the optional L-active **asynchronous** Test-Reset (/TRST) signal become required some time ?
    - Layout GND connections generously (see Drawing 2 page 4).
    - Provide means of **switching scan paths** (jumpers, Scan Path Linkers, Scan Bridges, …). For example while system programming the scan chain should be as short as possible (IEEE1149.7 relieves you from that because it allows star topology of scan paths). Another scenario: Optional not fitted ICs within the scan path cause breaks in a serial chain – therefore provide solder bridges between TDI and TDO of the device affected in order to close the

3 This is a very rare case but it indeed happened to me once.

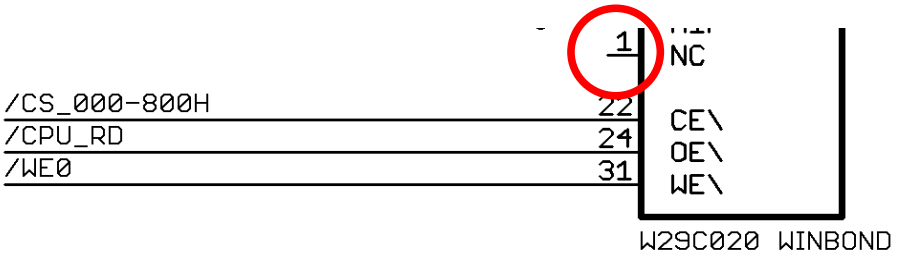
break.

- Alternatively, devices needed for in system programming may be placed in a separate scan path. Short scan paths → less programming duration → lower production **costs** !
- /WE, /RD or /CE signals should be wired to pin heads or jumpers to allow driving them by the boundary scan system externally. This way in system programming can be sped up to a large extent.
- Programming adapters of programmable logic manufacturers may require a reference voltage provided **by the UUT** (see Drawing 2 page 4), whereas boundary scan systems may **output** a voltage to supply the UUT. Make sure these voltages can't get shorted.

16. Design for Test (DFT)
    - **Note:** Your product will make it to series production some time.
    - Care for safe, fast and easy testing of the board in advance !
    - Contact PCB and assembly houses during board design.
    - If the majority of devices is of analog nature the board is likely to be tested by ICT (In-Circuit-Test) or manually. Have you provided test pads on critical signals ? Also have a look at drawings 14 and 15 on page 18.
    - Multilayer boards fitted with BGA packages usually don't allow **mechanical access** to signals !
    - Digital areas of the board should be tested by Boundary Scan/JTAG according to IEEE1149.1/6/7. See point 15. page 15 .

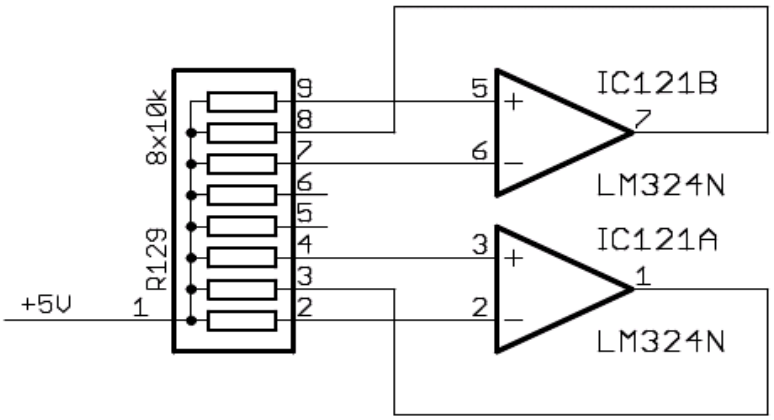
17. Make unused IC pins accessible.
    - Do not just hard wire them to GND or leave them open as this makes extensions or modifications difficult or impossible.
    - This also applies for pins that are named with NC or “no connect”. NC means: *The pin is not bonded or wired inside the IC.* In case there will be a **replacement** by a similar device later, this pin may become important. See following example: A successor of the W29C020, an AM29F040 has pin 1

serving as address input A18. If left open in the original design (where it was an NC pin), the successor IC would have A18 floating free – **fatal !**[4]



Drawing 12: NC pins may become a problem

- The next example shows two left over operational amplifiers of a quad Op-amp (LM324). Should there be a modification later they are accessible since they are “soft” wired via R129.

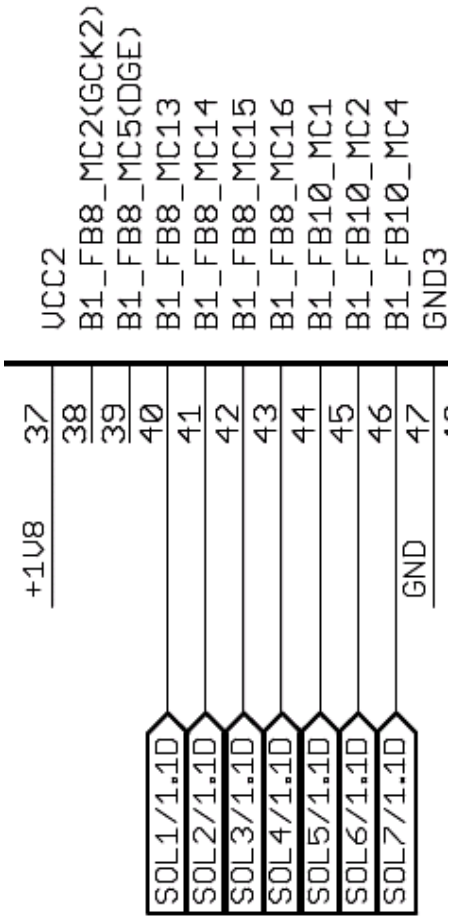


Drawing 13: soft wiring of leftover op-amps or gates

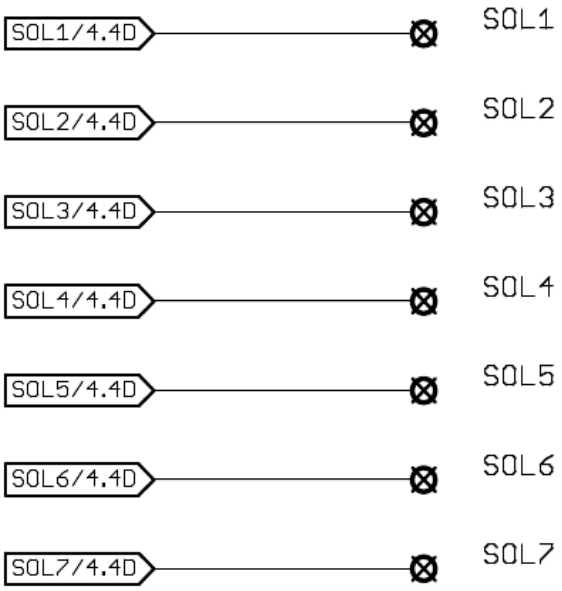
- If you also wire these pins with solder pads or pin header mechanical access becomes simple. Apply this method when using fine pitch packages like TQFP-144 or BGAs.
- I also recommend to wire these pins with pull resistors externally.

4 Some datasheets do not allow to connect NC pins at all, which makes in-advance-connecting of NC pins impossible.

- The example below (left hand) shows 7 unused pins of a Xilinx Cool-Runner which have been wired to solder pads (right hand).



Drawing 14: solder pads



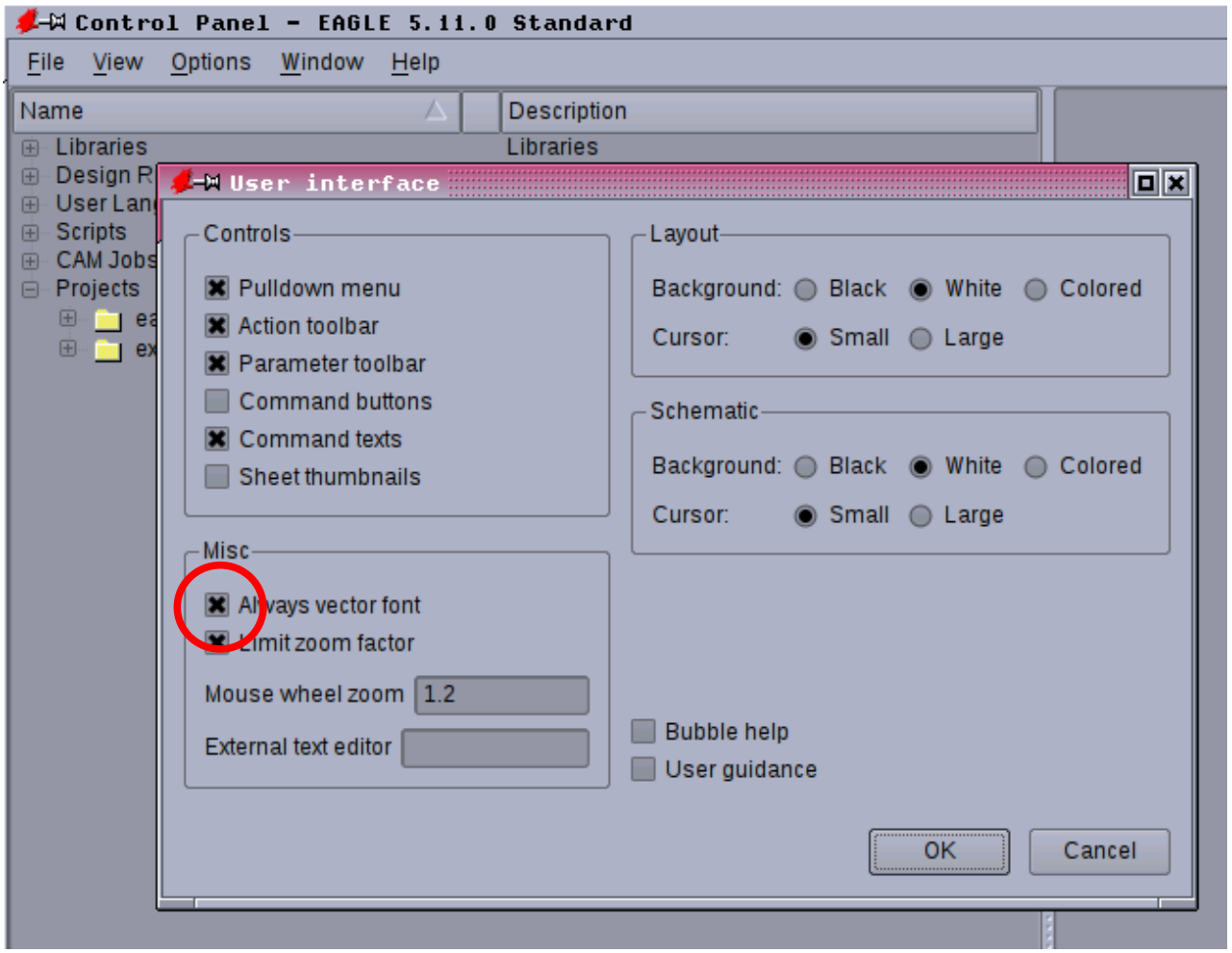
Drawing 15: solder pads

18. Have you named **all** nets explicitly ? Even those without **label** ?
    - Just relying on automatic naming of your design tool does not give much of value as far as later board debugging goes. E.g. a net name „CATHODE_RESET_LED“ does say more to your college than just „N$1701“.

19. Have you defined **net classes** ?
    - For later layout design its important to define track widths and clearances of certain nets **in advance**.
    - Contact your PCB house as far as minimal structure dimensions goes.

2.1 CadSoft EAGLE specific

1. I recommend to **always** use **vector font**.
   - Vector font guaranties constant text size independent of zoom when printing on paper.
   - Vector font is also required for production of your PCB as varying text size may result in **shorts** when using proportional font in copper layers.
   - By default vector font is **disabled** in *EAGLE*[5].
   - By the way, vector font looks much more professional ...

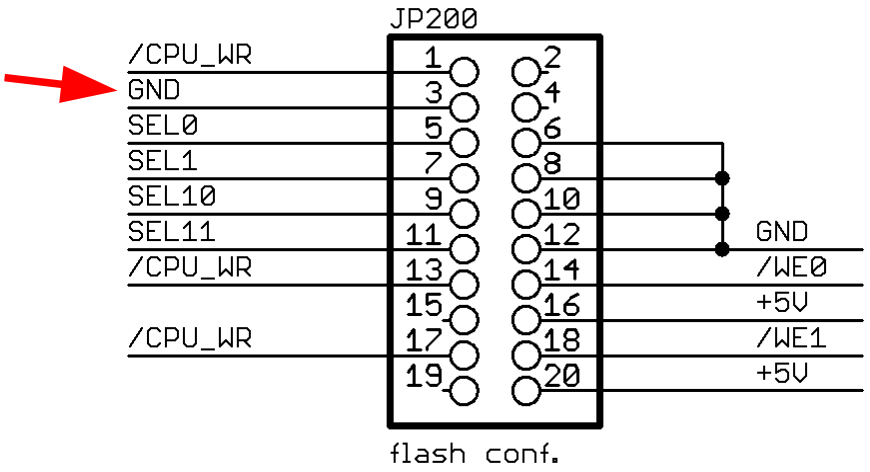


Screenshot 1: always enable vector font

5 I recommend *CadSoft* to make vector font default.

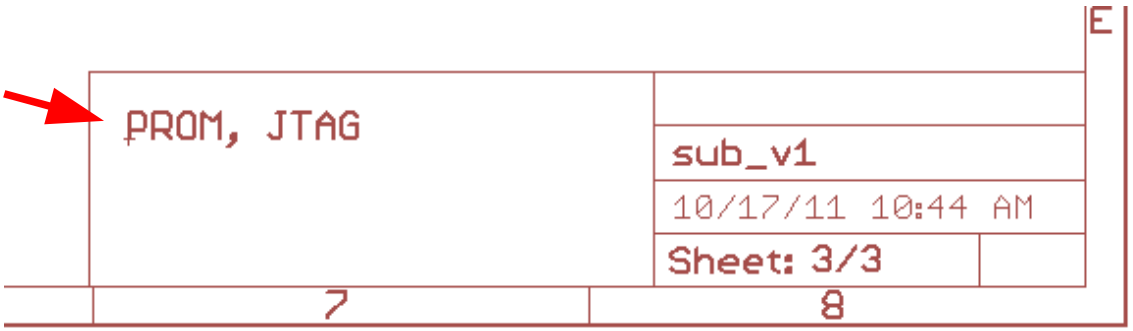
2. Text size

- I recommend to place net labels with size 0,056 inch or 56 mil (Drawing 16).



Drawing 16: net labels

- Texts located in the doc field should be of size 0,1 inch (Drawing 17).



Drawing 17: doc field

3. Do not forget to run the ERC from time to time during schematics design.

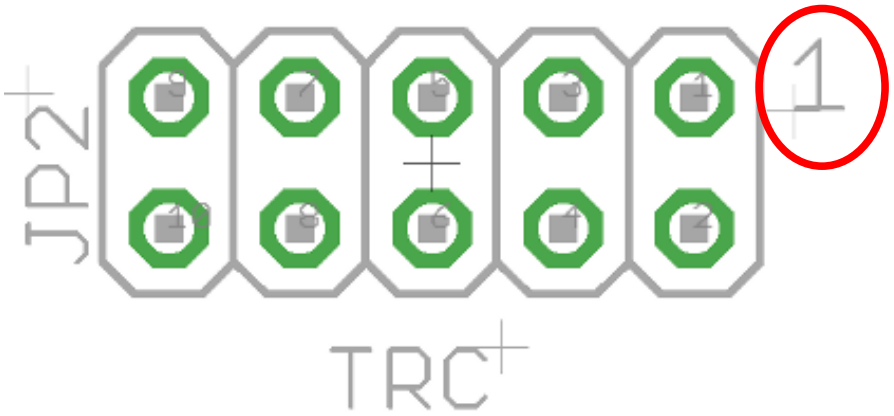
3 PCB Layout

3.1 Prior to routing

1. Check mechanical properties of your device models regarding:
   - package dimensions
   - drill sizes of pads (It's very disgusting if the pins of a screw clamp do not fit into the drills.). Consider maximum drill tolerances guarantied by your PCB house !
   - Care for keep out areas (important with SMD packages) (layers 39 & 40 in CadSoft EAGLE).

2. mounting holes
   - Make sure the hole diameter (including tolerances) is correct.
   - Should there be an electrical connection to **protective ground** ?
   - Use the "lock" command to nail down holes so that they can't be moved inadvertently.

3. placing of connectors, jumpers, switches, LEDs, displays:
   - What are the requirements of the target application ?
   - **Your customer determines, not you alone !**
   - Have you marked pin 1 of connectors (Drawing 18) ?
   - Use the "lock" command to nail down these devices so that they can't be moved inadvertently.



Drawing 18: mark pin 1

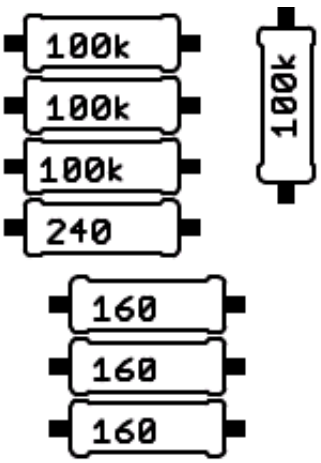
- Arrangements of multiple LEDs should have all the anodes pointing to the same direction (See Drawing 20).
- Pin headers and connectors should have a clearance around so that the counterpart plugged there does not collide with other neighboring devices[6].
- If your target application poses mechanical stress on connectors, I recommend **not** to place them directly on the board as pads tend to get loose or ripped out. Place those connectors in the **housing walls** instead.

4. Make sure devices on top of bottom side do **not overlap** ![7] Consider heat sinks and their accessories that usually require more space than you think.

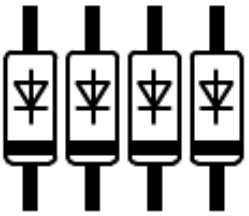
5. In case you plan to **manually** assemble the board (even when using SMD): As a courtesy to your colleges consider following (as far as signal integrity allows of course):

   - Resistors of equal value may be placed in groups.
   - Diodes and capacitors may be placed with their cathodes/poles pointing to the same direction.
   - These measures make the assembling less error prone, faster and **cheaper**.

6 Most THT packages shipped with the EAGLE library do not have a keepout frame.
7 Most THT packages shipped with the EAGLE library do not have a keepout frame.



Drawing 19:

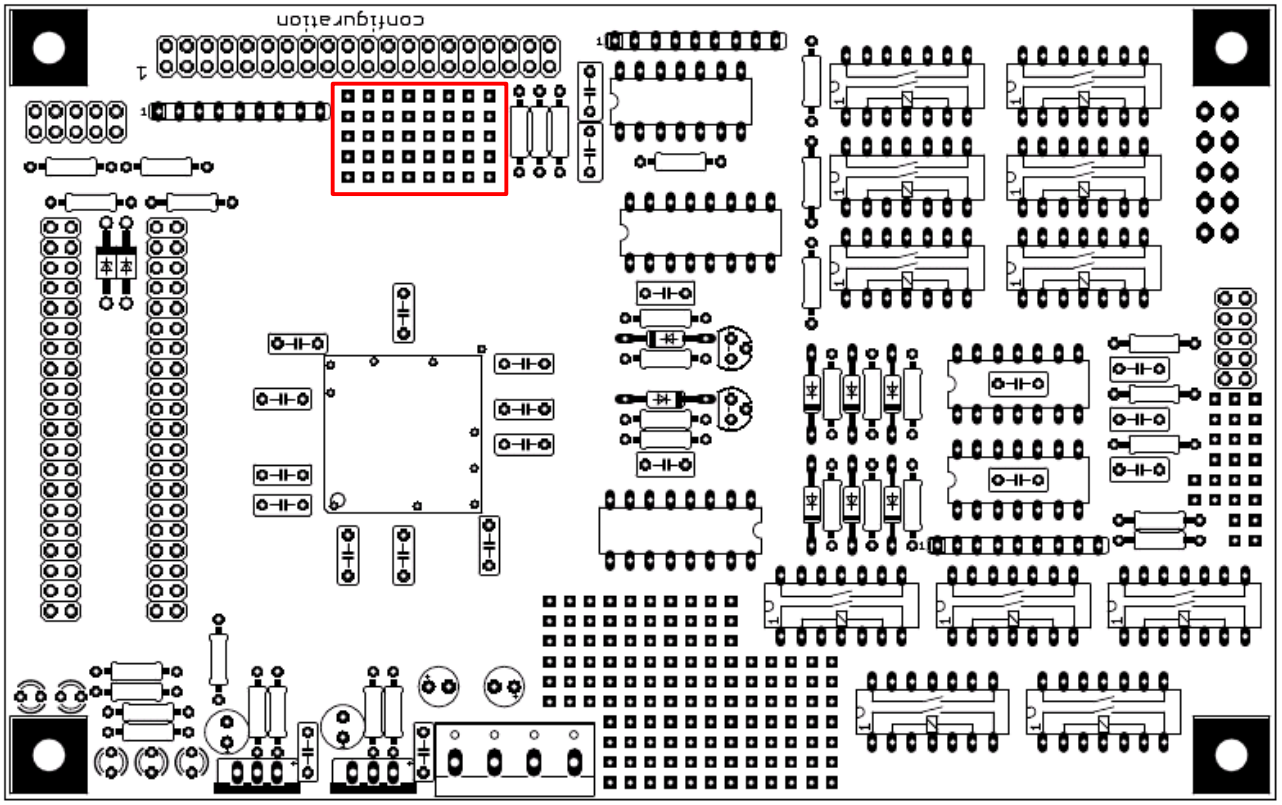


Drawing 20:

6. Board version, top / bottom side marking
   - Place a text in the copper like “top” or “bot” to identify sides
   - Place board name/version in top or bottom side copper like “RS232-IF_V3.2b”

7. Reference marks
   - Contact PCB assembly house as how and where reference marks should be placed on the board. They are required by mounting machines !

8. Solder pads for modifications or extensions
   - When in prototype phase, you are required to make changes on your board like additional pull resistors or just a status LED. As precautions provide solder pads **in advance** as shown in the drawing below framed **red**.

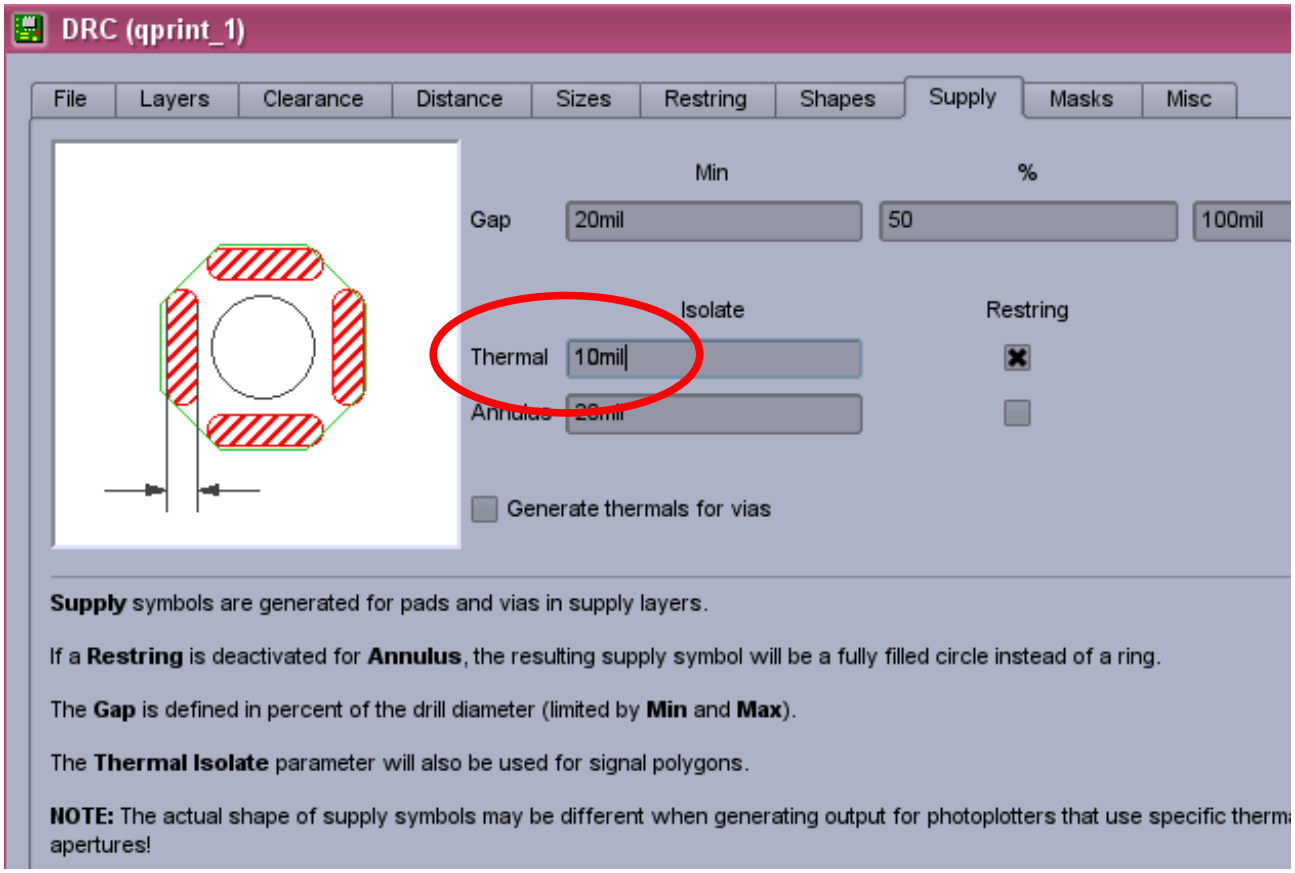


Drawing 21: solder pad arrays

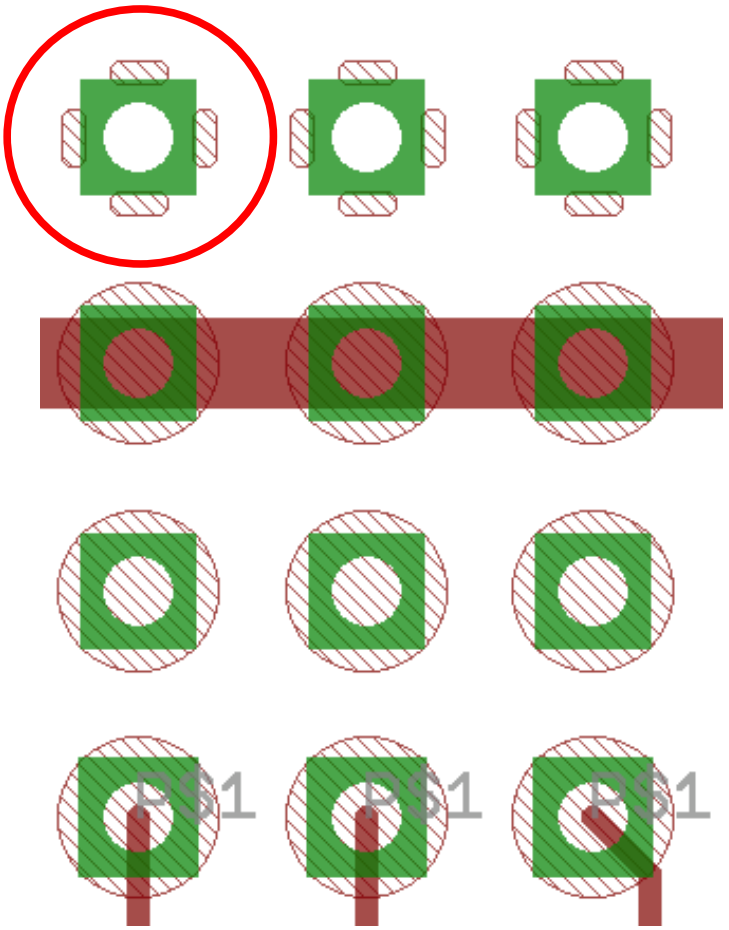
- Solder pad arrays of this kind can be made by pads or lots of vias. Pads, as shown in Drawing 15 page 18, are “real” electrical devices (with symbol and package). Vias can be placed in the board only. They do **not** have a counterpart in the schematic.

  Note : Check the “via thermal” option in the DRC settings. This way vias become “solder able” . Otherwise they become connected to the surrounding copper plane which draws away all the heat that comes from the solder machine (Screenshot 2).[8]

8 This option does apply for inner layers only as these layers are allowed to be supply layers. Thus via thermals can not be created in “top” and “bottom” layers.



Screenshot 2: via thermal setting in DRC



Drawing 22: via thermals in inner layers

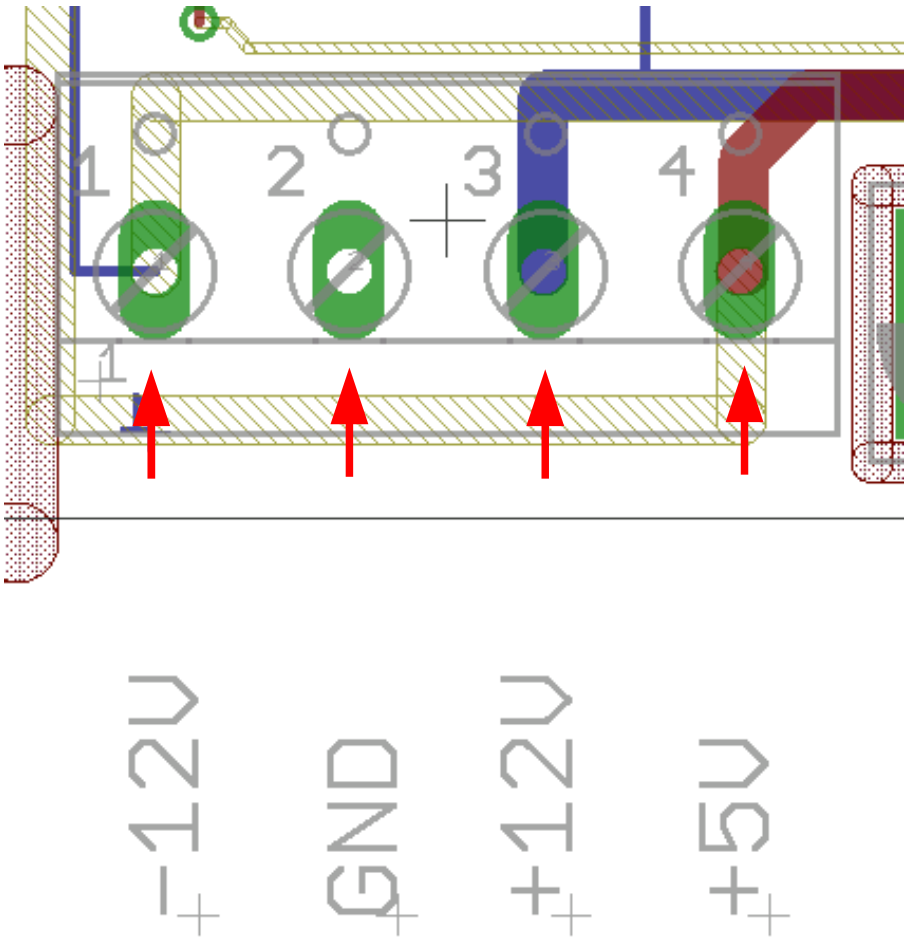
- Modifications or extensions soldered there look relatively good and are of stable mechanical nature.
- See examples in Drawing 14 and Drawing 15 on page 18.

3.2 While routing

1. Run DRC from time to time and **finally** before sending the BRD-file to production !

2. Please consider the maximum current load of copper tracks (see Appendix).
   - Unavoidable **critical** conditions should not lead to **blown tracks**. See also section 2, page 11, point 10. In a harsh industrial environment an output may get shorted inadvertently temporarily.

3. In the proximity of screw clamps or connectors the board is probably going to be stressed heavily.
   - Avoid tracks in top layers where wires are inserted. In Drawing 23 the red arrows indicate the direction wires are inserted. They may **disrupt** a track or scratch the solder stop lacquer – **danger of shorts !**



Drawing 23: no tracks in top layer near a screw clamp

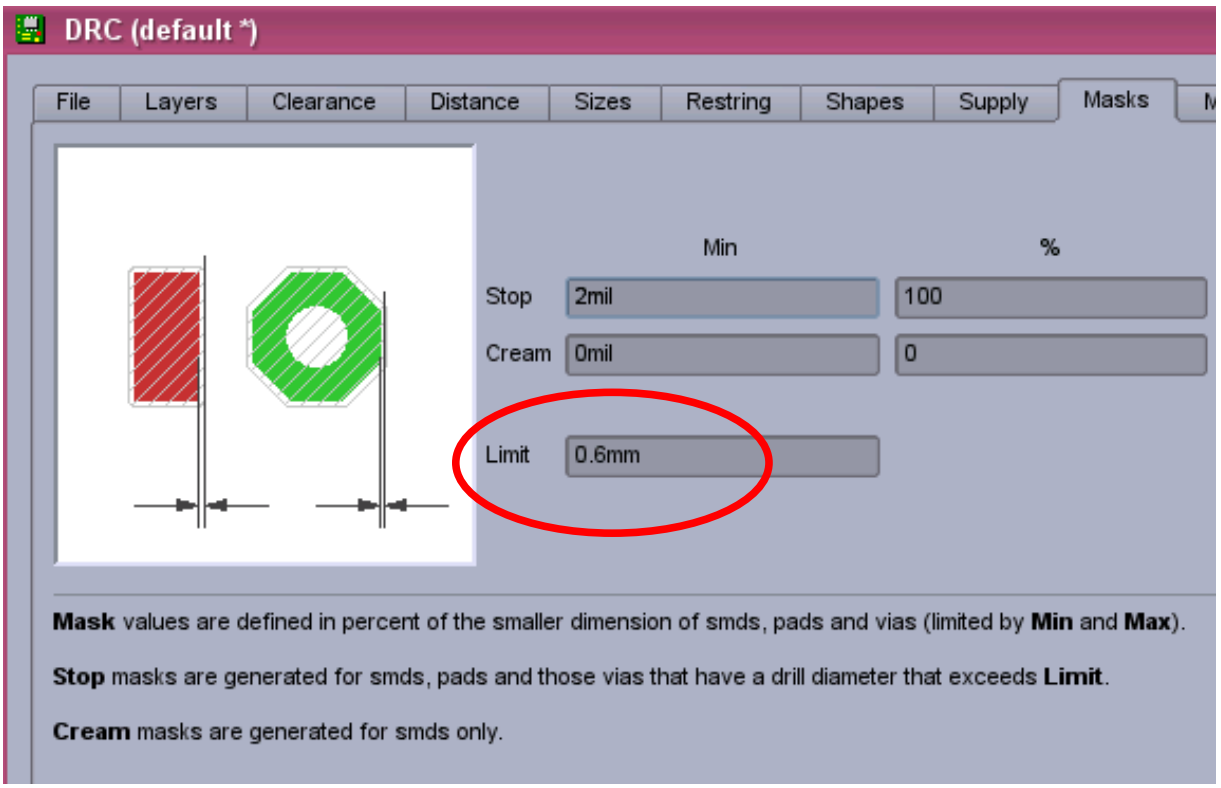
- Generously, as far as possible, provide copper areas on pads or mounting holes in order to improve mechanical stability. Usually device models provide just a minimum of copper around their pads.

4. Solder stop lacquer

   - Vias should be **covered** with lacquer always.

     Note: If the board is to be **wave soldered** shorts between vias and pads may result otherwise !
   - Keep vias free from lacquer in **exceptional** cases only. Screenshot 3 gives an example of how to set this rule in the DRC menu: All drills with a size greater

than 0.6mm do **not** become covered with lacquer[9]. That includes all vias.



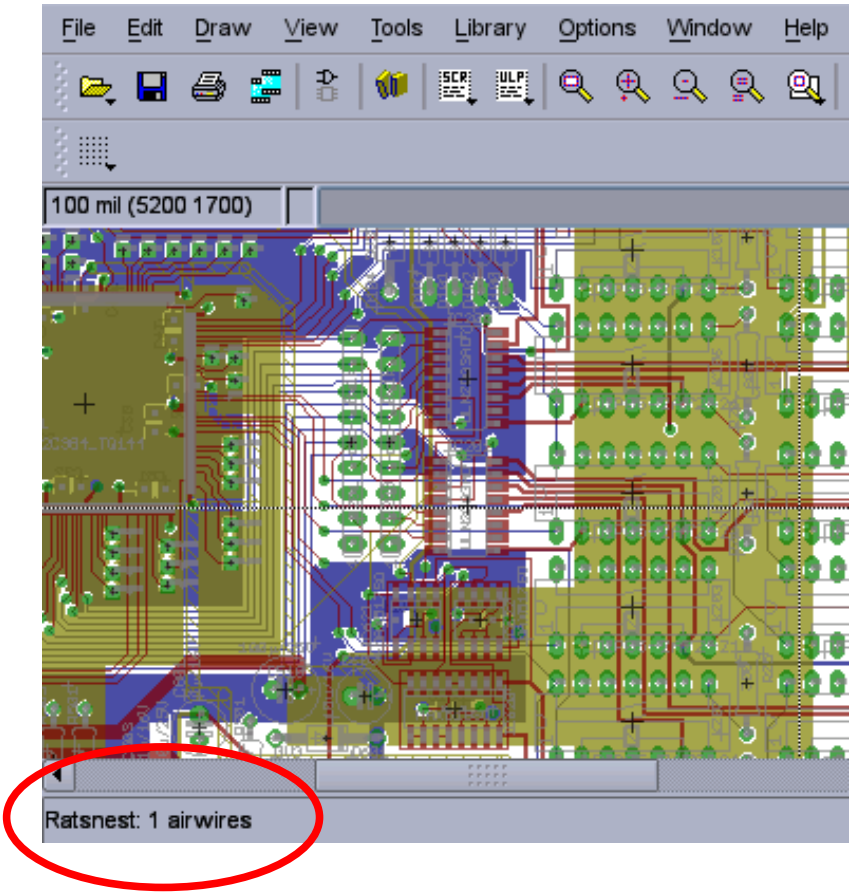
Screenshot 3: solder stop mask on vias

5. Polygons

- Polygons are a great invention. By drawing a polygon in a copper layer and naming it with a signal name like GND lots of routing time can be saved.
- The **disadvantage** is: Polygons may **fall apart** during manual or automatic routing. To make sure polygons are still integer run the command "ratsnest" from time to time. The ratsnetst result is shown in Drawing 24.[10]

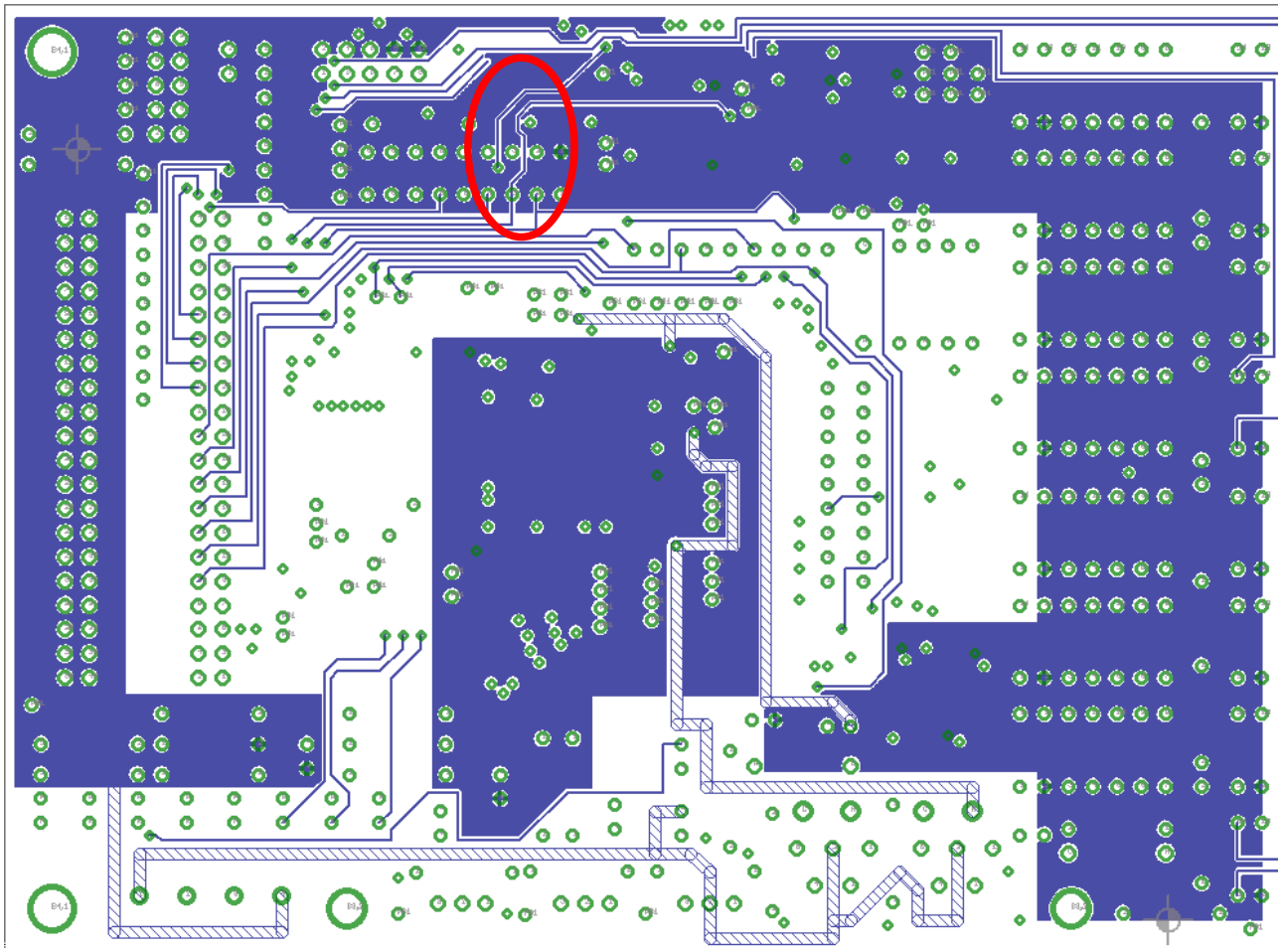
9 The default setting here is 0 mm. Means no drills are to be coated with lacquer.

10 I recommend *CadSoft* to make those airwires flashing so that they can be found easily. Especially very short airwires are hard to find in large drawings.



Drawing 24: ratsnest command

- Another critical issue are bottlenecks from one copper area to another. Drawing 25 shows an example. Red marked is a very narrow strip of copper connecting the left and the right area. This very narrow "bridge" spoils signal integrity and may be blown when carrying high currents.



Drawing 25: bottleneck in a polygon

4 Appendix

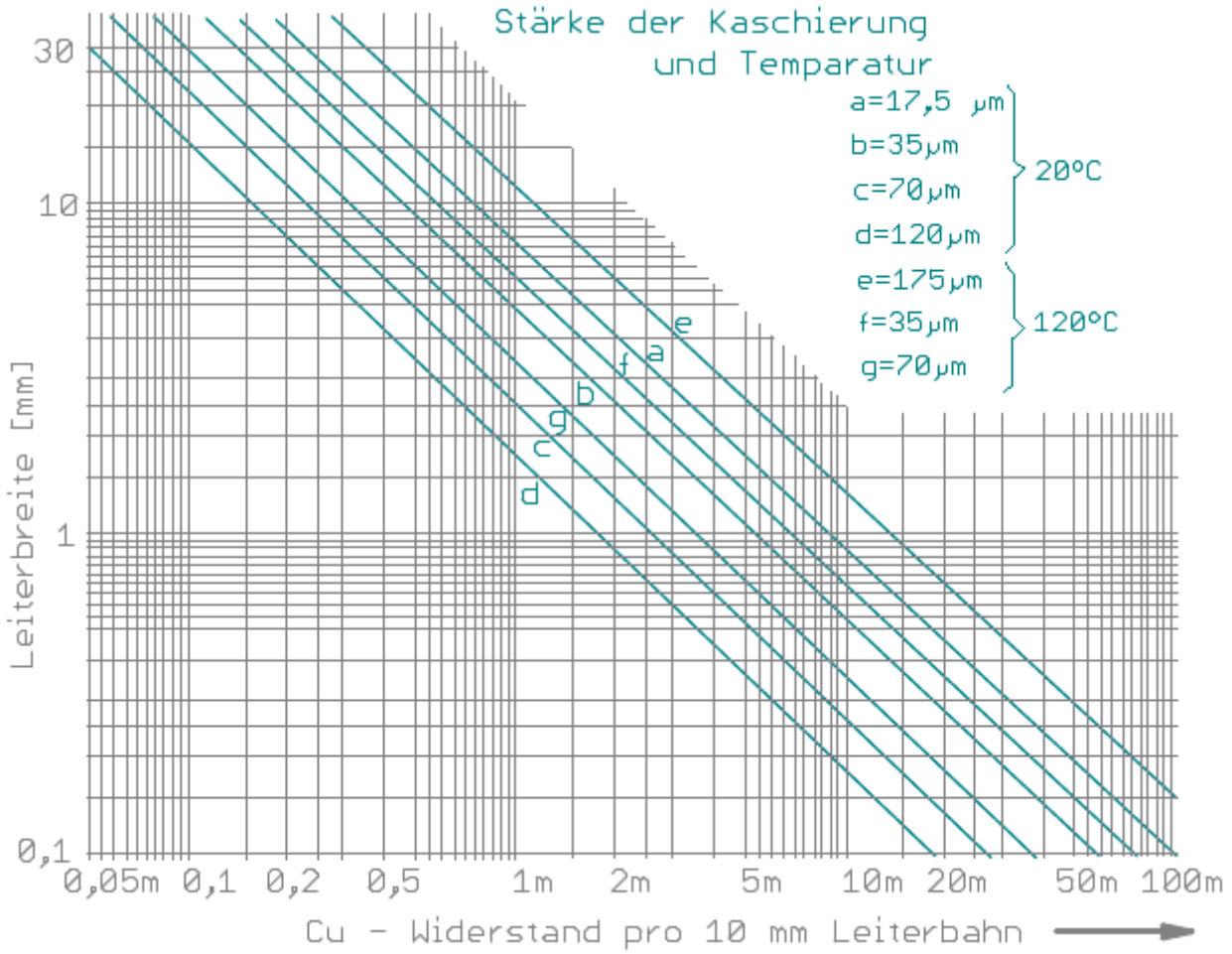


Diagram 1: maximum current load vs track width at various temperatures

German	***English***
Stärke der Kaschierung und Temperatur	thickness of copper coating and temperature
Leiterbahnbreite	track or wire[11] width
Widerstand pro 10mm Leiterbahn	resistance per 10mm track/wire length

11 The EAGLE terminology uses the word “wire” for all kinds of lines regardless if this is real copper or just documentation or measurement.

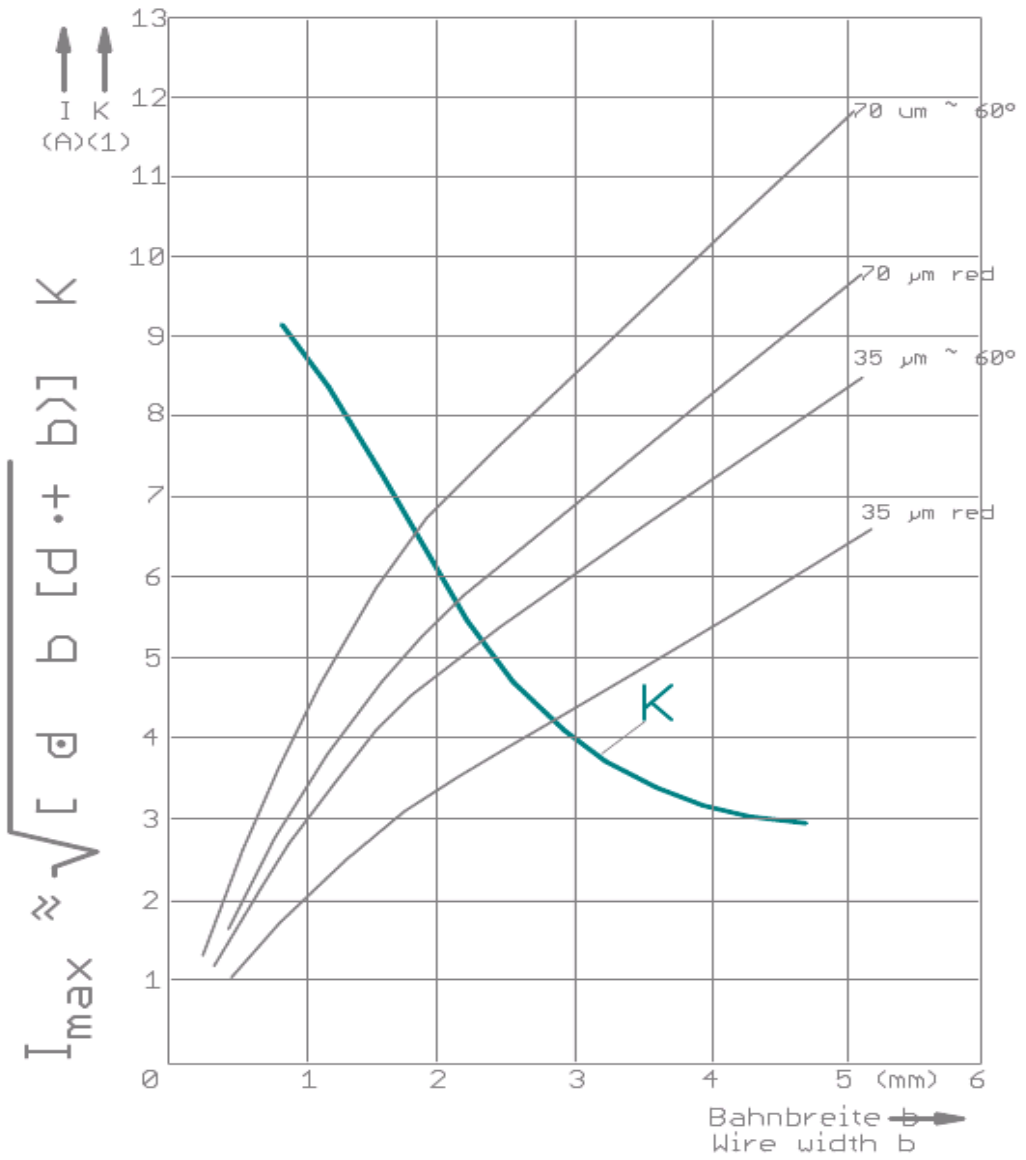


Diagram 2: maximum current load vs track width at various temperatures

Leiterbahnwiderstand

$$R = \frac{\rho \cdot l}{d \cdot b}$$

l [mm] = Länge der Leiterbahn

d [µm] = Stärke der Kaschierung

b [mm] = Breite der Leiterbahn

R [Ω] = Leiterbahnwiderstand

Metall	spez. Widerstand ρ bei 20 °C $\left[\frac{\Omega \cdot mm^2}{m}\right]$	Temperaturbeiwert α $[10^{-3} / °C]$
Kupfer	0,0174	4,33
Silber	0,0159	4,10
Gold	0,0224	4,0
Nickel	0,078	6,75
Zinn	0,123	4,6
Blei	0,208	3,8
Palladium	0,108	3,77
Rhodium	0,0454	4,43

Beispiel :

Leiterbahnlänge l = 50 mm

Kupferkaschierung d = 35 µm

Breite b = 1 mm

$$R = \frac{\rho \cdot l}{d \cdot b} = \frac{0{,}0174 \cdot 50}{35 \cdot 1} = 24{,}9\text{m}\,\Omega$$

Table 1: track/wire resistance calculation of various conducting metals

German	***English***
spez. Widerstand bei	specific resistance at
Temperaturbeiwert	temperature coefficient
Leiterbahnwiderstand	resistance of track/wire
Länge der Leiterbahn / Leiterbahnlänge	length of track/wire
Stärke der Kaschierung / Kupferkaschierung	thickness of coating / copper thickness
Breite der Leiterbahn	track/wire width
Kupfer (Cu)	copper
Silber (Ag)	silver
Zinn (Sn)	tin
Blei (Pb)	lead
Beispiel	example
Breite	width

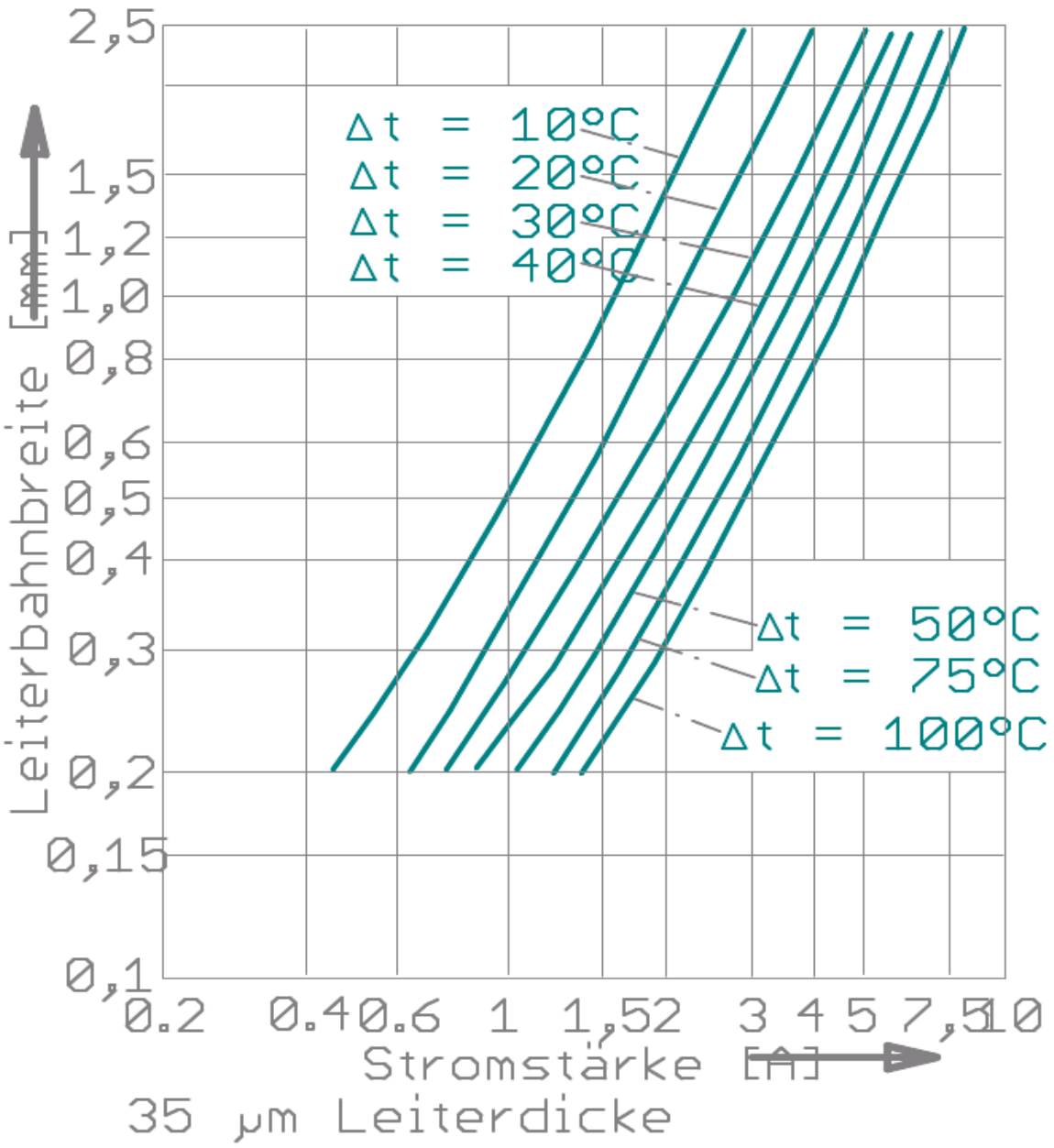


Diagram 3: maximum current load vs track width at various temperatures

German	***English***
Leiterbahnbreite	track/wire width
Stromstärke	(electrical) current
Leiterdicke	track/wire thickness

Isolationsabstand von Leiterbahnen
(USA MIL-Std. 275B)

A = normale Umweltbedingung
B = staubige/schmutzige Umgebung
Spannungswerte für Gleichspannung bzw. Spitzenwert der Wechselspannung

Spalte I ohne Schutzüberzug in Höhen von 0 ... 3048 m
Spalte II ohne Schutzüberzug in Höhen über 3048 m
Spalte III mit Schutzüberzug in Höhen von 0 ... 3048 m
Spalte IV mit Schutzüberzug in Höhen über 3048 m

Spannung [V]	IA	IB	II	III	IV
0 ... 50	0,381	2,032	0,660	0,381	0,559
51 ... 100			1,575		0,762
51 ... 150	0,660	2,032		0,559	
101 ... 170			3,17		1,524
151 ... 300	0,575	3,17		0,762	
171 ... 250			6,35		3,17
301 ... 500	3,17	6,62		1,524	
251 ... 500			12,70		6,35
> 500	0,0076 a Volt	0,0152 a Volt	0,025 a Volt	0,0051 a Volt	0,0127 a Volt

Die angegebenen Zahlen sind Mindestwerte in mm.

Table 2: clearances between tracks/wires

German	***English***
Isolationsabstand von Leiterbahnen	clearance between tracks/wires
normale / staubige / schmutzige Umweltbedingung	normal / dusty / dirty environment
Spannungswerte für Gleichspannung bzw. Spitzenwert der Wechselspannung	DC voltage or AC peak voltage
Spalte	column
ohne/mit Schutzüberzug in Höhen von/über x m	with/without protective coating at altitudes of/above x **Meters**

Spannung	voltage
Die angegebenen Zahlen sind Mindeswerte in mm.	Numbers given are minimal values in **Millimeters**.

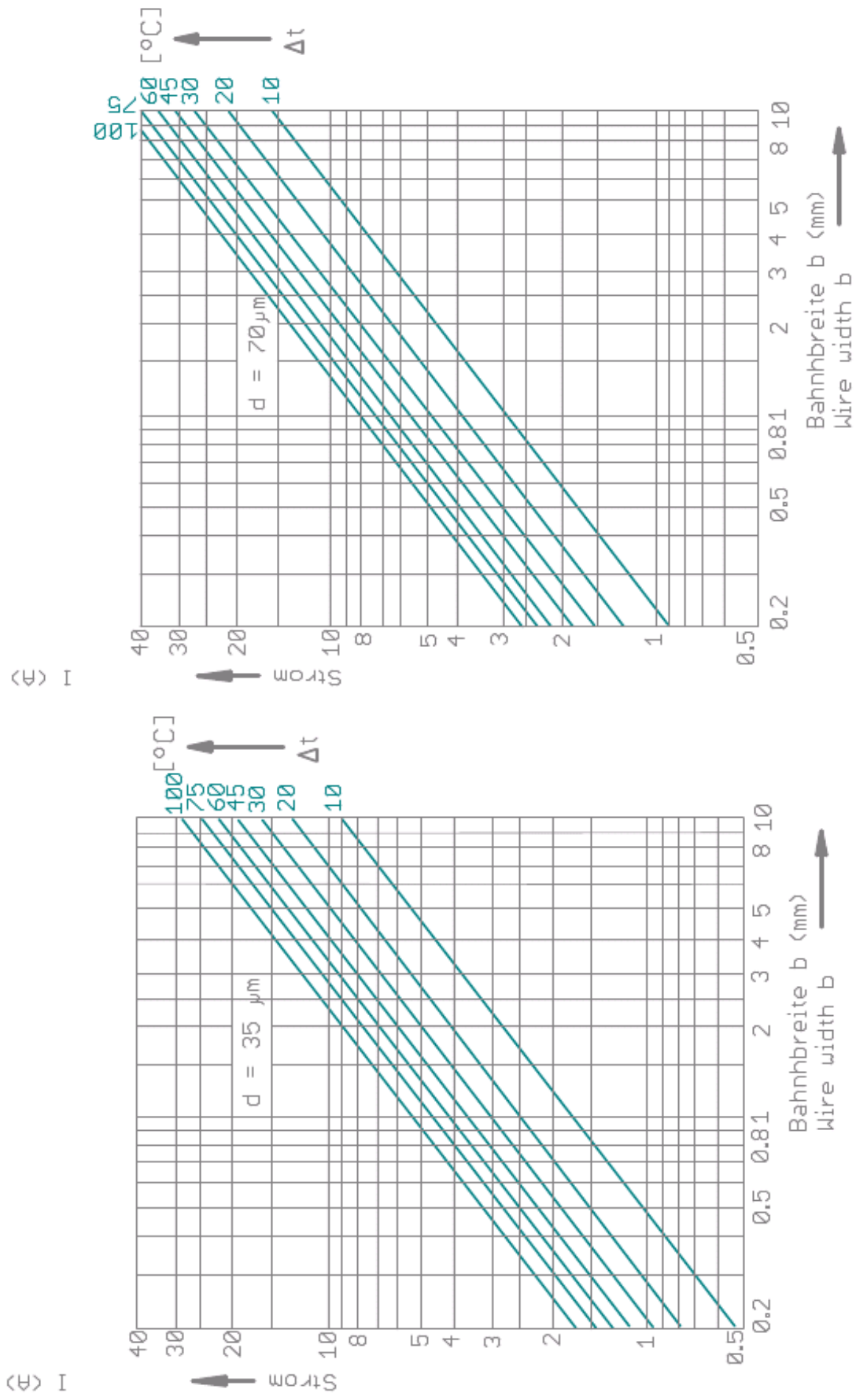
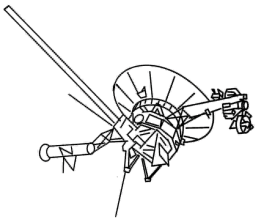


Diagram 4: maximum current load vs track width at various temperatures

5 Useful Links



- *CadSoft EAGLE Training* – a reasonable way to reasonable work at http://www.train-z.de
- Find updates of this checklist at http://www.train-z.de
- Debug SPI, I²C, Boundary Scan/JTAG and other hardware with the *Logic Scanner* at http://www.train-z.de/logic_scanner/index.html
- *EAGLE* - an affordable and very efficient schematics and layout tool at http://www.cadsoftusa.com/

- The office alternative : *OpenOffice* at http://www.openoffice.org

6 Disclaimer

This document is believed to be accurate and reliable. I do not assume responsibility for any errors which may appear in this document. I reserve the right to change it at any time without notice, and do not make any commitment to update the information contained herein.

My Boss is a Jewish Carpenter

Mario Blunk - electronics & IT engineering / Buchfinkenweg 5 / 99102 Erfurt / Germany +49 (0) 176 2904 5855 / http://www.train-z.de