Getting started in high performance electronic design

Wojtek Skulski
Department of Physics and Astronomy
University of Rochester
Rochester, NY 14627-0171

skulski _at_ pas.rochester.edu

First presented May/23/2002

Updated for the web July/03/2004

Getting started with
High performance electronic design

- 3-hour class
  - Designing high performance surface mount and multilayer boards.
  - What tools and resources are available?
  - How to get my design manufactured and assembled?
  - Board design with OrCAD Capture and Layout.
- When and where:
  - Thursday, May/23/2002, 9-12am, Bausch&Lomb room 106 (1st floor).
  - Slides updated for the web July/03/2004.
- Reserve your handout.
  - Send e-mail to skulski@pas.rochester.edu if you plan to attend.
  - Walk-ins are invited, but there may be no handouts if you do not register.
- See you there!

The goal and outline of this class

- Goal:
  - Describe the tools available to us for designing high performance electronic instruments.
- Outline
  - Why do we need surface mount and multilayer boards?
  - What tools and resources are available?
  - How to get my PCB manufactured?
  - How to get my board assembled?
  - Designing with OrCAD Capture and OrCAD Layout.
- The audience
  - You know the basics of electronics.
  - … and you need to get going quickly with your design.

Disclaimer

- I am describing tools and methods which work for me.
- I do not claim that this information is complete.
- My methods are not necessarily the best ones.

The key to high performance boards

High performance

=

surface mount technology

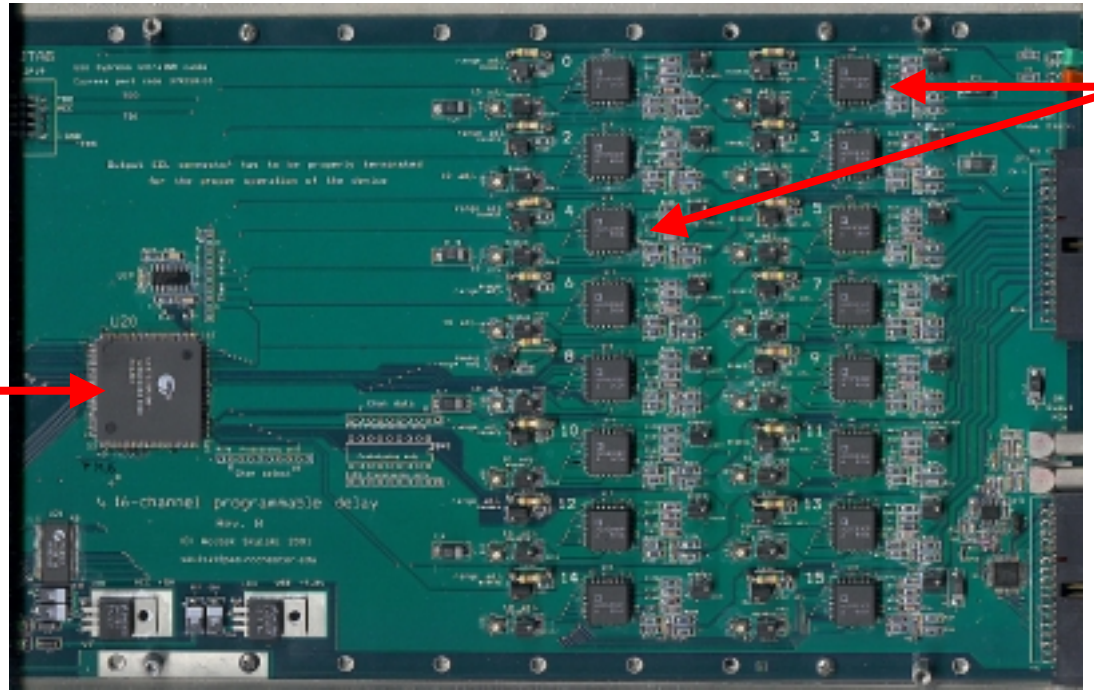
+

ground and power planes

A high-performance electronic board

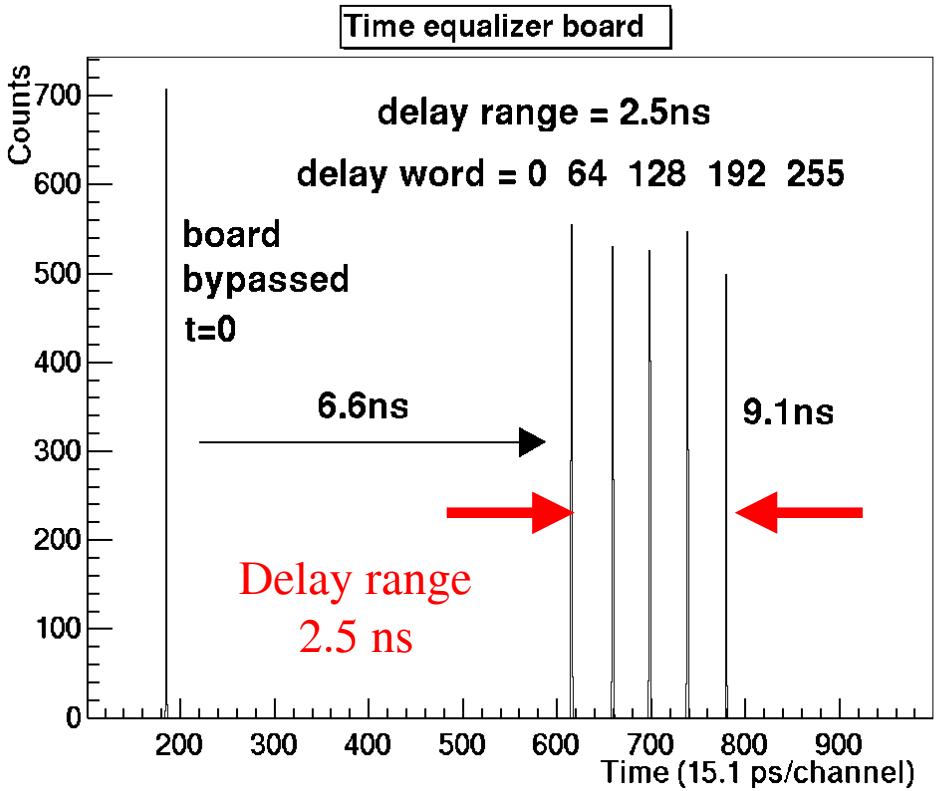
(I developed it using OrCAD)

- Surface mount technology (SMT) parts on both sides of the PCB.
- Two inner planes (ground and power) and two signal routing planes.
- Controlled impedance copper traces.
- Minimum trace width and spacing 7 mils.
- Solder mask (top and bottom) and silkscreen legend (top and bottom).



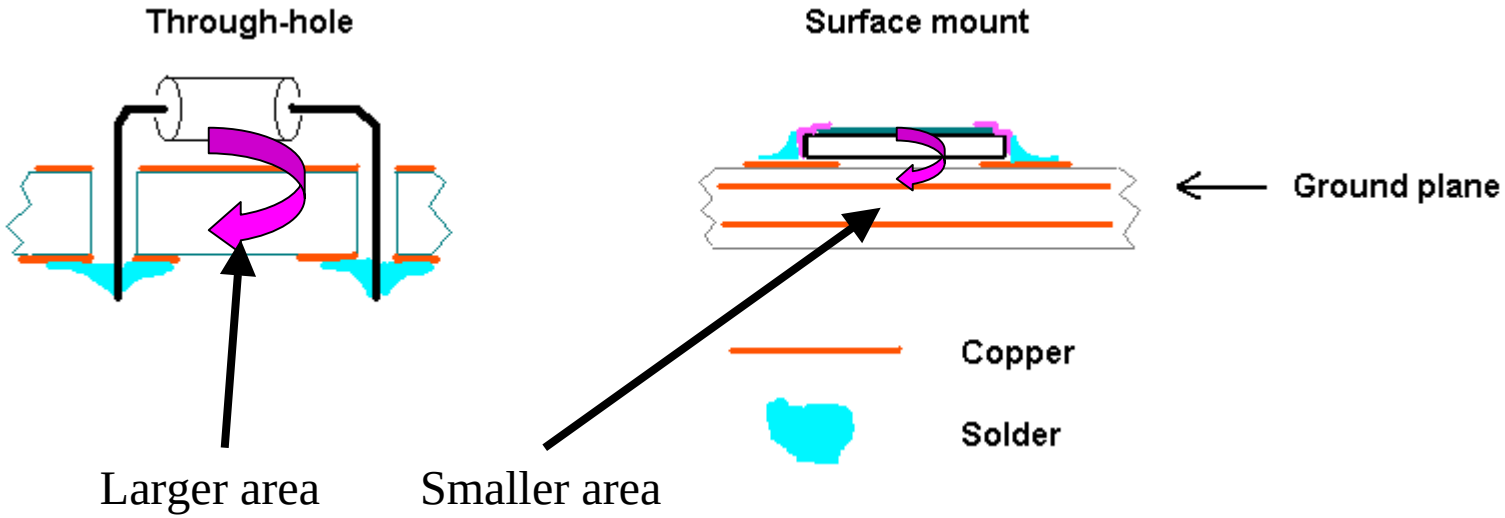
Good board => good performance

- The board contributes only 5 ps (RMS) to the width of the peaks.
- Light will travel only 1.5 mm during 5 ps!
- It cannot be done any better with this particular chipset.



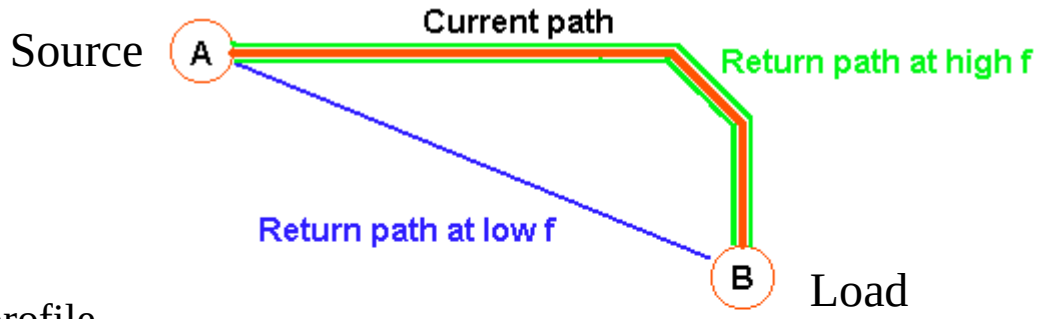
Why do we need surface mount?

- Self-inductance and mutual inductance are both proportional to the area of the circuit.
- The size of SMT parts is much smaller than the size of leaded parts.
- The area of the SMT circuitry is therefore smaller.
- The SMT circuit can handle higher frequencies than a traditional circuit can.



Why do we need ground and power planes?

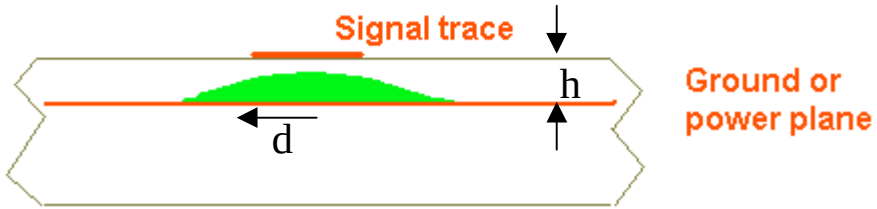
- At low frequencies the return current follows the path of least resistance.
- At high frequencies the return current follows the path of least inductance…
- … if there is one!
- Small area ➔ low inductance & low crosstalk ➔ we need ground and power planes.



Current density profile

$i(d) = 1/[1+(d/h)^2]$

See “Black magic” chapter 5



Resources

Literature
Software
Electronic parts
Manufacturing and assembly

Selected books and Application Notes

(sometimes they give conflicting advice!)

- Book: High Speed Digital Design: A Handbook of Black Magic, ISBN 0-13-395724-1.
- Analog Devices www.analog.com
  - Book: "Practical Analog Design Techniques", section 9 "Hardware Design Techniques"
  - AN-280 "Mixed Signal Circuit Techniques"
  - AN-333 "Design and layout of a video graphic system for reduced EMI"
- Texas Instruments www.ti.com
  - sloa089 "Circuit Board Layout Techniques"
  - sdya009c "Designing With Logic"
- Xilinx www.xilinx.com
  - Signal Integrity Central
  - UG002 "Virtex-II Platform FPGA Handbook" page 464 "PCB Considerations"
- ON-Semi (former Motorola Semiconductor Division) www.onsemi.com
  - AND8020/D "Termination of ECL Logic Devices"

Web resources

- Analog Devices Seminar Materials www.analog.com
- Texas Instruments Application Notes www.ti.com
- National Semiconductors, Linear Technology, On-Semi, etc.
- Jan Axelson's Lakeview Research
  - www.lvr.com - serial & parallel port programming, USB, assembly techniques, etc.
  - Entry-level to mid-professional information, very well presented.
- List of electronic design software
  - www.terrypin.dial.pipex.com/ECADList.html - case sensitive URL.
- Comprehensive EE websites
  - www.engr.unl.edu/ee/eeshop/netsites.html - many useful links.
  - www.epanorama.net - links to design software, soldering techniques, etc.

What design software is available to UofR users?

- OrCAD
  - Popular, proven, available on campus.
  - Not easy to learn, some quirks and bugs.
  - Powerful enough for me: up to 16 layers. I designed my boards with OrCAD.
  - Annual “small maintenance fee” shared among power users (EE, CS, Physics).
  - Websites: www.orcad.com, www.cadence.com
- If OrCAD is not enough for you
  - You can install other more powerful Cadence software from the same CD’s.
- If OrCAD is too complex for you, then try free ExpressPCB
  - Easy, you can learn it quickly.
  - Built-in part procurement info (DigiKey part numbers).
  - Results are limited: 2 or 4 layers, non-standard CAM output.
  - Aimed at hobby projects. Use it to design simple instruments.
  - Website www.ExpressPCB.com - both the software and PCB fabrication.

Other electronic design software

- Free design software for Linux/Unix/Windows, up to 8 layers, GERBER output
  - Buyer beware: this is free open source software. Hacking is required!
  - Schematic: xcircuit.ece.jhu.edu
  - PCB layout: pcb.ece.jhu.edu
- Eagle www.cadsoftusa.com
  - Professional, about $1k per user with University discount.
  - Free limited version is available (limited to only two layers and 10x8 cm).
- Protel www.protel.com
  - Professional, rather expensive, 30-day unlimited trial version available.
- Design software from DigiKey or Jameco.
  - Relatively cheap, I do not know the quality.
- Comprehensive list of electronic design software
  - www.terrypin.dial.pipex.com/ECADList.html (case sensitive URL)

Where to find parts and tools?

- DigiKey.com.
  - I order most parts from DigiKey (capacitors, resistors, etc.).
- Other distributors.
  - Avnet, Newark, Arrow, Nu Horizons, Jameco, ...
- Directly from manufactures.
  - Analog Devices, Texas Instruments, Cypress, Linear, National Semi, On-Semi, ...
- Part Miner can help locate a part.
  - Website with a search engine: www.FreeTradeZone.com.
- Sometimes I buy tools (but not parts) from Ebay.
  - I bought a DVM and an oscilloscope on Ebay.
  - Sometimes one can even buy OrCAD and other ECAD software.
  - Buying on Ebay requires patience.

How to have your PCB manufactured?

- Sierra Proto Express www.2justforyou.com
  - High quality, excellent promotions.
  - Very fast turnaround.
  - They accept CAM design files via internet, GERBER format.
  - They manufactured all my PCBs. I am happy with the result.

- PCBExpress
  - Similar to Sierra Proto Express.

- ExpressPCB
  - Only 2 or 4 layers.
  - You have to use their PCB design software, which is rather limited.
  - Suitable for simple projects or hobby.

- Other companies
  - I have no experience with other companies.
  - Website with many links: www.lvr.com/pcbs.htm

How to have your SMT board assembled?

- Do it yourself, it is not very difficult!
  - Good light, fine pincers.
  - Use 1206 footprints for your first SMT board. They are easy to handle.
  - Edroy Stereoscopic OpticAid +7D (2.75x magnification).
    - Edroy model 717. Distributor: www.scslimited.com, 1 (800) 749 8425.
  - 15W RadioShack grounded iron works fine.
    - The iron has to be either grounded or “ESD safe”. Avoid anything else.
  - Water-soluble flux is essential when soldering multipin chips!
  - There are a few other tricks with multipin chips (talk to me).
- Hildy Licht company does it routinely.
  - www.hildy.com

Designing with OrCAD

Schematic Capture
PCB Layout
Manufacturing

Find the bugs before they bite you

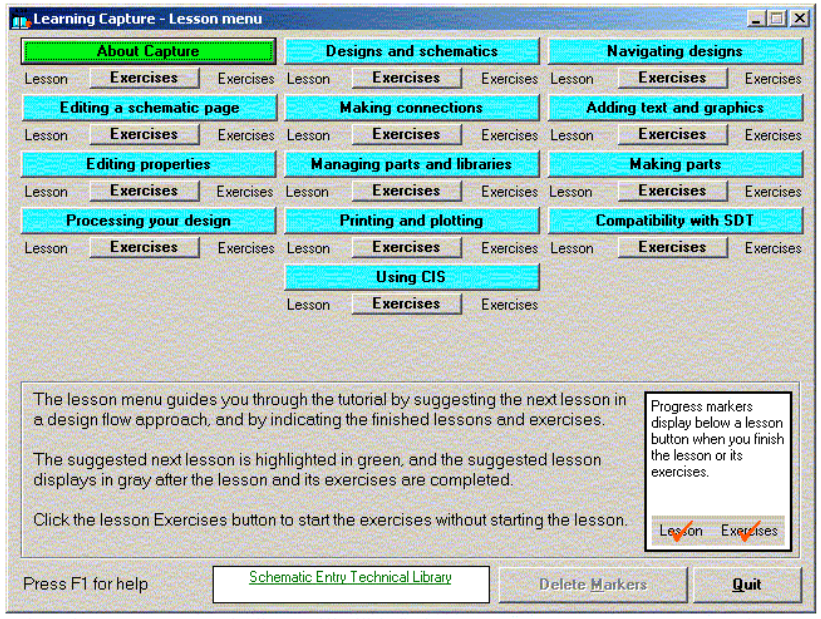
- Schematic drawings are more than just drawings.
  - Connections have to satisfy Design Rules.
  - Capture can automatically locate violations (Design Rules Check).
  - The netlist is passed to Layout only after all violations are corrected.
- PCBs are more than just drawings, too.
  - There can be accidental electric shorts and part-to-part spacing violations.
  - Layout can automatically locate such problems.
  - CAM files are created after correcting (or ignoring) all such violations.
- Before the board is manufactured.
  - CAM files need to be suitable for your manufacturer (check with them!).
  - Ideally: PSpice simulation using actual board layout (I have not done this).
- Ask (and pay) for the electrical check of your boards during manufacturing.

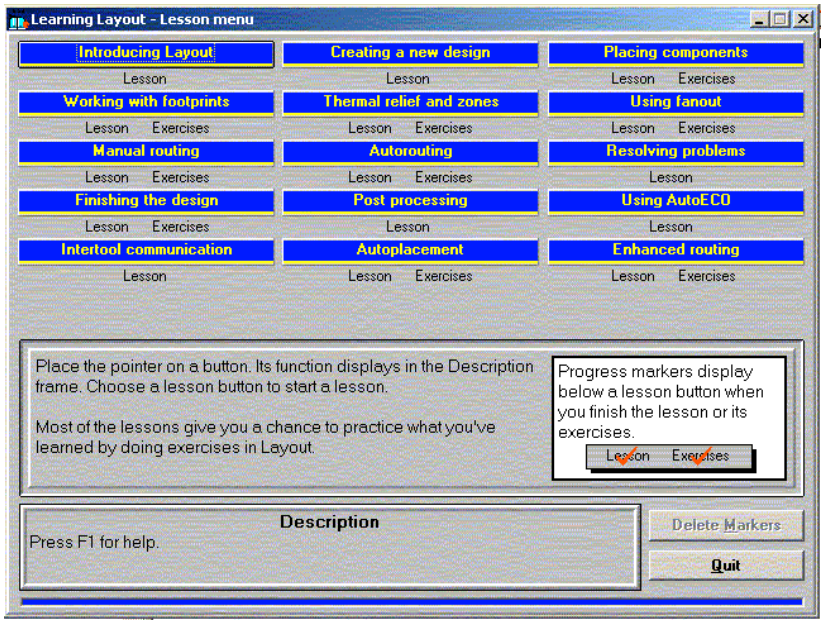
What to expect from the design software?

- Schematic design.
  - Convenient drawing capabilities.
  - Easy creation of new schematic symbols.
  - Automatic design rule checking.
  - Support for properties such as footprints, manufacturer part numbers, etc.
  - Netlist creation from schematics.
  - Access to schematic part database (but can live without it).
- Board layout.
  - Multi layer: at least 4 layers, possibly 10 layers or more.
  - Support for ground and power planes.
  - Narrow traces: down to 7 mils, possibly down to 4 mils.
  - Output files in a format suitable for manufacturing: GERBER format.
  - Automatic checking design rules, spacing violations, etc.
  - Good libraries of standard footprints.
  - Easy creation of new footprints.
  - Optional: a good autorouter can be very useful.
- Schematic $\Leftrightarrow$ layout integration (ECO, back and forth annotation).

Interactive Capture and Layout tutorials

- Capture tutorial directory …/Cadence/Capture/Tutorial
- Layout tutorial directory …/Cadence/Layout_Plus/Tutorial
- …but you will also need printed manuals.

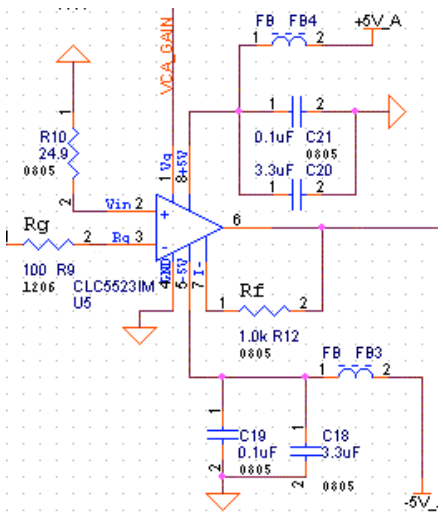




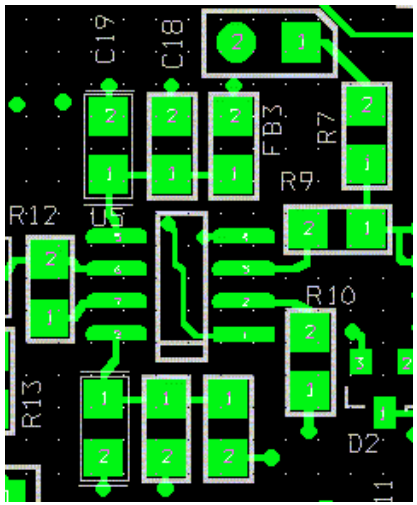
Design flow: schematic, layout, actual board

- Software has to manage the consistency between the schematic and the layout.
- Photoplotter CAM files are used to manufacture the physical board.

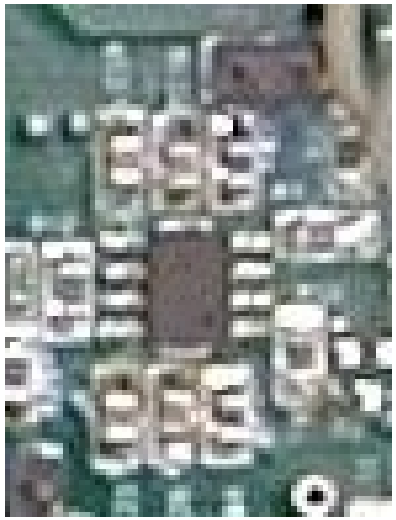
Schematic Capture



PCB Layout



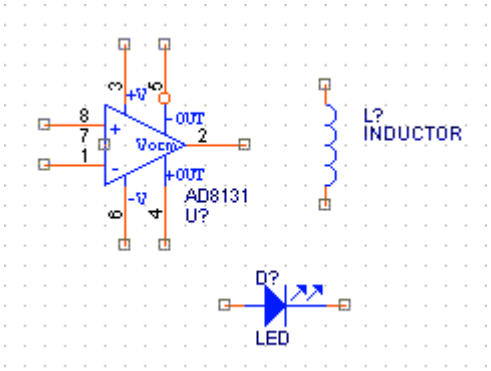
Actual board



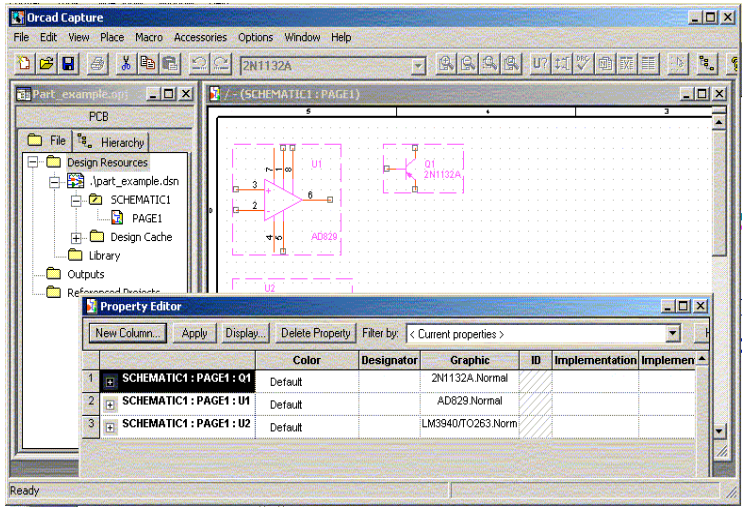
Schematic symbols are the building blocks

- Physical parts are represented by “schematic symbols”, which have properties.
- Visible properties: value (e.g., 5 kohms), schematic reference (e.g., R11).
- Invisible properties: footprint (1206, 0805, etc), distributor number (P24.9CCT-ND).
- Invisible properties are essential both for Layout and for the procurement.
- I used to manage invisible properties “by hand” using the property editor.

Symbols “as seen” in Capture



Managing properties “by hand”

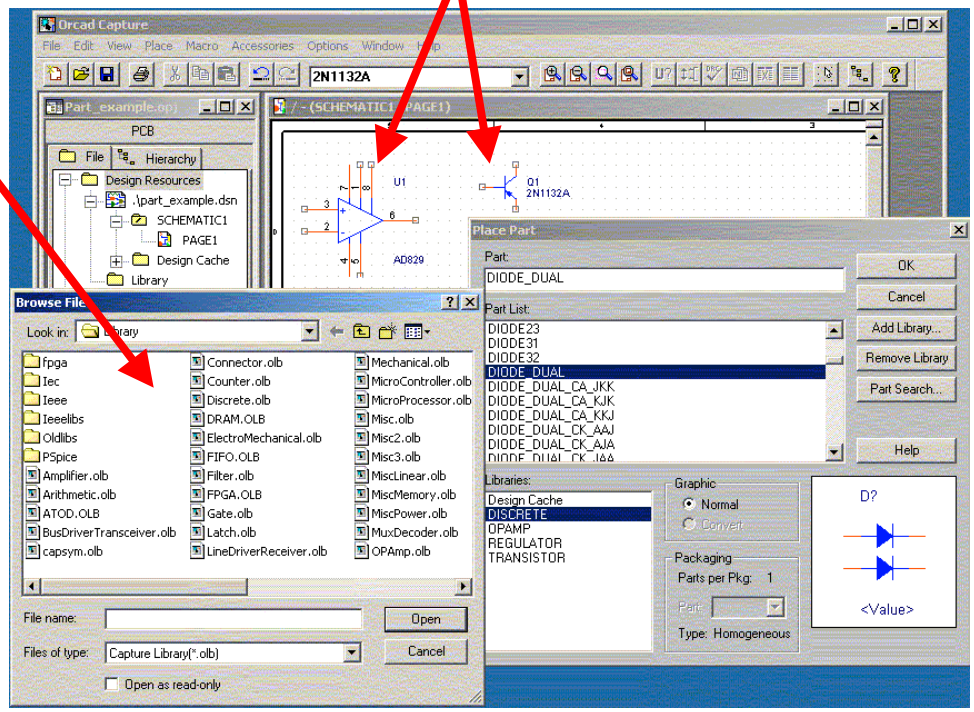


Capture provides many schematic symbols

- Schematic symbols can be picked from libraries.
- NB: distributor part numbers of actual parts are not provided in the libraries.

List of available libraries

Symbols already placed on a schematic page



Where to find actual parts and their “properties”?

- I mostly use the hardcopy DigiKey catalog.
- Their distributor part code looks something like P24.9CCT-ND.
- I type this code as an “invisible user property” named PARTNO.
- I print the PARTNO with the bill-of-materials (BOM).

A page from the DigiKey catalog

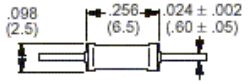
1% Metal Film Fixed Resistors — .40 Watt 5043ED Series

• Solder plated
• Copper leads

The resistors are coated with a light green lacquer which provides electrical, mechanical, and climatic protection. The encapsulation is resistant to all cleaning solvents in accordance with MIL-STD-202E, Method 215, and IEC 68-2-45.

CHARACTERISTICS:

• **Resistance Tolerance:** ±1% • **Temp.Coefficient:** ≤ ±100 ppm/°C • **Abs. Maximum Dissipation at TAMB = 70°C:** 0.40W • **Thermal Resistance, RTH:** 200°C/W • **Maximum Continuous Oper. Volt.:** 250 V (DC or RMS) • **Noise R ≤ 1 MΩ:** Maximum 0.1 µV/V • **Operating Temperature Range:** -55°C to +155°C



Description	Value Range	Digi-Key Part No.	Pricing 5	200	1,000	5,000	10,000	BC Components Part No.
.40 Watt, 1% Metal Film Fixed Resistor; Tape & Reel (5,000/Reel)	10.0 - 2.00M Ω	BC + Value + "YTR-ND"	—	—	—	22.50/M	19.20/M	5043EDxxxxxF12AF5
.40 Watt, 1% Metal Film Fixed Resistor; Tape	10.0 - 2.00M Ω	BC + Value + "YCT-ND"	.95	14.50	37.50	—	—	5043EDxxxxxF12AF5

1% Metal Film Fixed Resistor Kits
Digi-Key® Assortment Kits

5 Each of Values 10.0Ω to 97.6Ω.
BCY1-KIT-ND **$34.95**

5 Each of Values 100Ω to 976Ω.
BCY2-KIT-ND **$34.95**

5 Each of Values 1.00KΩ to 9.76KΩ.
BCY3-KIT-ND **$34.95**

5 Each of Values 10.0KΩ to 97.6KΩ.
BCY4-KIT-ND **$34.95**

5 Each of Values 100KΩ to 2.00MΩ.
BCY5-KIT-ND **$46.95**

BC Components Standard Resistor Values

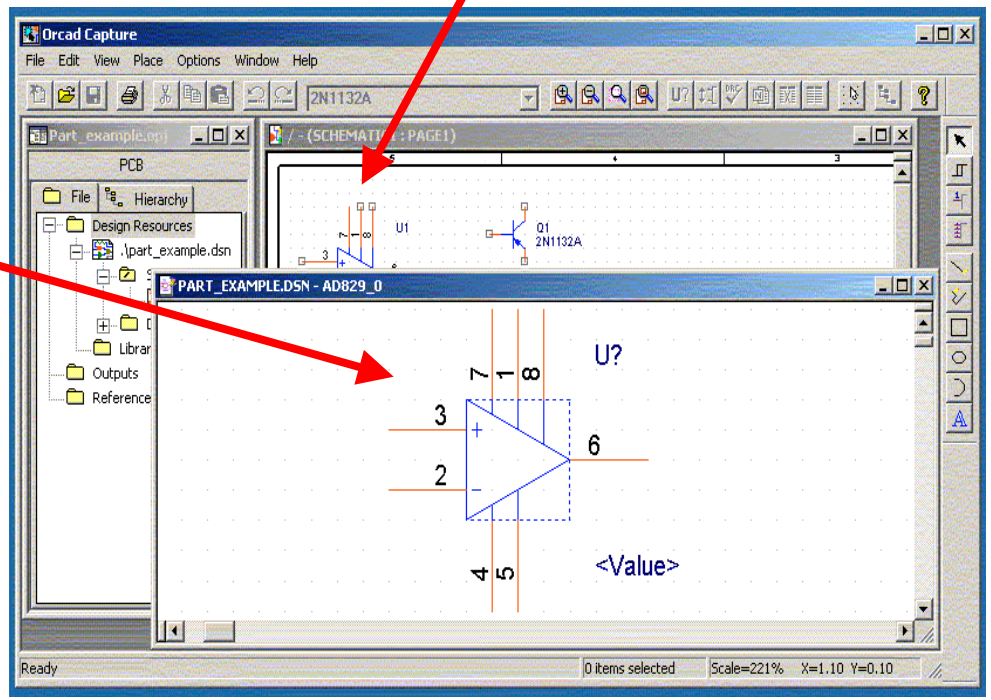
10.0	18.2	34.0	62.0	113	210	383	698	1.30K	2.43K	4.53K	8.25K	15.4K	28.7K	52.3K	95.3K	174K	316K	590K	1.07M	1.91M
10.2	18.7	34.8	63.4	115	215	392	715	1.33K	2.49K	4.64K	8.45K	15.8K	29.4K	53.6K	97.6K	178K	324K	604K	1.10M	1.96M
10.5	19.1	35.7	64.9	118	220	402	732	1.37K	2.55K	4.75K	8.66K	16.2K	30.1K	54.9K	100K	182K	332K	619K	1.13M	2.00M
10.7	19.6	36.5	66.5	121	221	412	750	1.40K	2.61K	4.87K	8.87K	16.5K	30.9K	56.0K	102K	187K	340K	634K	1.15M	

Often Capture does not provide the symbol that you need…

- For example, Analog Devices opamp type AD830 is not there.
- I have to make the symbol for it.

Symbol which was provided: AD829

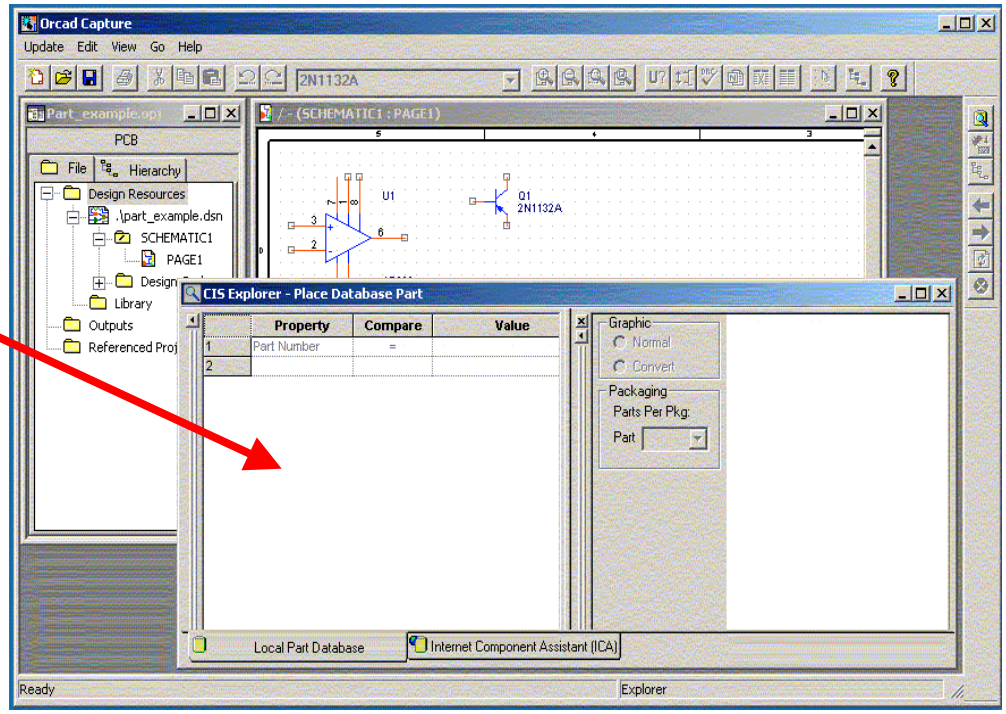
Here I am constructing a new symbol AD829_0, which I will later rename to AD830 which I need.



CaptureCIS provides tools for managing the part database

- CIS means “component information system”, which is a terrific idea.
- It would be very helpful for managing the actual part database.
- I have not used it yet.

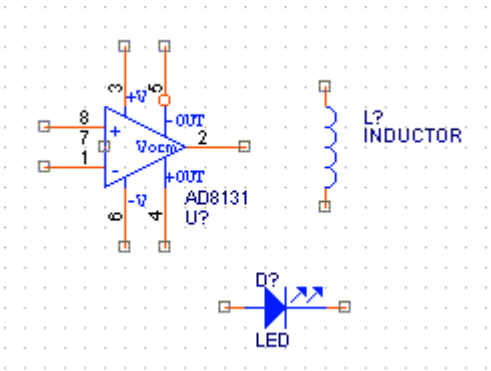
CIS access window,
unfortunately empty.
(I have not used it yet.)



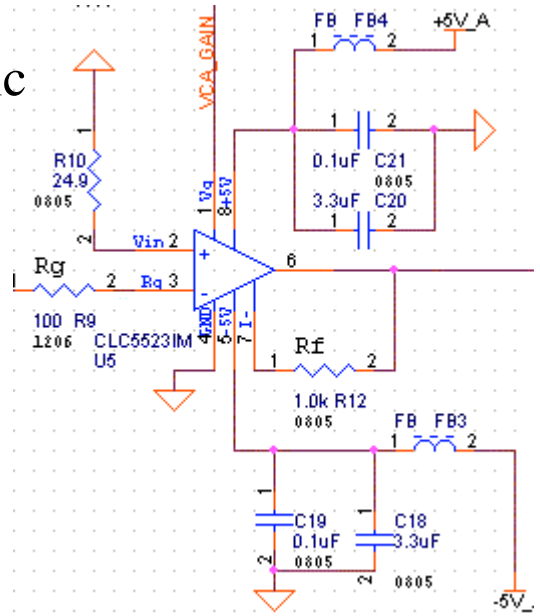
Symbols + connections = schematic

- In addition to properties, symbols also have pins.
- Pins are connected with wires in order to form a netlist.
- The netlist turns a loose collection of parts into the schematic.
- The netlist can be written to disk. Layout will read it from there.
- The PCB layout program will turn connections into copper traces.

Individual symbols



Schematic

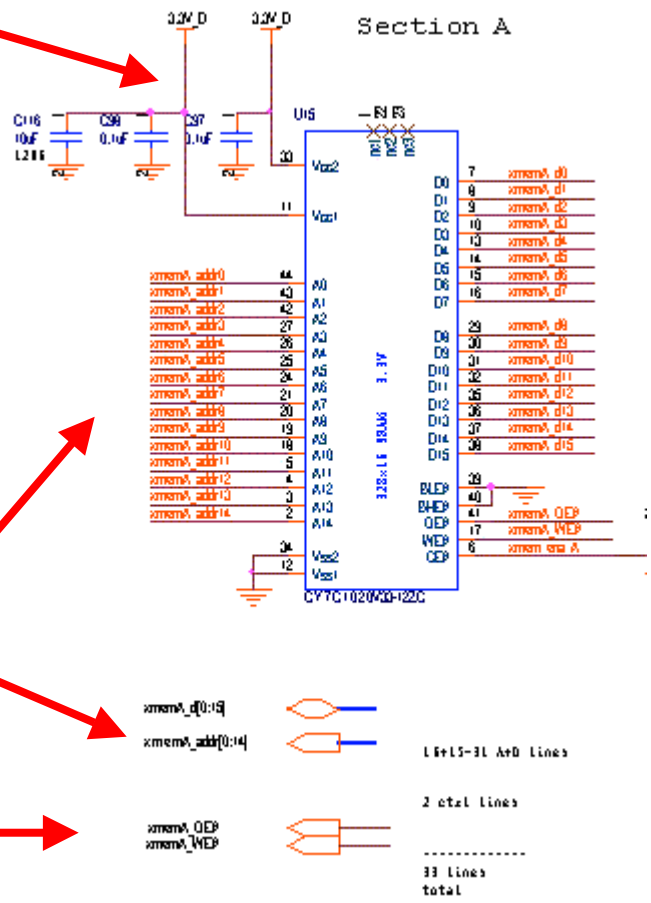


Connections do not have to be drawn explicitly

- Connection is either explicit (drawing a wire) or implied (two wires with the same names are assumed to be connected).
- As you can see, many wires are not drawn, but rather implied by name. For example, all the memory wires labeled in red red are implicitly connected to the “off page port” with the same name.
- It is a matter of style.

Wires are
connected to this port

Other off-page “ports”

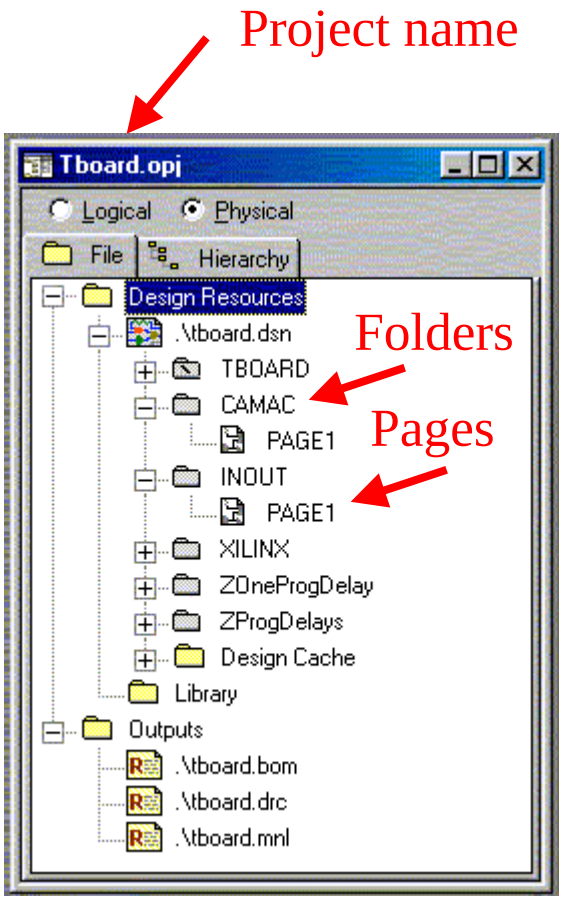


A large schematic becomes a “project”

- A small schematic can fit onto one page.
- But a large one will take several pages.
- We need to organize many such pages into one coherent schematic.

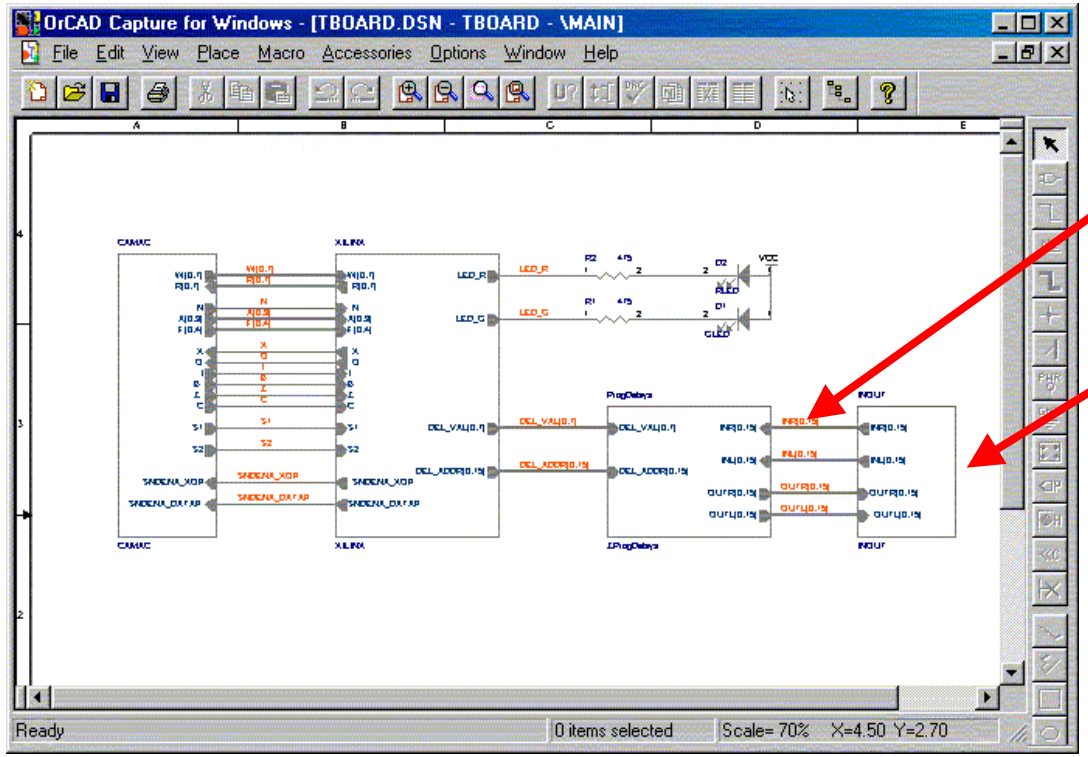
- Many pages put together form a project.
- Project is divided into folders.
- Every folder contains schematic pages.
- Wires and busses extend between pages.
- The top level page connects all other pages.

- There can be no unconnected wires or pins.
- Automatic *Design Rules Check* enforces this, and many other things.



The top-level schematic page

- The top level page contains all other pages.
- Wires and busses connect pages together.



Connections between pages

Individual pages

What did we learn thus far?

- Schematic is more than just a drawing: a lot is going on behind the scenes.
- Parts = symbols + properties.
- Schematic = symbols + connections.
- A connection can be either explicit (drawing a line) or implied by names.
- A large schematic is broken into hierarchical parts (folders and pages).
- Connections between hierarchical parts tie the whole design together.
- There are *Design Rules* that one must obey. Capture can check the rules.
- Capture can write the design to disk in a format, that Layout will understand.
- There are many Capture features that I did not touch upon. My guess is that you do not need these features to design your first few boards.
- In addition to OrCAD, there is other design software available to us. Make your own choice in case OrCAD offers too much or too little.

Layout: twelve steps to perfect boards

- 1: Choose and/or prepare footprints.
- 2-3: Place components.
- 4-5: Route connections.
- 6-10: Examine and clean up the layout.
- 11-12: Prepare and submit photoplotter files.

Example board: 445 components, 1298 connections

- Mixed signal: digital circuits and and low-noise analog on the same board.
- Digital clock frequency up to 65 MHz.
- Design goal: analog noise level no more than about 10^{-3} V
- I will use this example to walk you through the design process.

Before you start: check manufacturing details

- Look at the PCB manufacturer's website, learn their specs.

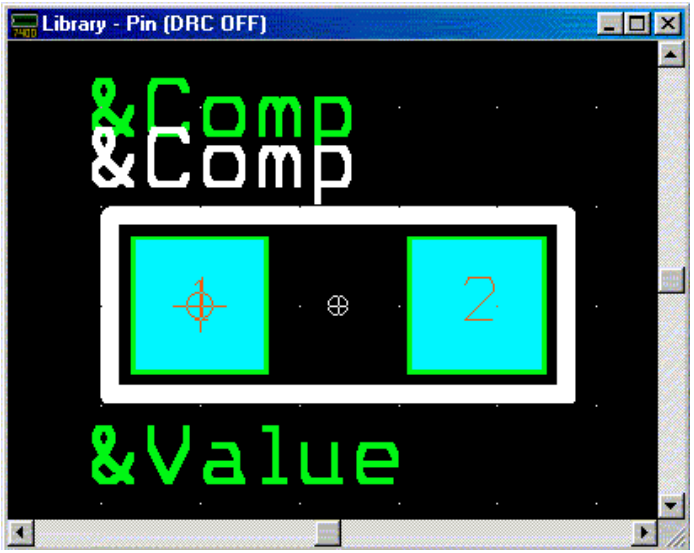
- What is the min/max size of holes?
- Minimum PCB trace width and separation?
- Minimum clearances?
- Maximum board size?
- Limitations concerning the board shape?
- Required file format? (Most likely GERBER, but check.)
- Etc.

- Prepare your "board template" according to the specs.

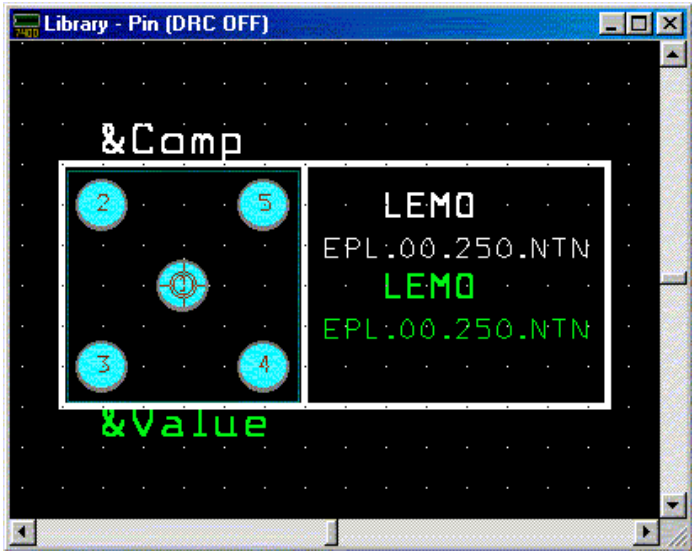
Layout step 1: prepare the footprints

- Choose footprints from one of many Layout libraries.
- Some footprints need to be custom made.
- It is a good idea to cleanup footprint labels at this step.

Standard footprint
1206-size component



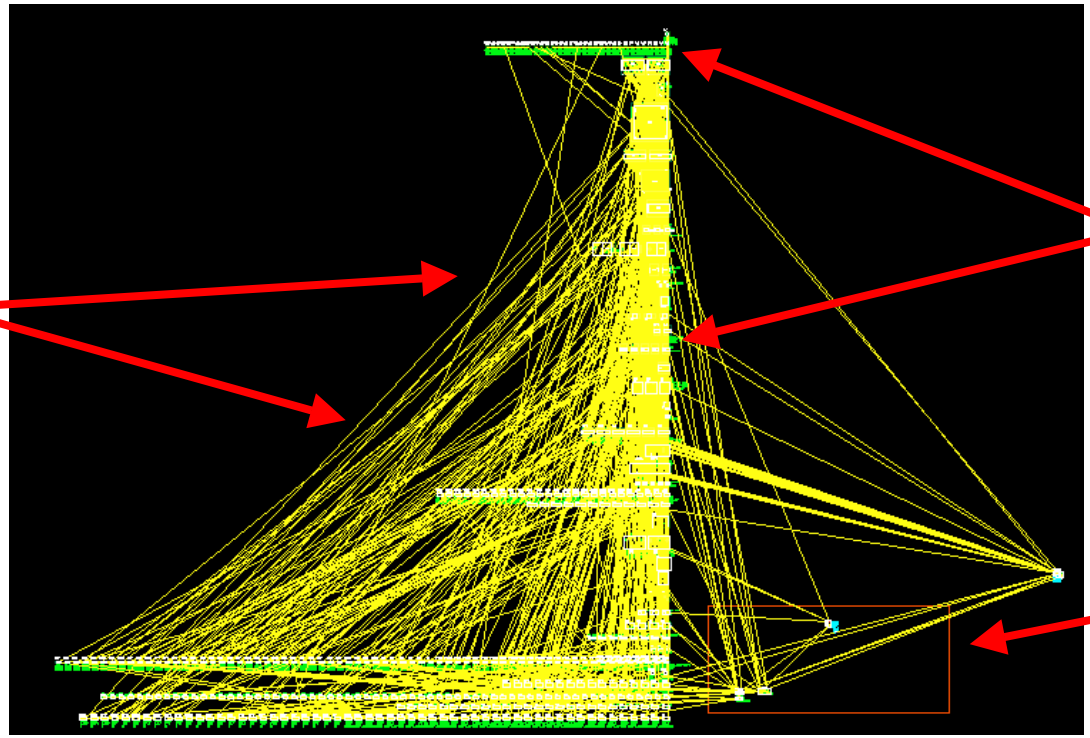
Nonstandard footprint
LEMO socket



Layout step 2: Netlist + board template = ratsnest

- Layout will take an empty board template and the netlist from Capture.
- You will see a somewhat depressing picture showed below.

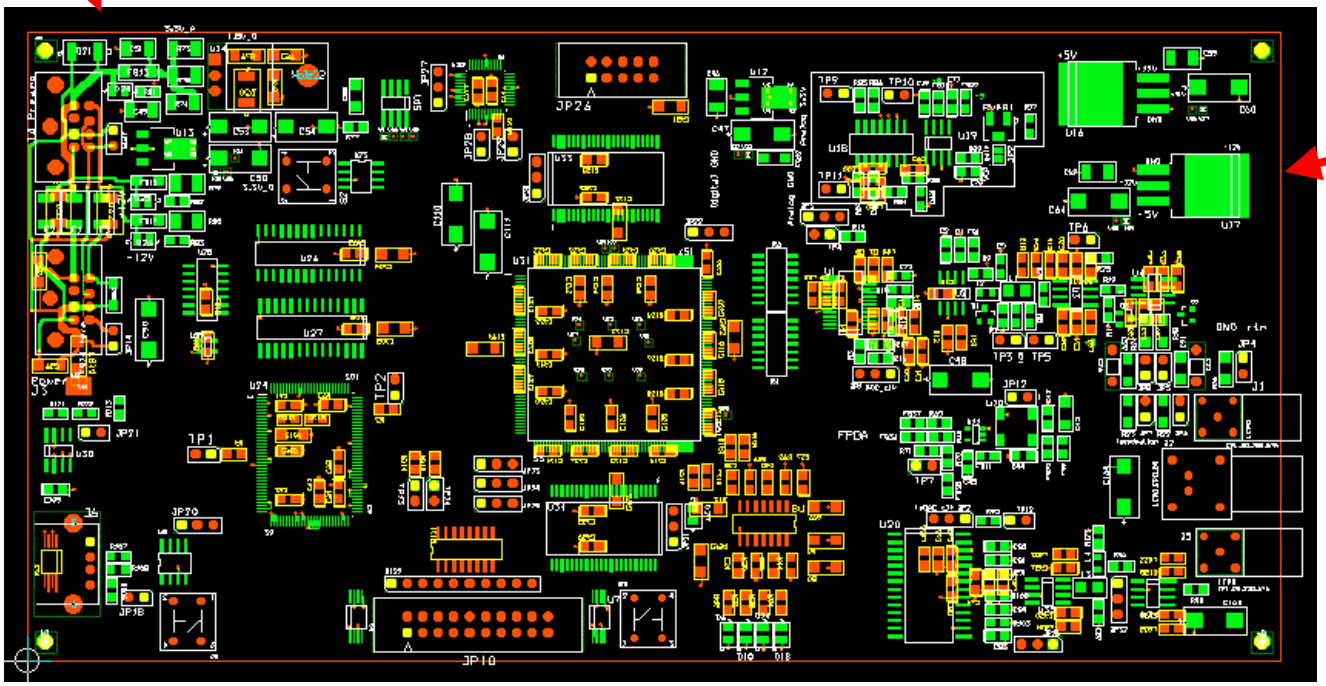
Ratsnest



Layout step 3: placing the components

- Green: top side, red: bottom side of the board

A few routed connections

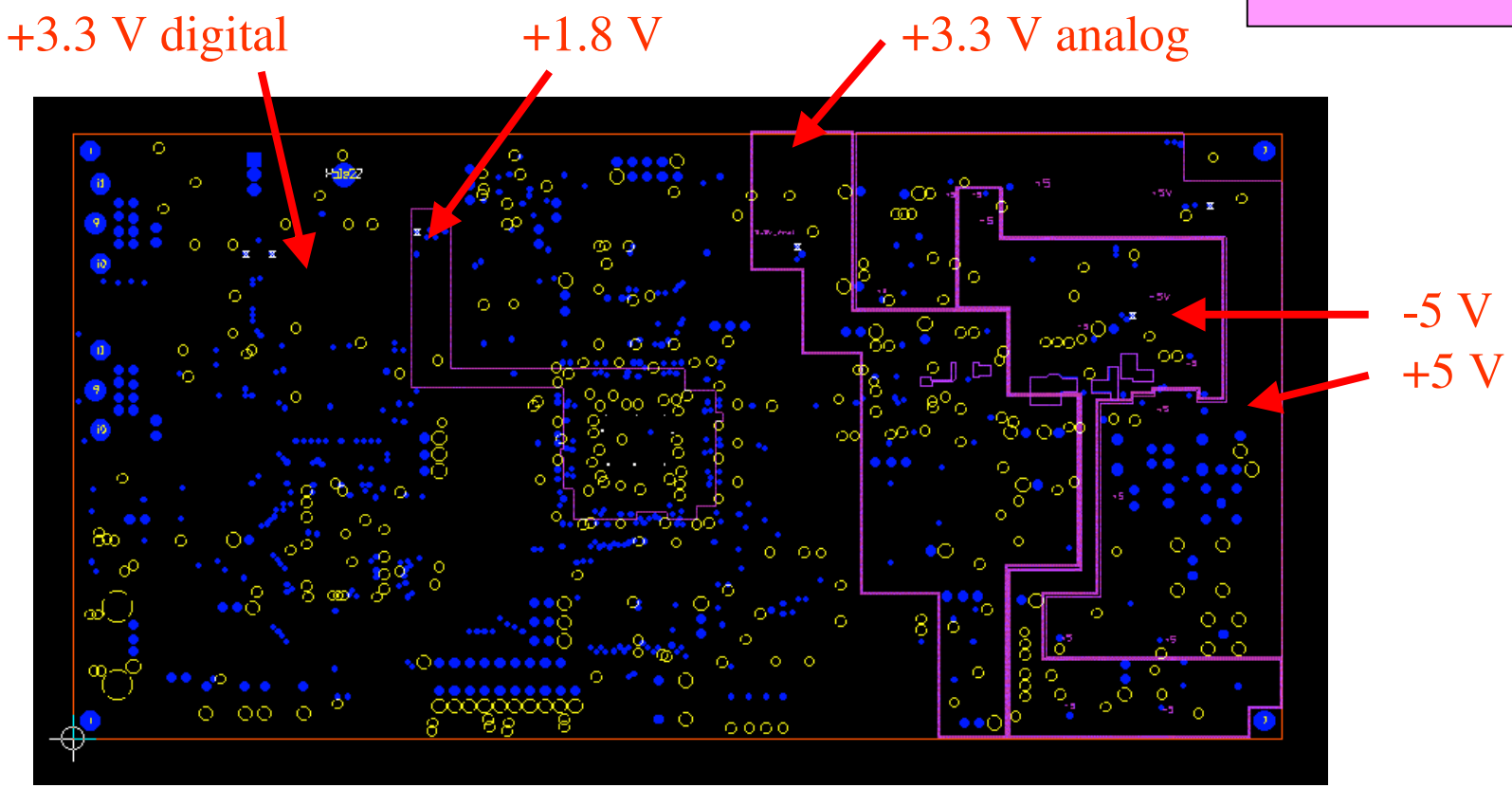


Board outline

Layout step 4: ground and power distribution

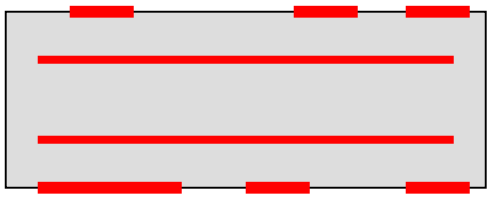
- Blue: ground plane, violet: power plane (in negative)
- Yellow: connections to the planes with “thermal reliefs”

Dark is copper
light is etched



Ground and power distribution cont.

- The number of plane layers must be even. The planes have to be symmetric. Otherwise the board will warp.
  - Example: one GND and one PWR plane.

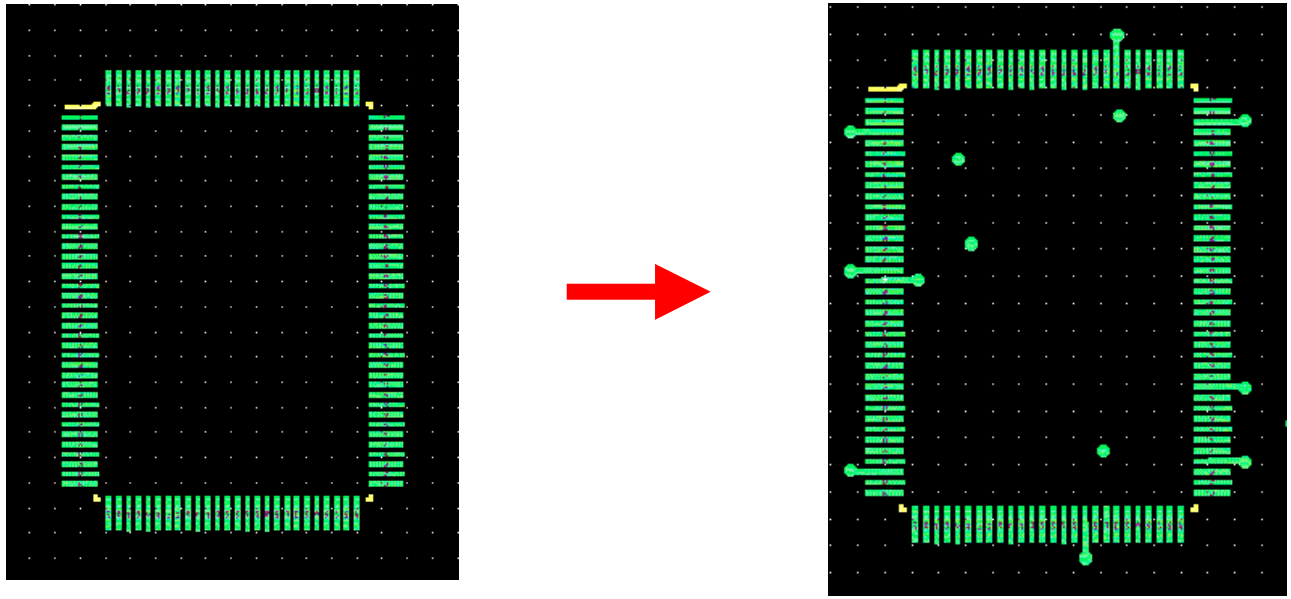


- The PWR plane can be partitioned if there is more than one power source.
  - Example: +/- 5V, 1.8V and 3.3V digital, and 3.3V analog are all on the same plane.

- GND plane can be partitioned between the analog and the digital grounds.
  - If you do not feel comfortable doing it, don't! A solid ground plane may be the best.
  - Consult Analog Devices and Texas Instruments Application Notes for details.
  - Read "Black Magic" by Howard & Johnson.

Ground and power distribution cont.

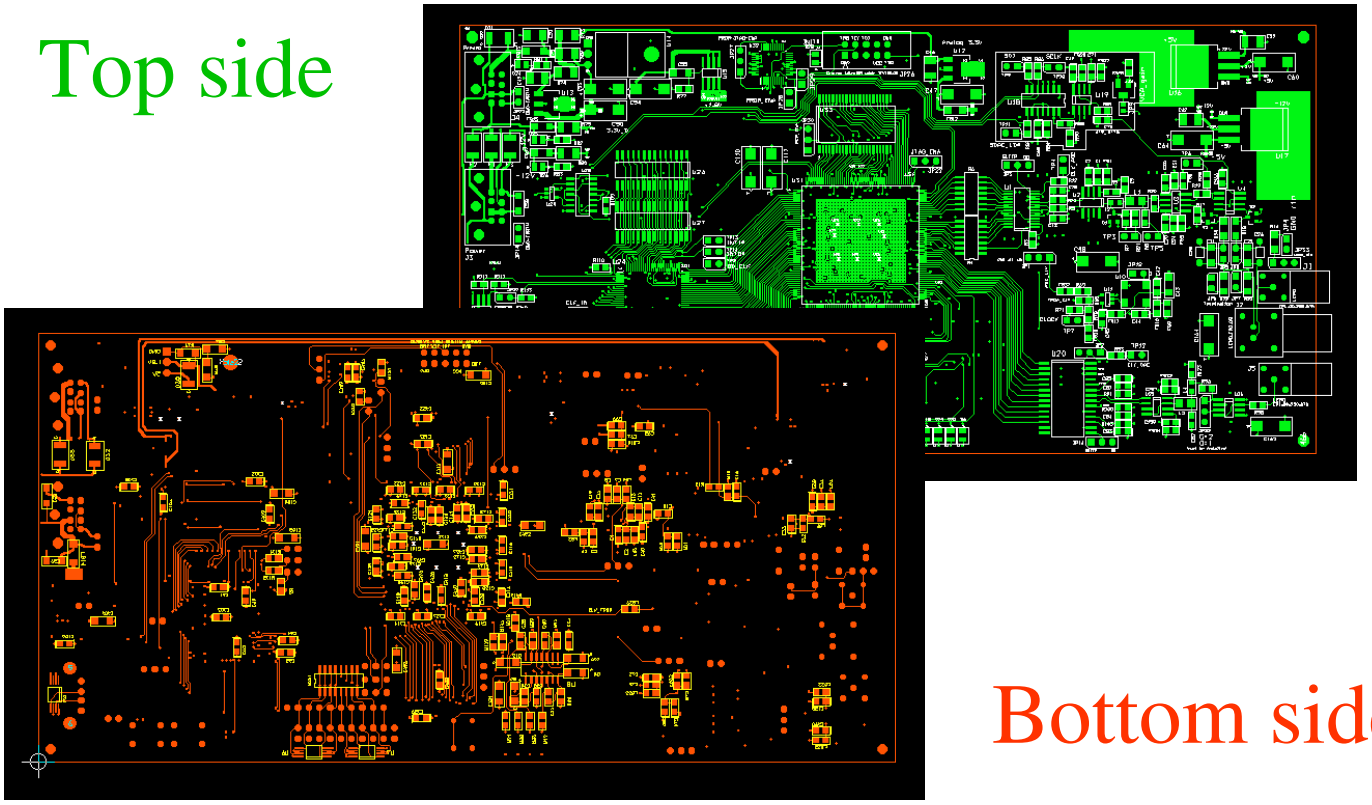
- By definition, SMT components imply no holes. The connections to the GND and PWR planes need to be established. It can be done either automatically or manually.
  - This step is called “fanout” in OrCAD-speak.



Layout step 5: routing the connections

- Routing connections “by hand” can yield good boards, but it is tedious.
- I routed my boards entirely by hand.

Top side

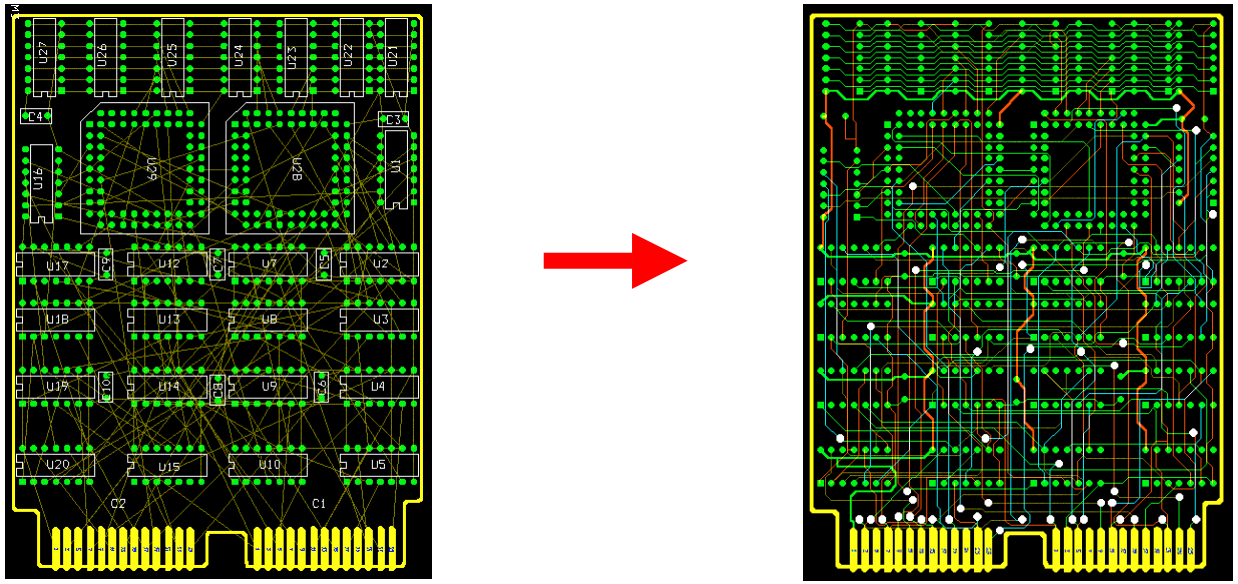


Bottom side

Routing the connections with an autorouter

- The autorouter may even work :-).
- You need to fine tune parameters such as priority of nets to be routed.
- Some manual cleanup is usually necessary.
- Digital boards can be autorouted.
- Analog boards are better suitable for hand routing.

Example: 6 layers, 35 components, 286 connections, 3 minutes on Pentium 160 MHz.



Autorouter, continued

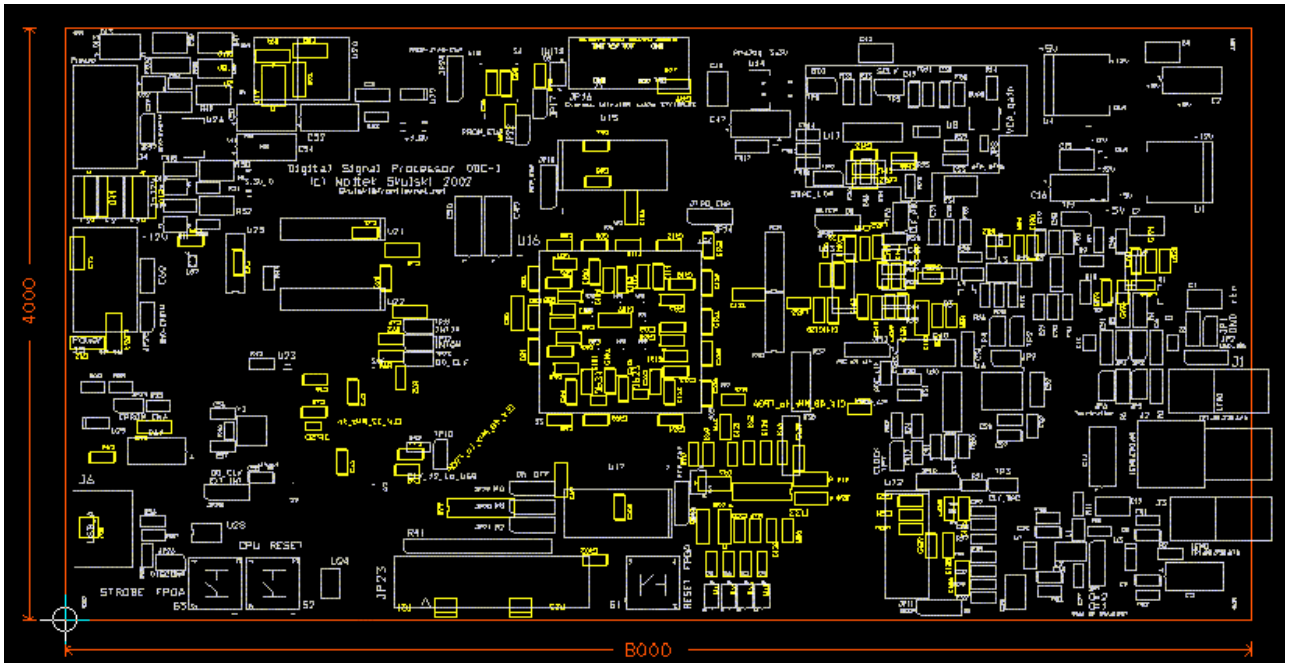
Layout Plus has two autorouters: a “trial-and-error” router that is very slow, and a “SmartRoute” that uses simulated annealing. SmartRoute can route a digital board pretty fast. However, when it comes to an analog or mixed-signal boards, I am routing them by hand.

Autorouting is an area of intense competition among ECAD companies. Autorouters keep improving, but do not expect miracles if you are laying out an analog board.

Layout step 6: cleaning up the silkscreens

- Silkscreen tells you what is what on the board. It is very useful.
- Grey: top, yellow: bottom.

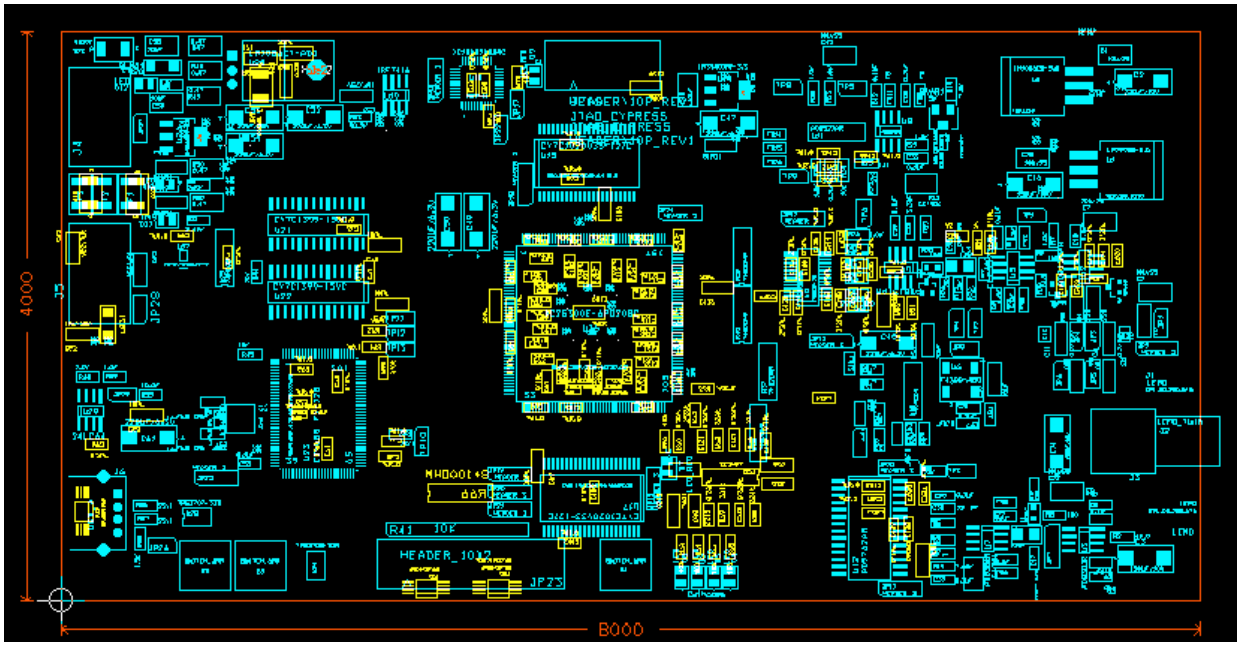
- NB: on the display the default Layout top color is white, not good.
- Change it to grey.



Layout step 7: cleaning up assembly drawings

- You need assembly drawings if someone else is assembling the board.
- Blue: top, yellow: bottom.

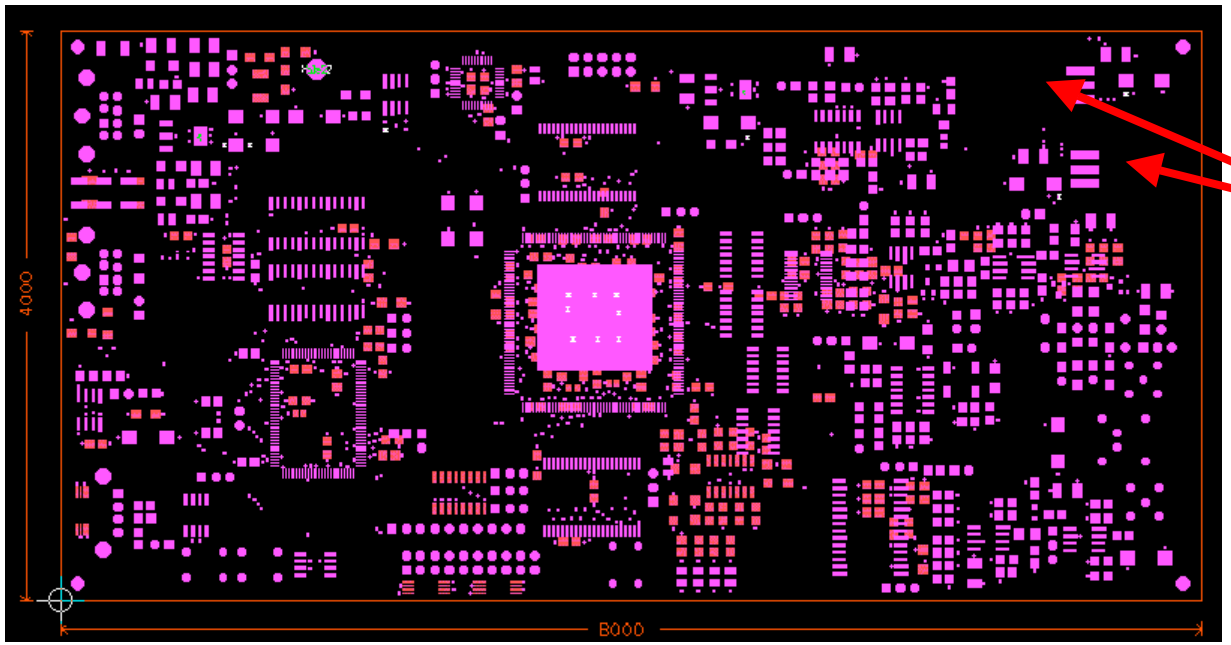
- By default, Layout makes extremely messy silkscreens and assembly drawings.
- NB: this drawing was mostly cleaned, but not quite.



Layout step 8: cleaning up solder masks

- Solder masks cover everything except places where you solder.
- Violet: top, red: bottom.

- Check it carefully, otherwise you will have to scrub the board.
- It happened to me, look at the arrows!

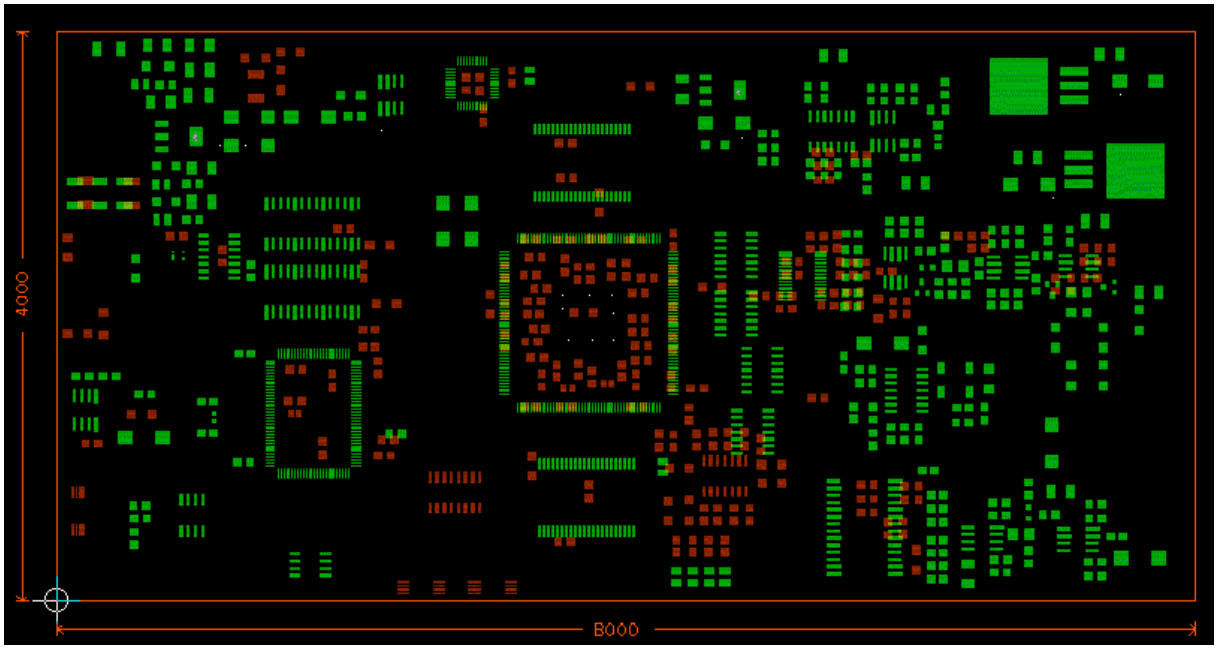


Two openings
are missing
in these spots

Layout step 9: cleaning up solder paste masks

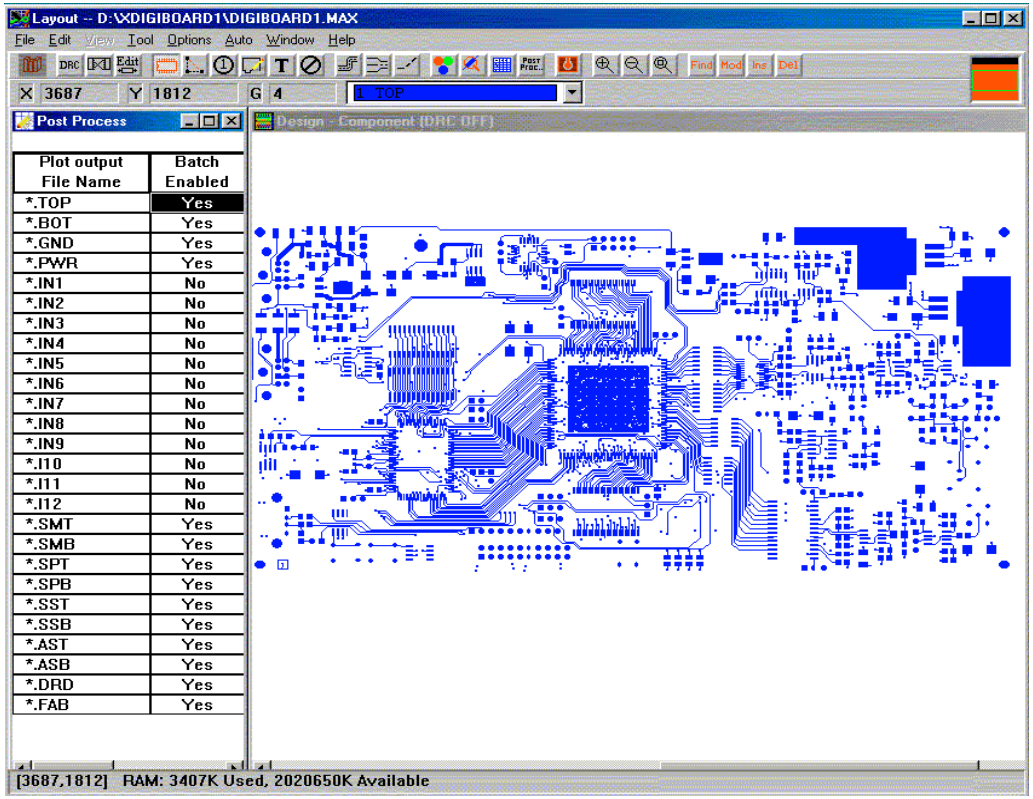
- Green: top, brown: bottom.

- Only needed for automated assembly.
- Not needed for hand assembly.



Layout step 10: examine photoplotter output

- You will get what you send to the foundry!
- Preview the rendition of the layers. This is your last chance to catch mistakes.
- Watch for any mishaps (e.g., several close vias form a slit in the GND plane).



Plot output File Name	Batch Enabled
*.TOP	Yes
*.BOT	Yes
*.GND	Yes
*.PWR	Yes
*.IN1	No
*.IN2	No
*.IN3	No
*.IN4	No
*.IN5	No
*.IN6	No
*.IN7	No
*.IN8	No
*.IN9	No
*.I10	No
*.I11	No
*.I12	No
*.SMT	Yes
*.SMB	Yes
*.SPT	Yes
*.SPB	Yes
*.SST	Yes
*.SSB	Yes
*.AST	Yes
*.ASB	Yes
*.DRD	Yes
*.FAB	Yes

Layout step 11: create manufacturing files

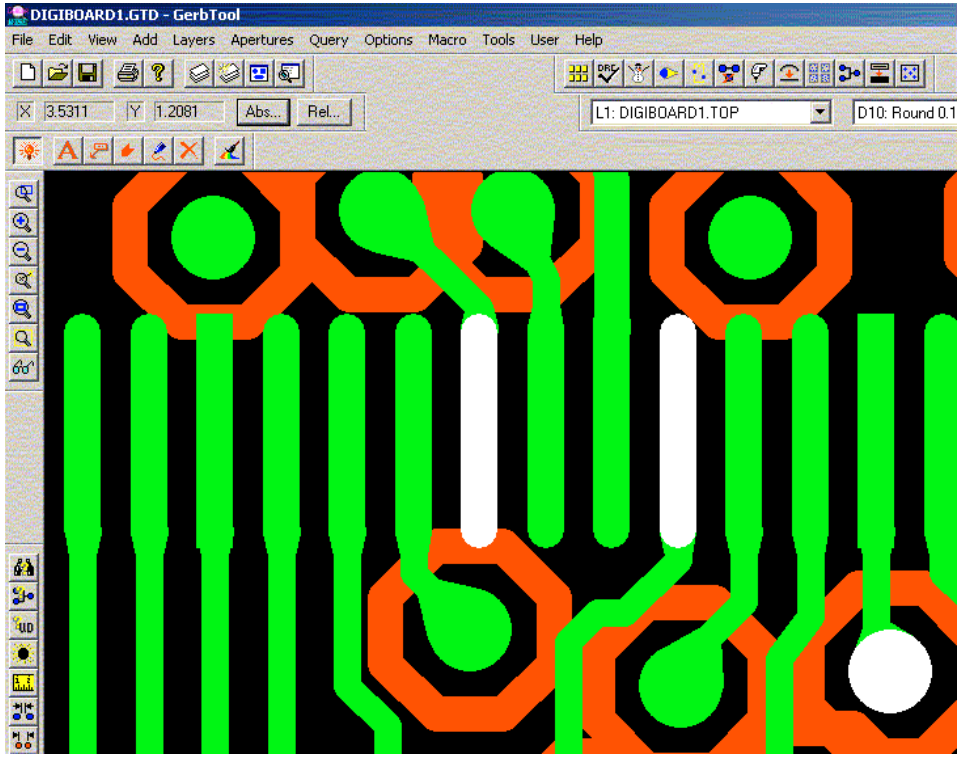
- Check with them which files and what format they need.
- The following is “extended GERBER” accepted by Sierra Proto Express.

Post Process

Plot output File Name	Batch Enabled	Device	Shift	Plot Title
*.TOP	Yes	EXTENDED GERBER	No shift	Top Layer
*.BOT	Yes	EXTENDED GERBER	No shift	Bottom Layer
*.GND	Yes	EXTENDED GERBER	No shift	Ground Layer
*.PWR	Yes	EXTENDED GERBER	No shift	Power Layer
*.IN1	No	EXTENDED GERBER	No shift	Inner Layer 1
*.IN2	No	EXTENDED GERBER	No shift	Inner Layer 2
*.IN3	No	EXTENDED GERBER	No shift	Inner Layer 3
*.IN4	No	EXTENDED GERBER	No shift	Inner Layer 4
*.IN5	No	EXTENDED GERBER	No shift	Inner Layer 5
*.IN6	No	EXTENDED GERBER	No shift	Inner Layer 6
*.IN7	No	EXTENDED GERBER	No shift	Inner Layer 7
*.IN8	No	EXTENDED GERBER	No shift	Inner Layer 8
*.IN9	No	EXTENDED GERBER	No shift	Inner Layer 9
*.I10	No	EXTENDED GERBER	No shift	Inner Layer 10
*.I11	No	EXTENDED GERBER	No shift	Inner Layer 11
*.I12	No	EXTENDED GERBER	No shift	Inner Layer 12
*.SMT	Yes	EXTENDED GERBER	No shift	Soldermask Top
*.SMB	Yes	EXTENDED GERBER	No shift	Soldermask Bottom
*.SPT	Yes	EXTENDED GERBER	No shift	Solder Paste Top
*.SPB	Yes	EXTENDED GERBER	No shift	Solder Paste Bottom
*.SST	Yes	EXTENDED GERBER	No shift	Silkscreen Top
*.SSB	Yes	EXTENDED GERBER	No shift	Silkscreen Bottom
*.AST	Yes	EXTENDED GERBER	No shift	Assembly Top
*.ASB	Yes	EXTENDED GERBER	No shift	Assembly Bottom
*.DRD	Yes	EXTENDED GERBER	No shift	Drill Drawing
*.FAB	Yes	EXTENDED GERBER	No shift	Fabrication Drawing

Optional “beautification” step

- GERBER files can be examined and fine-tuned using the GerbTool utility.
- Example: “teardropping” makes manufacturing easier.
- You need to be a GERBER expert to know what GerbTool is doing.
- I rarely perform such “beautifications” with my boards.

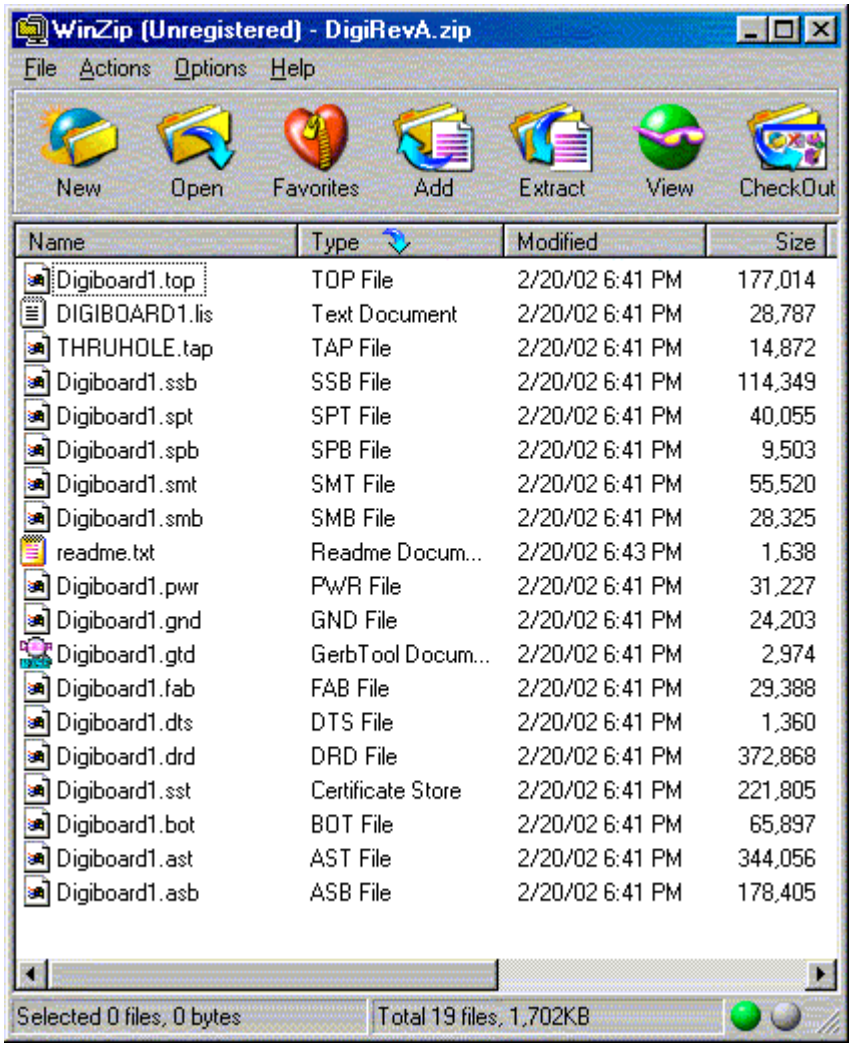


Layout step 12: send manufacturing files

- Pack the files into a ZIP archive.
- Include a README with your address and contact info.
- Explain relevant details in the README.
- Follow manufacturer's instructions.

Example README for my board

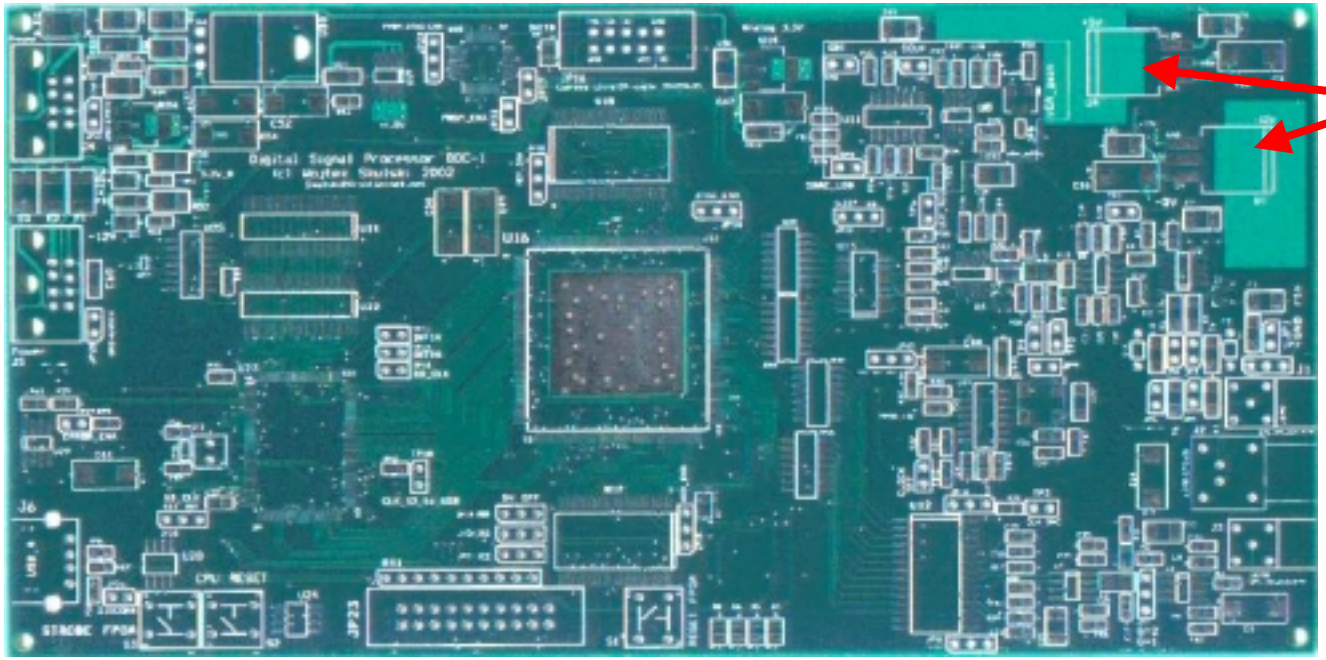
Board size: 4.0" x 8.0"
Board thickness: 0.062" FR4
4 metal layers, 1 oz copper
Minimum trace width 8 mils
Green soldermask
White silkscreen
Use your standard tolerances



After a week your boards should arrive...

- It is a terrific moment!

- Remember what I said: what you send is what you will get?
- Look at the arrows. I will have to remove the solder mask...



Two openings are missing in these spots

Some assembly required!

- Now it is time to solder!

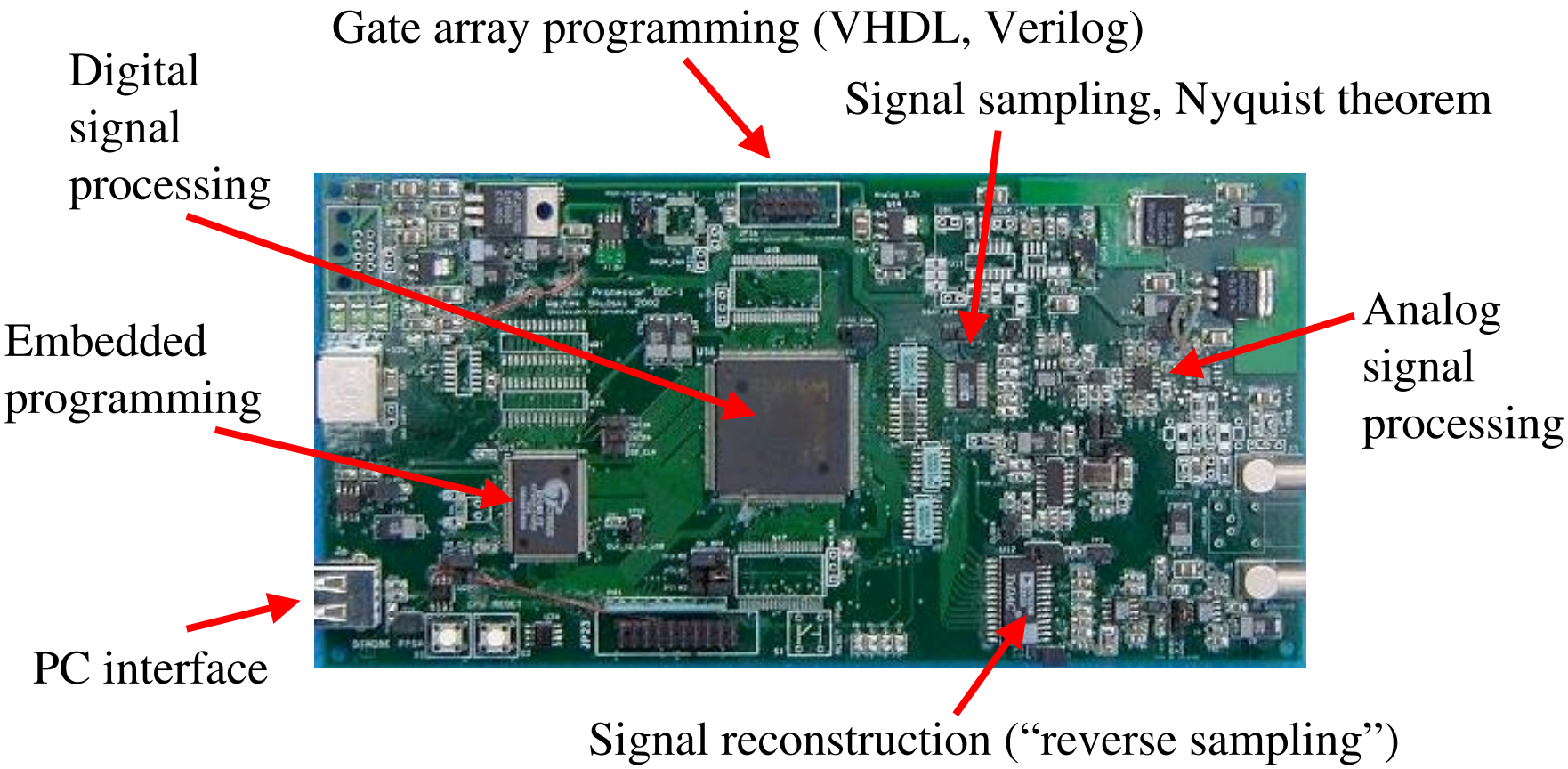
- Doing it yourself has one big advantage: you can test piecewise as you go.
- I assembled this board myself, including the FPGA in the middle.

Some good advice learned the hard way

- OrCAD is almost bulletproof. Save your work often.
- Read the manuals. OrCAD is complicated. Reading the help is not quite enough.
- Use the ohmmeter. If the resistance is zero, it is most certainly a short.
- If it is a short, do not apply power.
- Use safety goggles the first time you apply power, just in case.
- Apply power for a few seconds, switch off, touch the components. Are they hot?
- Do not be afraid of trying. If a component burns, it costs only a few bucks.
- Have fun.

More than just a board…

- A modern instrument involves many different topics.

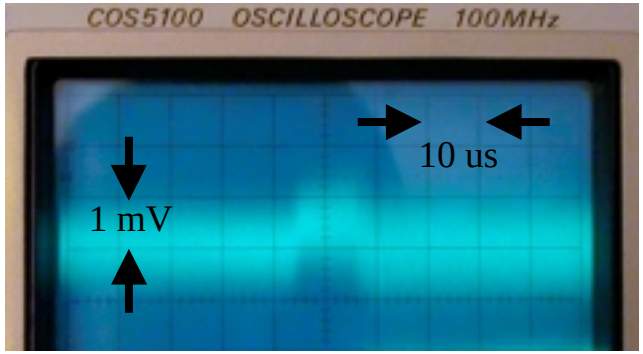


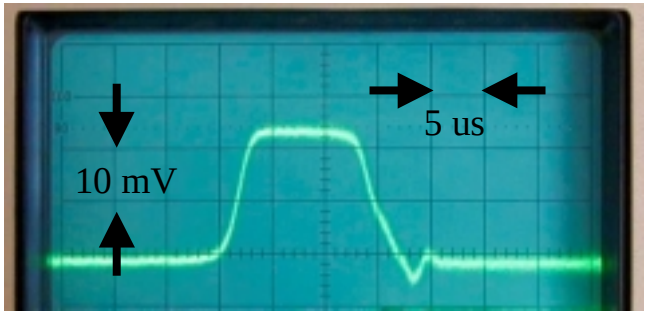
This can be your signal...

- Before filtering
- Signal buried in noise
- Signal/noise ratio = 1/1

Black magic

- After filtering
- Signal clearly visible
- Signal/noise ratio = 10/1





Summary of the class

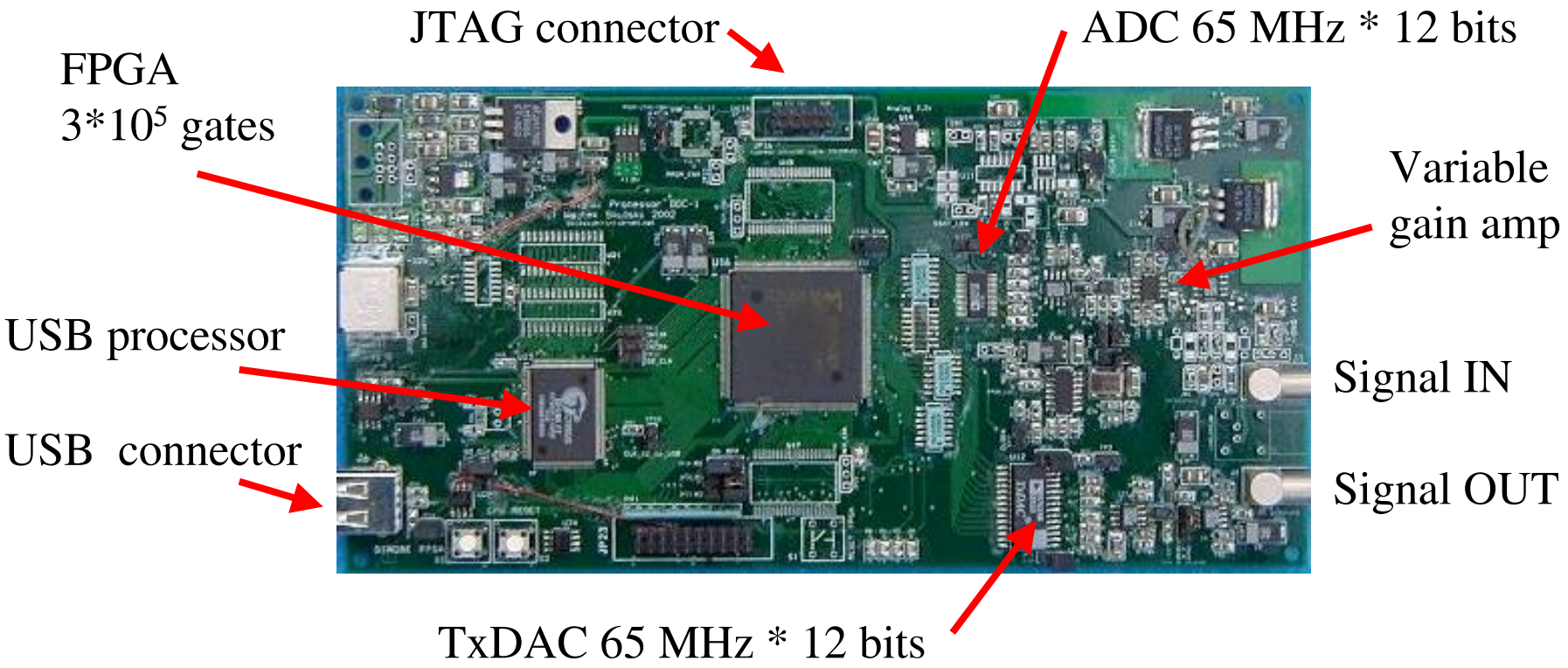
- Modern instruments are powerful. They look like black magic from outside.
- When zoomed in, there are many diverse (and interesting!) topics inside.
- Board design is an essential part of the whole art.
- Surface mount and multilayer boards are important for high performance design.
- OrCAD Capture and Layout provide the tools. Other similar software is readily available.
- To make sure it works, you can simulate signals using PSpice.
- Then you can send PCB files for production and you will get your boards.
- You can assemble your boards yourself (not very difficult), or have them assembled.
- Designing and assembling electronics is more fun than watching TV.
- Get ECAD software and a soldering iron and get started.
- Have fun with electronics.

Partial list of my boards

- Time Equalizer: 16-channel programmable delay, 10 picosecond time step.
  - On-line vertex selection for PHOBOS at RHIC (Brookhaven National Lab.)
- DDC-1: single-channel digital pulse processor, 12 bits @ 65 MSPS.
- DDC-8/NIM: 8-channel digital pulse processor, 10 bits @ 40 MSPS.
  - Work performed with DDC-1 and DDC-8:
    - Two APS presentations, one Master Thesis, one SBIR Phase I.
- DDC-8/VME: 8-channel digital pulse processor, 10 bits @ 40 MSPS.
  - Daughter card for a commercial VME motherboard.
- DDC-x/DM: a standalone x-channel digital pulse processor, 12 bits @ 65 MSPS.
  - Under construction (July/2004).
  - Prototype expected soon.
  - Will be used in Dark Matter Search experiment.

DDC-1: Digital Signal Processing Laboratory

- Low noise, pseudo-differential analog input.
- Digital signal processing in the FPGA.
- Output: both waveform capture via USB and direct digital synthesis.
- Applications: research instruments, student labs, DSP courses.



DDC-8/NIM: Universal Trigger Module